GIBBSCAM® 14

Version 14 : September 2020

SolidSurfacer

CAMBRIO

Proprietary Notice

Contents

Introduction

About this Guide

SolidSurfacer lets you define parts using solid and surface modeling techniques. This guide describes the GibbsCAM interface for solid and surface modeling, provides reference information about modeling, including machining solids, contour and rough machining, surfacing processes, advanced 3D machining, plunge rough processes, and more.

SolidSurfacer uses three primary methods to create machining parts. The first method is by creating solid models from part blueprints using the GibbsCAM solid modeling functions The many powerful modeling functions include: adding, subtracting and intersecting solids; automatic chamfering and filleting; and several methods for generating solid bodies from geometry.

Secondly, solid file formats generated by other CAD programs can be opened directly in GibbsCAM. Most formats can be directly opened or imported by the system (some formats require the purchase of an additional option).

The final method is by importing 3D surface files. The system recognizes and imports many surface entities. Once a surface file is brought into the system, it can be solidified, or it can be kept as a surface model, and machined.

Regardless of the method used to define the part, the final model can be machined using the 3D surfacing capabilities. The standard Roughing and Contouring functions can be applied to solid bodies and surfaces. The Surfacing function also provides several methods for generating 3-axis toolpaths to efficiently cut complex solid and surface part models.

Before using this guide, you should be familiar with the basics of the system outlined in the *Geometry Creation*, *Mill*, and *Advanced CS* guides. This guide assumes a level of proficiency with creating geometry, coordinate systems, and basic machining.

Definitions

The terms and definitions provided below are used to describe objects and elements used by the system, and throughout this guide. For more information, consult the Glossary.

Body
The term “body” is a generic term that refers to both solids and sheets. A solid body can be thought of as a bowling ball. A sheet body is more like a balloon with an infinitely thin wall.

Face
A face is one surface of a solid or sheet. A Sheet face includes the positive and negative sides. A Solid includes only the positive side. Faces are surfaces that have knowledge of the surfaces that surround them. For example, one side of a cube would be considered a face. Each face is bound by loops. A simple face is bounded by one loop.

Surface
A surface is either a face, a group of faces (depending on how the surface was created) of a solid, or the side of a sheet. Sheets have two surface sides and a solids have only one.

Solid
A solid is a body composed of faces and the area enclosed by the faces. Solids have volume. Solids bodies are used as the building blocks in creating part models in GibbsCAM. Unlike sheets, solids only have a positive side.

Sheet
A sheet is an surface with two sides, positive and negative. A sheet has no volume or thickness associated with it.

Edge
An edge is a curve/line between two faces. An edge of a solid must have exactly two faces connected to it. Note that more than two faces at an edge produces an invalid solid. The edge of a sheet can have a single face connected to that.

Loop
A loop is a series of connected edges that outline a face.

Vertex
A vertex is an endpoint of an edge.

Interface

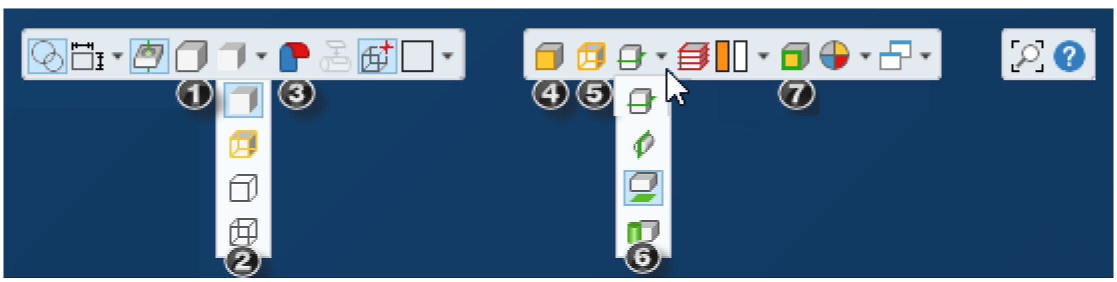
This section describes interface items that are specific to solids in the following topics:

- "Floating Toolbar" on page 11
- "Command Toolbar" on page 12
- "Bodies" on page 13
- "Solids Context Menus" on page 20
- "Machining" on page 29

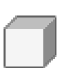
For more information on interface items, see the *Getting Started*, *Common Reference*, *Geometry Creation*, and *Mill* guides.

Floating Toolbar

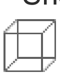
The Floating Toolbar contains seven items that are part of the SolidSurfacer interface. For more detailed information on these Toolbar items, see the Interface section in the *Common Reference* guide.



1. Show Solids
2. Render/Wireframe
3. Indicate Sheet Side
4. Face Selection
5. Edge Selection
6. Profiler
7. Pre-Selection Highlighting

 Show Solids

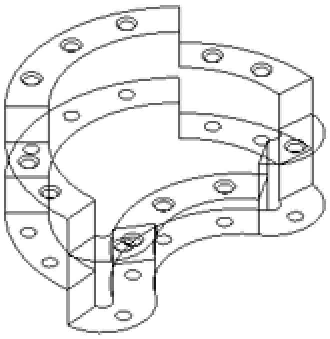
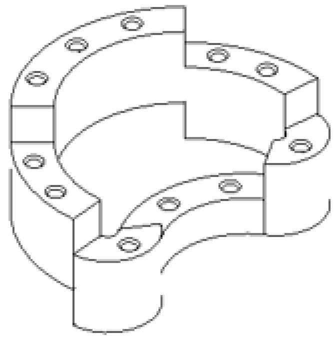
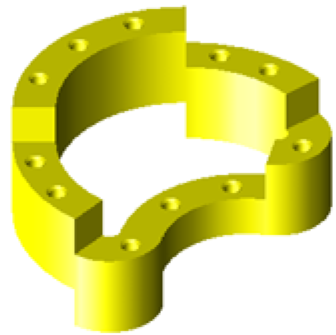
Show or hide all bodies, including sheets.

 Render/Wireframe

Click to toggle through four display appearances: edges of bodies, wireframe bodies, solids with edges, and rendering the solid.

Edges only	Wireframe	Solids with	Solid body

	bodies	edges	only

		
Edges only	Wireframe bodies	Solid body only

Indicate Sheet Side

Indicate the positive and negative sides of a sheet.

Face Selection

Enable or disable face selection mode.

Edge Selection

Enable or disable edge selection mode.

Profiler

Enable or disable the Profiler grid.

Pre-Selection Highlighting

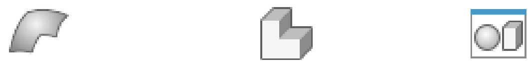
Pre-Selection Highlighting shows you which face your mouse is currently hovering over, making face and edge selection easier.

Command Toolbar

You use the Command Toolbar to access the Surface Modeling and Solid Modeling palettes. When you click a palette button, a dialog opens or processing occurs. For more information, see the *Getting Started* guide.

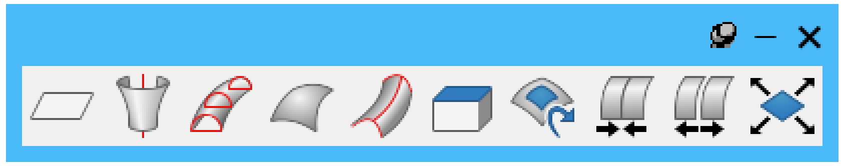
Toolbar Command groups

The Toolbar includes the Surface Modeling, Solid Modeling, and Body Bag buttons. You use the Surface Modeling and Solid Modeling palettes to create and modify bodies. You use the `Body Bag` to organize bodies.



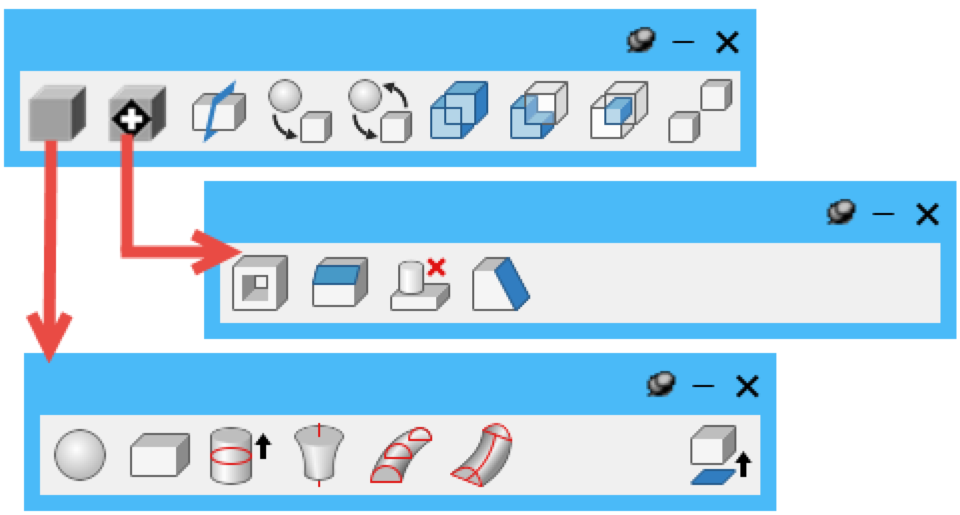
Surface Modeling Solid Modeling Body Bag

Modeling Palettes



You use the Surface Modeling palette to perform sheet, or surface, modeling. You can extract surfaces from faces or create surfaces from geometry. Booleans from Solid Modeling palette are also used in sheet modeling. For more information see “ Surface Modeling Palette” on page 37.

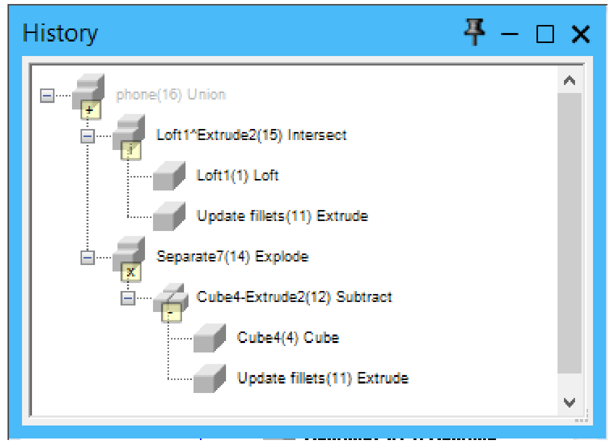
Solid modeling has three palettes, the main Solid Modeling palette and two sub-palettes. One for creating simple atomic bodies and another palette for more advanced modeling. For more information, see “ Solid Modeling Palette” on page 44.



Bodies

Each body (solid or sheet) contains a written history of how it was created and details about its physical and display properties. Bodies can be hidden and placed in a container called a Body Bag. They may even be rendered in wireframe and hidden.

History



You access the History list from the body context menu. See Body Context Menu. The History list displays the creation list for any selected body. All of the bodies and functions that were used to create the selected solid or sheet appear in the History list, even if they are no longer active bodies. The system maintains the history of all bodies that are created. Imported models do not have an existing history, they are essentially atomic bodies.

You can use the History list to access any body included the construction of the selected solid. You can access the history of a model to make changes to an earlier step in the modeling process, to easily incorporate changes to the final model without having to recreate the model from the beginning.

All non-atomic bodies have histories which contain information on the “parents.” “Parents” is the term used to describe the solids or sheets that were used to create the selected body. Solids may have one parent, as in cases of rounding or slicing, or two parents, as in the case of Boolean operations. Bodies removed from the Workspace as the result of a Boolean operation are maintained in the History list. Bodies contained in the History list are considered dormant bodies, while bodies in the workspace and Body Bag are active. Operations such as rounding or any of the Boolean functions can be performed on active bodies only.

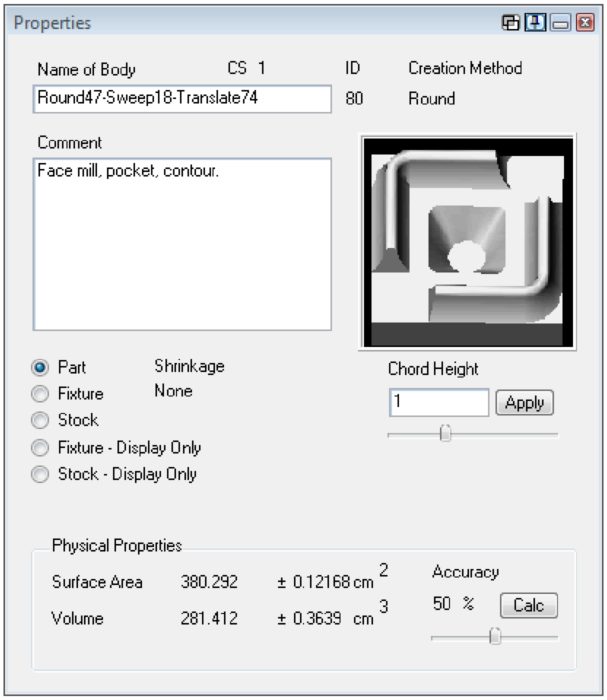
The History list is structured in a hierarchical format, where the selected body is at the top and all other bodies used to create it appear below in steps or branches. Double-clicking the icon next to the name of a body makes that body active and displays the body in the workspace. To make modifications to a body contained in the History list and incorporate those changes into the existing history, you must use Recreate. For information on Recreate and Rebuild, see Body Context Menu.

Properties

You access the Properties dialog from the body context menu. For more information, see “Body Context Menu” on page 21.

The Properties dialog contains items that apply to the selected body. You can change the name of the solid or sheet and enter a comment. The coordinate system that was last modified for the selected solid or sheet appears at the top of the dialog. The ID is a system-assigned positive integer that uniquely identifies each body. The Creation Method lists the action that was used to create the current body, such as Import, Sphere, Extrude, and so forth.

When the Properties dialog is open, you can select different bodies and the Properties dialog updates to reflect the body selected. You can select bodies in the History dialog to view in the Properties dialog by selecting the cube icon in the History list.



Part, Fixture, or Stock

Solids and sheets can be designated as Part, Fixture, or Stock. Additionally there are the Fixture - Display Only and Stock - Display Only options. When solids and sheets are created, they are designated as a Part by default unless the setting is changed in this dialog. Solids or sheets designated as a Fixture are rendered in red and are used as constraints when creating machining operations. Solids and sheets designated as Stock are rendered in dark blue and are used as the initial stock condition when creating machining operations.

Fixture-Display Only and Stock-Display Only display a body as a fixture or stock, and are used in rendering, but are not used in the toolpath generation calculation. When any stock or fixture bodies are present, the system may attempt to use 3D toolpath rather than 2D in some cases. The need to account for potentially hundreds of fixture bodies when generating toolpath can slow down system performance. Using Display Only stock and fixtures settings can greatly improve system performance, making this feature very important for TMS.

Chord Height

Chord Height sets the degree of render faceting for the selected solid or sheet. To change the chord height, enter a value and click the Apply button. This value only applies to the selected solid or sheet. For more information, see “Render Faceting” on page 28.

Physical Properties

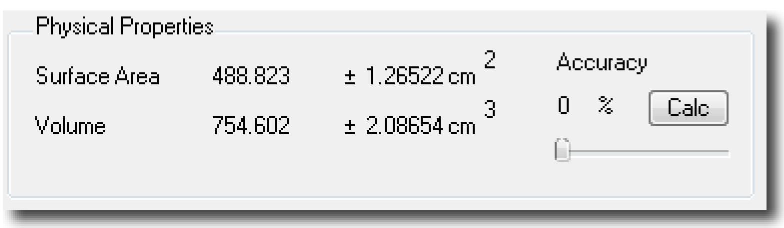
The Physical Properties section provides surface area calculations for solids and sheets, volume calculations for bodies, and surface periphery calculations for sheets. The Physical Properties section includes an Accuracy slide bar and a Calculate button. The Accuracy slide bar designates the amount of processing time and effort to allocate for the calculations. With the slide bar closer to the negative end of the spectrum, the calculations are less accurate and vice versa. Note that all calculations fall within a certain set range of accuracy, regardless of the setting of the accuracy slide bar.

The percentage values do not directly correlate to the calculation, in that a 0% accuracy still provides a reasonably accurate calculation. The Accuracy setting affects the calculation processing time. As bodies become more complex, the calculation time increases. In those cases, it may be desirable to designate a lower accuracy in order to speed up the process. The system always provides the +/- accuracy tolerance so that you can monitor the accuracy of calculations.

The following are conversion values for taking the volume in cubic inches, as shown in the Properties dialog, to measurements in ounces and liters.

1 cubic inch = 0.55409 oz.

1 oz. = 29.57353 ml



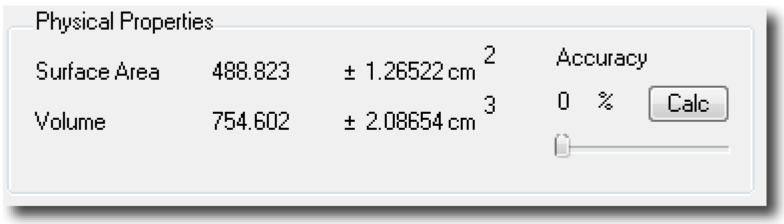
Solid Properties measurements

There is also a Physical Properties section which provides surface area calculations for solids and sheets, volume calculations for bodies, and surface periphery calculations for sheets. The Physical Properties section includes an Accuracy slide bar and a Calculate button. The Accuracy slide bar designates the amount of processing time and effort the system allocates for the calculations. With the slide bar closer to the negative end of the spectrum, the calculations are less accurate and vice versa. Note that all calculations fall within a certain set range of accuracy regardless of the setting of the accuracy slide bar.

The percentage values do not directly correlate to the calculation, in that a 0% accuracy is still going to provide a reasonably accurate calculation. The Accuracy setting affects the calculation processing time; as bodies become more complex, the calculation time increases. In those cases, it may be desirable to designate a lower accuracy in order to speed up the process. The system always provides the +/- accuracy tolerance so that you can determine whether the calculation is accurate enough for your purposes.

The following are conversion values for taking the volume in cubic inches (as is shown in the Properties dialog) to such measurements as ounces and liters.

1 cubic inch = 0.55409 oz.
1 oz. = 29.57353 ml

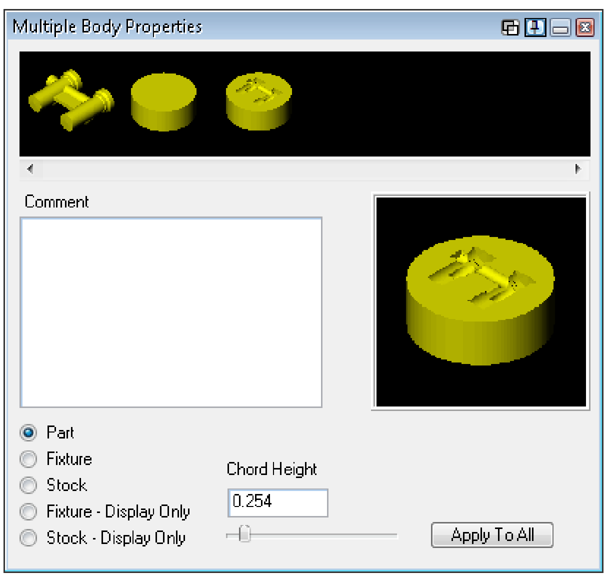


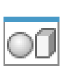
Solid Properties measurements

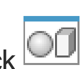
Multiple Body Properties

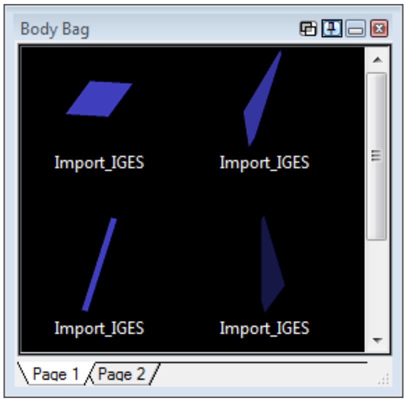
The Multiple Body Properties dialog appears when you select more than one body and then select the Properties command. You use this dialog to assign properties to many bodies at once.

Add or change bodies in the dialog by changing the selection. Quickly set all bodies in the dialog as a Part, Fixture, or Stock type. You can also set the Chord Height and add a Comment. Click Apply To All to apply the new settings to all the bodies in the dialog.



 Body Bag

To open the Body Bag window, from the Main palette, click  Body Bag. You use the Body Bag to organize the Workspace by storing bodies during part creation. `Double-click` a body to move it from the Workspace to the Body Bag. To move bodies between the Workspace and the Body Bag, you can also use the Bag It/Un-Bag It and Bag/Un-Bag Selected items. See the Body context menu and "Body Bag Context Menu" on page 25. Items in the Body Bag are active when the Body Bag is open. For example, you can select, modify, and machine items in the Body Bag. Bodies in the Body Bag appear as icons that you can select, move, and resize.



Resized Icons in the Body Bag

To arrange items in the Body Bag, you can drag the items. To display items in the Body Bag as small icons, large icons, tiles, or as a detailed list, you use the View items. See "Body Bag Context Menu" on page 25. You can also select the information you want to display for the items, such as Solid ID and Solid Creation Method.

About the Body Bag

Items displayed in the Body Bag are icons that include a snapshot of the body when it was moved into the Bag. Moving or resizing the icons does not effect the corresponding bodies. Icons are unaffected by Floating Toolbar buttons such as Show Solids, Render/Wireframe, Indicate Sheet Side, and color modes.

To move a body into the Body Bag:

Double-click the body in the Workspace or right-click the body and choose Bag It from the context menu. The object moves to a Body Bag page. The Body Bag page that the object moves to depends on whether the object previously resided in the Body Bag:

- If the object previously resided in the Body Bag, it moves to the Body Bag page that last contained it, and that page displays.
- If the object did not previously reside in the Body Bag, it moves to the Body Bag page most recently displayed.

To increase or decrease the size of icons in the Body Bag:

Click in the Body Bag and select `CTRL+mousewheel`. The size of icons increases or decreases on all Body Bag pages that are set to display bodies as Large Icons or Tiles. See "Viewing Body Bag Pages" on page 19.

To move an object from the Body Bag to the Workspace:

Double-click the object on the Body Bag page. The object displays in the Workspace.

Body Bag Color Display

Objects in the Body Bag display in the following colors:

Color	Body Type	Selected or Unselected
Gray	Solid	Unselected.
Light blue	Sheet	Unselected.
Dark blue	Stock	Unselected.
Red	Fixture	Unselected.
Red	Body in "Recreate" mode	Selected.
Yellow	Part (solid or sheet)	Selected.
Striped red/yellow	Fixture	Selected.
Striped gray/black	Stock	Selected.

Note: Regardless of the color preferences you set, objects in the Body Bag display in the colors listed in the table above.

Body Bag Pages

You can add pages to a Body Bag to organize and categorize items in the Body Bag. To access a page, click the tab for that page. To add, delete, and rename pages, you use the “Body Bag Page Context Menu” on page 26. To reorder pages, click and hold a page tab, then drag the tab to the new position.

You can create multiple pages within the Body Bag. Each page maintains its own view settings. When you create a new page, its view settings initially match those of the last-viewed page.

You can move objects from one Body Bag page to another by selecting and dragging to another tab; as the cursor passes over the tab, a preview of the page displays. You can create a new Body Bag page on the fly by dragging a Body Bag selection to an empty area to the right of the rightmost tab.

To insert, delete, or rename a Body Bag page, right-click the corresponding tab. A tab name in gray indicates an empty page. Only empty pages can be deleted.

You can select Clean Up Page from the context menu to perform a one-time Auto-Arrange on the current page without modifying view settings. You can select Clean Up Body Bag from the context menu to delete all empty pages and perform a Clean Up Page on all pages that remain.

Viewing Body Bag Pages

To display or modify the view settings of a page in the Body Bag:

1. Display the page
2. Right-click the Body Bag title bar, and select View.
3. Select the option you want. Refer to the table below to select the appearance you want.

Select	Result
Large Icons or Small Icons	Page displays each body's icon and name only.
Details or Tiles	Page displays each body's icon, name, type (Part, Stock, or Fixture), resolution (chord height), and CS (coordinate system).
Align to Grid	Icons or tiles are prevented from overlapping.
Auto-Arrange	Overlapping and empty slots are prevented. An icon or tile's location on the page depends on the Body Bag window size: Icons move to fill the top row from left to right, then the next row, if necessary, and so forth.

Selecting Body Bag Objects

Selection sets can include bodies in one, some, or all Body Bag pages, and can include or exclude Workspace items.

To select a body and deselect all others, click the body.

To add or remove a body in the selection set, `Ctrl+click` the body.

To select all objects in the Workspace and the currently displayed Body Bag page, from the Edit menu, click Select All.

• Edit > Select menu items (and Ctrl+A) operate on bodies in the Workspace and the currently displayed Body Bag page only.

• Edit > Deselect menu items and Edit > Invert Selection operate on all bodies, that is, the Workspace and all Body Bag pages combined.

To select or deselect objects in all pages, right-click the Body Bag title bar and use context menu items Select Body Bag or Deselect Body Bag.

To add all of a bodies on a page to a selection set, from the Body Bag context menu, click Select Page to specify the page. To remove all of a page's bodies from a selection set, use context menu Deselect Page to specify the page. You cannot select or deselect empty pages.

Context menu items such as Bag Selected or Show Properties of Selected or User Color of Selected operate on all objects in the selection set.

Solids Context Menus

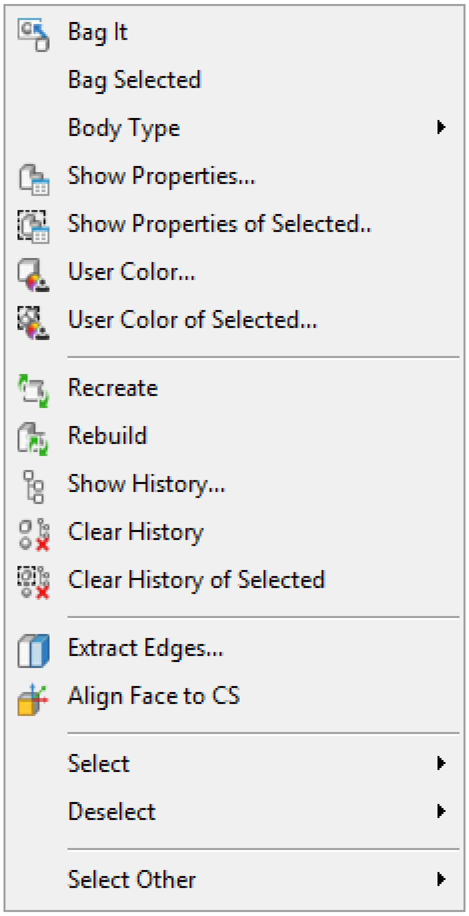
To access a context menu, right-click an object. To display a context menu for some dialogs, right-click the title bar. You can display context menus for the following items:

- Body Context Menu
- “Edge Context Menu” on page 24
- “History Context Menu” on page 25
- “Body Bag Context Menu” on page 25

- "Body Bag Page Context Menu" on page 26
- "Profiler Context Menu" on page 26

Body Context Menu

To access the body context menu, right-click on a body or history entry.



Bag It/Un-Bag It:

Bag It places a selected body into the Body Bag. If the body is in the Body Bag, Un-Bag It item places it in the workspace. This function does not apply to multiple selections. For multiple selection functions, see "Multiple Body Properties" on page 17.

Bag/Un-Bag Selected:

Moves all selected bodies into or out of the Body Bag.

Body Type

Designate the selected solid as Part, Stock, Fixture, Stock - Display Only, Fixture - Display Only. For more detail on these selections, see Properties.

Show Properties:

Opens the Properties dialog for a solid or sheet. For more information, see " Properties" on page 14.

Show Properties of Selected:

Shows the properties of all currently selected bodies. See " Properties" on page 14 and "Multiple Body Properties" on page 17.

User Color:

Enables custom color display of individual edges or faces. See " Properties" on page 14 and "Multiple Body Properties" on page 17.

User color of Selected:

Change the color of all currently selected faces or bodies. See " Properties" on page 14 and "Multiple Body Properties" on page 17.

Recreate:

The Recreate mode takes the selected body back to its creation action to be modified. The selected body is drawn in red and any changes made will permanently replace the selected body. To cancel Recreate mode, `right-click` a body and choose Exit Recreate or click the red body.

Rebuild:

Reprocesses the History list and incorporates any changes made using Recreate, Swap, or Replace into a new final part model. The Rebuild function is limited in that models cannot be rebuilt if the changes require a significant alteration to the topology. For example, if the change created any new edges, the final model cannot be rebuilt.

Show History:

The History list displays the creation process of the selected body. All bodies used to create the selected body appear in the History list. To restore a dormant body in the History list to the Workspace, double-click the icon in the History list.

Clear History:

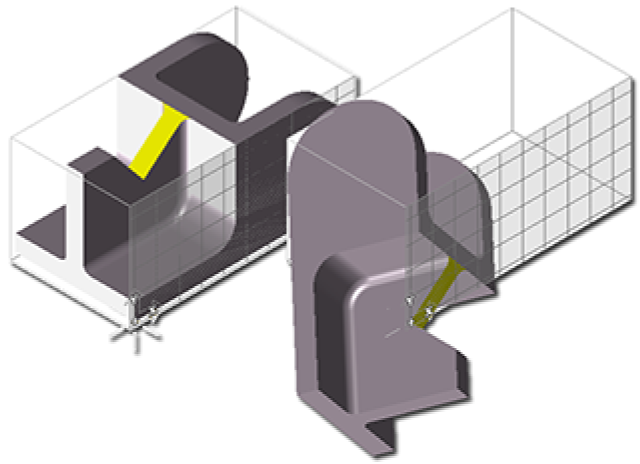
Clears the history of the selected body, essentially turning the solid into an atomic body. You cannot undo this action.

Extract Edges:

Extracts the selected edges and creates geometry from them. Please note this may take some time depending on the amount of geometry selected.

Align Face To CS:

When Face Selection is active, you can choose a face and align it to the current CS. Right-click a face to select this command. Choosing this command orients the part to the CS as if the following steps were taken.



a. Create a new CS from the target CS, that is, the CS you want to align to.

b. Select a planar, cylindrical, or complex face.

c. Select Align Plane Through & Move (right mouse menu choice) or `Alt`-click the Align CS button. For a cylinder, use Align CS Normal & Move.

d. Apply the Modify CS (XYZ) command to the solid to assign it to the new CS.

e. Select the target CS.

f. Apply the Modify CS (HVD) command to the solid to assign it to the target CS and move it.

g. Delete new CS.

Face Selection Mode Options

The following items for selecting and deselecting faces are only available when the system is in Face Selection mode. These options are useful when multiple faces must be selected for modeling or machining functions, and frees you from having to select one face at a time.

Tangent Faces:

Selects or deselects the target face and all of the faces reachable by a tangency.

Faces Above:

Select/deselect neighboring faces if they have an upper boundary that lies above the upper boundary of the target face. Next it will branch out to the adjoining faces of the neighboring faces and repeat the selection/deselection using the neighbors upper boundary as the condition (rather than the target face). A special condition exists for flat faces that neighbor the target face. They are selected/deselected based on the lower boundary of the target face.

Faces Below:

Select/deselected neighboring faces if they have an lower boundary that lies below the lower boundary of the target face. Then, it will use the adjoining faces boundary and repeat the selection/deselection. However, adjoining flat faces are selected/deselected based on the upper boundary of the target face.

Floor Faces:

Select/seselect all floor faces connected to the target face. A floor face is approximately normal to the depth axis of the current CS; the approximation is set by the Floor/Wall Angle Tolerance value set in File > Preferences > Interface > Selection.

Wall Faces:

Select or deselect the target face and any connected face that is parallel to the depth of the current CS. Angled walls that fall within the Floor/Wall Angle Tolerance value set in File > Preferences > Interface > Selection are also selected.

3D Faces:

Select/deselect faces connected to the target face that are not defined as a floor or wall. Next it will branch out to the adjoining faces and select/deselect them using the same logic.

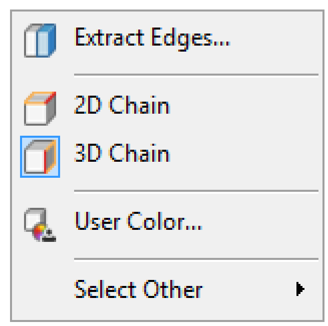
Transition Faces:

Select/deselect all transition faces connected to the target face. A transition face is a smooth blend that is connected to a wall and floor face.

Fillets:

Select/deselect all constant radius fillet faces that are connected to the target face. The target face is also selected. The system only selects fillets that have the same constant radius as the target face if the target face is a fillet.

Edge Context Menu



Right-click a selected edge to access options that affect edge selection. When you double-click an edge, the system tries to build a closed loop of edges starting with the selected edge. The 2D Chain and 3D Chain options affect how the system chooses the next edge to be connected at each vertex.

2D Chain:

When you select this option, double-clicking an edge attempts to select a loop of edges that are planar to the current CS (or those closest to it), resulting in a 2D loop. If there is more than one possible choice for an edge at a vertex, the system chooses the one that is closest to the same direction.

3D Chain:

When you select this option, double-clicking an edge attempts to select a loop of edges that are normal to the current CS (or those closest to it), resulting in a 3D loop.

User Color:

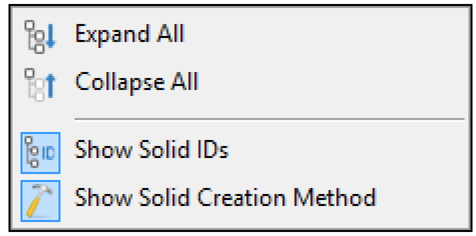
This allows you to assign colors to individual edges.

Select Other:

This displays a list of edges that intersect with the selected. Scroll list to choose. You can also choose the entire solid.

History Context Menu

To access the History context menu, right-click the title bar of the History list. The History context menu contains the following items:



Expand All:
Expands the tree, displaying all branches which contain the bodies used to create the selected model.

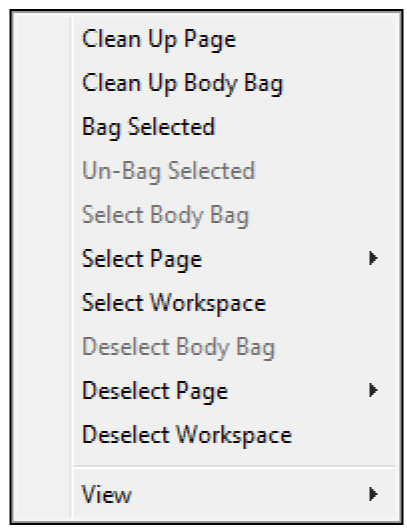
Collapse All:
Shows only the selected model icon at the top of the History list and hides the branches.

Show Solid IDs:
The Solid ID is a system-assigned positive integer that uniquely identifies each body.

Show Solid Creation Method:
The Solid Creation Method lists the action that was used to create the current body, such as Import, Sphere, Extrude, and so forth.

Body Bag Context Menu

To access the Body bag context menu, right-click anywhere in the Body bag title area. The Body Bag context menu contains the following items.



Clean Up Page:
Arranges the Body Bag icons on the selected Page so that you can view all icons and none overlap.

Clean Up Body Bag:
Reorganizes the Body Bag icons so that you can view all icons and none overlap.

Bag Selected:
Places any solids or sheets that are selected in the drawing window into the Body Bag.

Un-Bag Selected:
Take any selected Body Bag icons and place the solids/sheets back into the drawing window from the Body Bag.

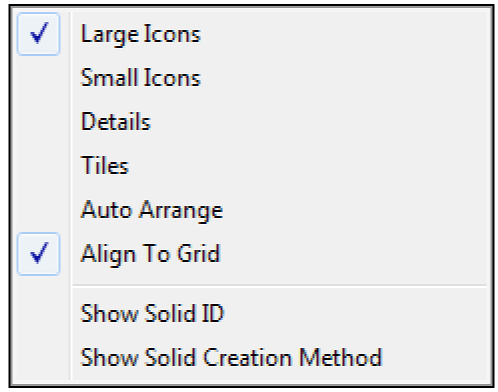
Select/Deselect Body Bag:
Selects or deselects all of the bodies in the Body Bag. You can use this item to isolate problem areas by analyzing surface files.

Select/Deselect Page:
Selects or deselects all the of the bodies on the Page.

Select/Deselect Workspace:
Selects or deselects all entities (including bodies and geometry) in the Workspace. You can use this item to isolate problem areas by analyzing surface files.

View:

Click View to display the following items.



Large Icons:

Display large Body Bag icons.

Small Icons:

Display large Body Bag icons.

Detail:

Display detail list of Body Bag icons.

Tiles:

Arrange Body Bag icons as tiles with Solid or Sheet type, creation method, solid ID, chord height and current CS.

Auto Arrange:

Automatically arrange Body Bag icons so that you can view all icons and none overlap.

Align to Grid:

Align Body Bag icons to a grid.

Show Solid ID:

Show solid ID for Body Bag icons.

Show Solid Creation Method:

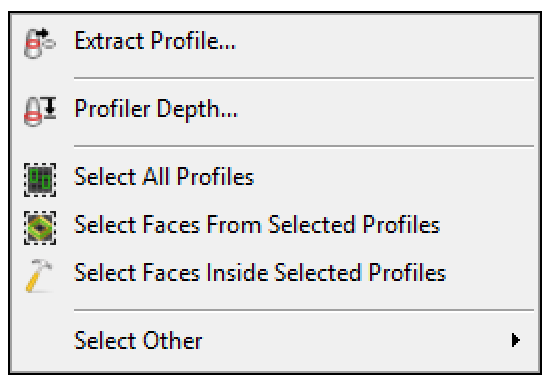
Show solid creation method for Body Bag icons.

Body Bag Page Context Menu

To access the Body Bag Page context menu, right-click a Page tab at the bottom of the Body Bag window. See “ Body Bag” on page 17. With this menu you can insert delete or rename pages.

Profiler Context Menu

To access the Profiler context menu, right-click on any visible part of the Profiler grid.

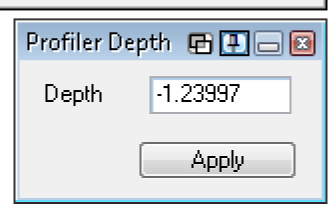


Extract Profile:

With the Profiler on, the Profiler grid visible, and one or more bodies selected, click Extract Profile. The Geometry Extraction dialog appears. Type a value for Tolerance and then click Do It. The profile is extracted as geometry.

Profiler Depth:

Select this option to access the Profiler Depth dialog. The Depth field shows the grid’s absolute depth. Drag the grid while this dialog is open and the field updates to reflect the current depth. To specify a new depth, enter the value in the field and click Apply.



Select All Profiles:

Selects all profiles generated by the Profiler grid. Selected profiles appear blue.

Select Faces From Selected Profiles:

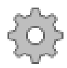
Selects all faces that are tangent to the profiles generated by the Profiler grid. When a profile has been selected for machining, you can only select the faces that are tangent to the toolpath, or the portion of the profile between the start and end machining markers.

Select Faces Inside Selected Profiles:

Selects all faces that are contained fully within the boundary of the selected profile and that are not tangent to the profile. Any face within the boundary is selected, regardless of its depth.

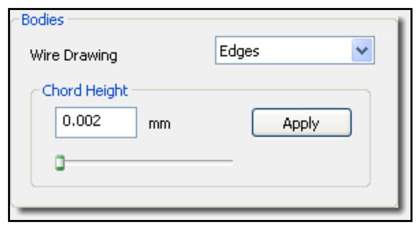
Preferences

To access GibbsCAM preferences:

From the File menu, select  Preferences.

This topic describes items on the Display tab that affect the graphic display of solids and sheets. To view display preferences, click the Display tab. You can adjust the degree of faceting during render operations. See “Render Faceting” on page 28.

You can display bodies as rendered solid objects or as wireframe drawings. The Render/Wireframe button in the Floating toolbar determines whether solids and sheets are rendered as objects or wireframe drawings. The Wire Drawing setting defines whether it displays edges or facets of bodies. For more information about Preferences, see the *Common Reference* guide.

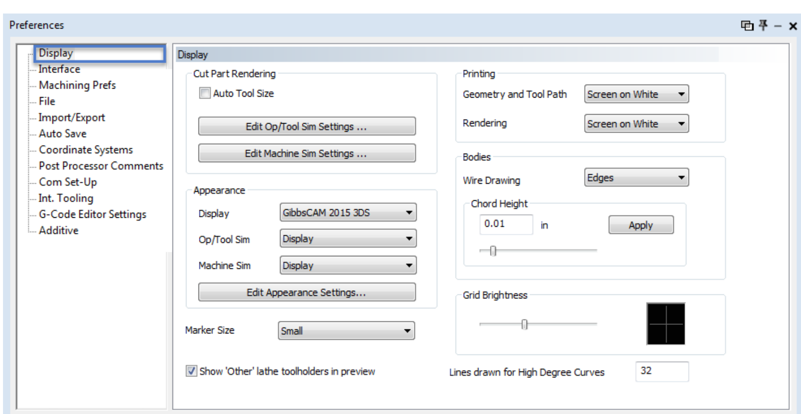


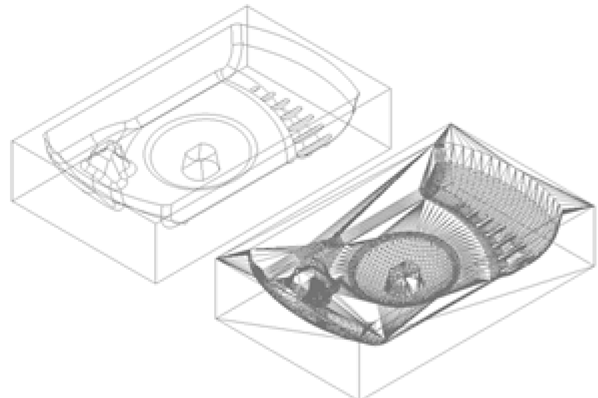
Display Tab

To access GibbsCAM preferences:

From the File menu, select  Preferences.

This topic describes items on the Display tab that affect the graphic display of solids and sheets. For more information about Preferences, see the *Common Reference* guide.





Wire Drawing:

Solids and sheets can be displayed as rendered solid objects or as wireframe drawings. The Render/Wireframe button in the Floating Toolbar determines whether solids and sheets will be rendered objects or wireframe drawings. This setting lets you determine whether the system displays edges or facets of solids or sheets depending on the selection made in the Wire Drawing section.

Chord Height:

Enter the overall part chord height. The chord height determines the faceting resolution when solids and sheets are rendered. Click the Apply button to finalize the change to the faceting tolerance for selected bodies, as well as setting the value for new bodies to be created in the future. For more information on setting the Chord Height, see “Render Faceting” on page 28.

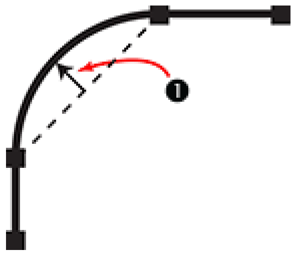
Render Faceting

Rendering is the process of displaying a picture of a model on the screen. When bodies are rendered, they are faceted. Facets are small planar surfaces that compose the rendered model. The more facets drawn, the closer the model resembles the actual mathematical model and the more time it takes for the system to render the model. Faceting affects the quality of the rendered bodies. It also affects overall system performance and speed. The faceting chord height should be set at a value that balances the quality of the model with system performance.

Note: The faceting tolerance does not affect on machining tolerances, only on the rendered image on the screen.

The tolerance used for surface machining is set locally in the Process dialogs Solids tab > Advanced Settings dialog and is labeled as the Cutting Tolerance and globally in the Document Control dialog as Use Global settings for Solids > Part Rough Tolerance. It is this specification which designates how closely the toolpath will follow the surface.

The number of facets used to render a model is determined by the chord height. A chord is a straight line that joins any two points on an arc or circle. The chord height is the distance from the chord to the arc or circle (see figure below). The smaller the chord height, the closer the facet will be to the arc or circle, resulting in a better rendered image of the solid or sheet (this is a 2D description of chord height; the system uses a 3D chord height for the faceting of solids and sheets, but the general idea is the same).



1. Chord Height

Chord Height

The system uses a global faceting chord height which is applied to the entire part model. The global chord height is applied to all solids and sheets that are created or imported.

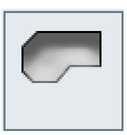
To set the global chord height:

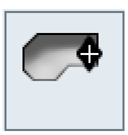
1. Click File > Preferences. The Preferences dialog appears.
2. Click the Display tab.
3. Under Chord Height, type a number in the text box or drag the slider to change the value.

You can set a different faceting chord height to individual solids and sheets. The Properties dialog, accessed by right-clicking on a solid or sheet, contains a chord height value which will only be used to facet the selected solid or sheet. For more information, see “ Properties” on page 14.

Machining

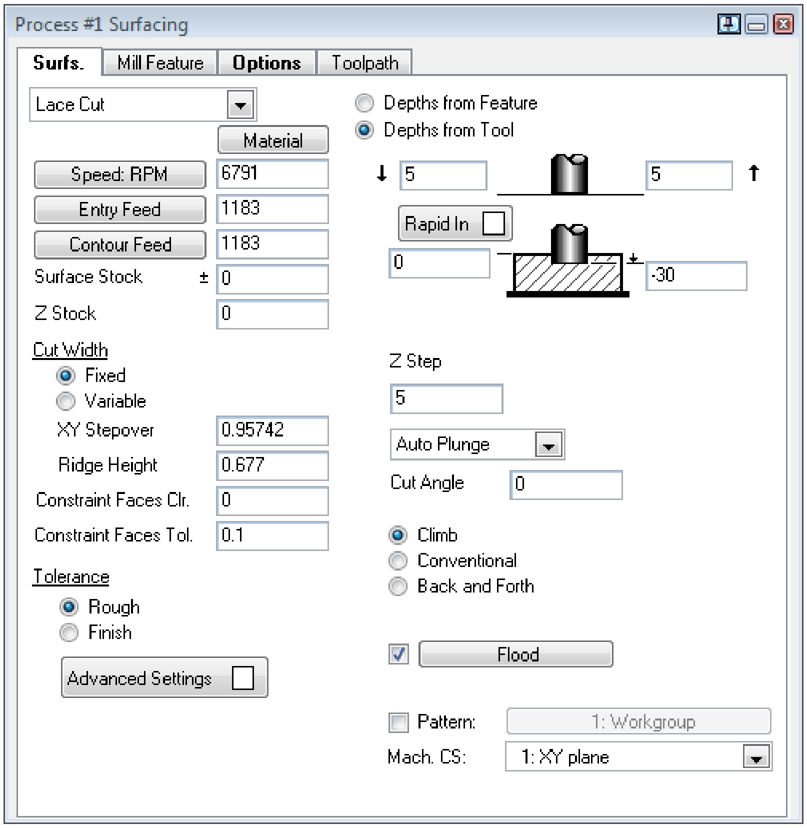
SolidSurfacer provides two processes for machining solids, Surfacing and Advanced 3D.





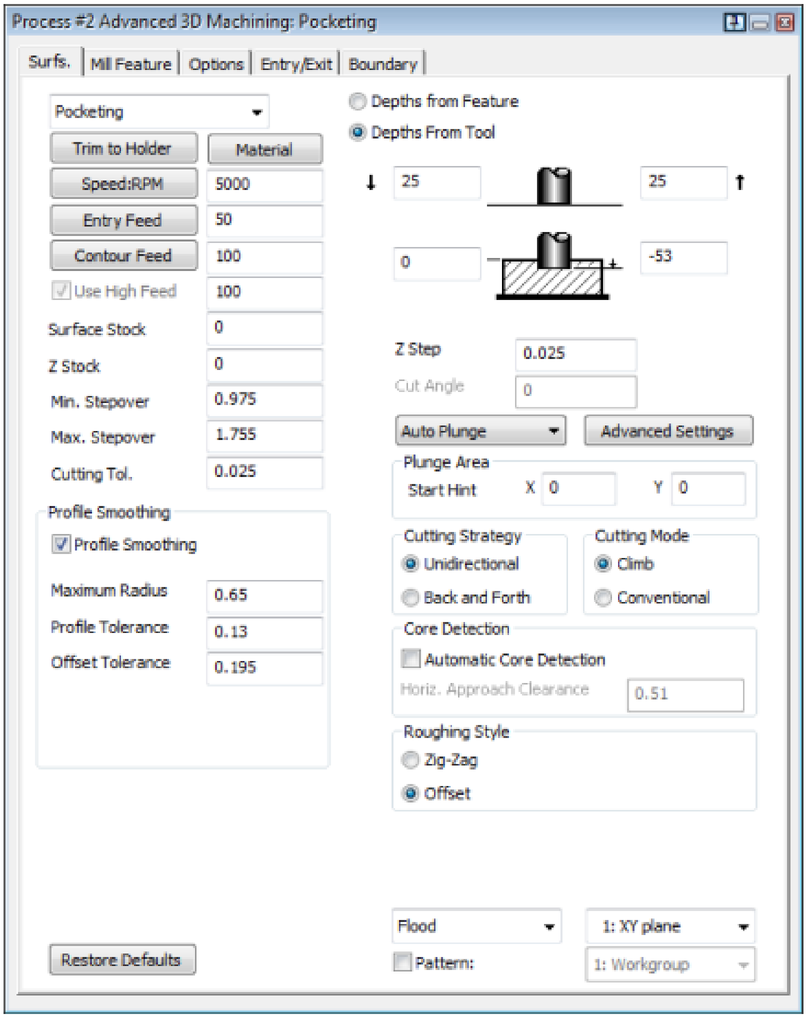
Surfacing:

The Surfacing process creates 3D toolpath from solids and surfaces. For more information, see “ Surfacing Process” on page 93. The Surfacing process creates full 3 Axis toolpaths. To create a Surfacing process, drag the Surfacing tile and a tool tile to a process tile. You can select flat, bullnose, and ball endmill tools for surfacing operations. You must select solids or sheets as the cut shape for surfacing processes. The Surfacing dialog contains three tabs, Surfs., Options, and Toolpath. The options in these dialogs vary, depending on the type of toolpath you select for the process. The six surfacing options are Lace Cut, 2 Curve Flow, Surface Flow Cut, Intersection-Edges, Intersections-Faces, and Intersections-Auto. For dialog items not mentioned here, see the *Mill* and *Advanced CS* guides.



Advanced 3D:

This is another process that creates 3D toolpath from solids and surfaces. Additionally, Advanced 3D creates H.S.M. (High Speed Machining) compatible toolpath. For more information see “Surfacing Process” on page 93.

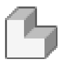


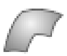
Introduction to Modeling

About Modeling

Modeling is the process of defining a part’s shape and dimensions on a computer. Common types of modeling include geometric modeling (both 2D and 3D), solid modeling, and surface modeling.

 Geometric modeling is the process of defining a model with simple geometric constructions such as points, lines, circles and splines. Geometry can be defined in either two- or three-dimensional space.

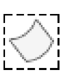
 Solid modeling is the process of defining a part as a solid object rather than as geometry or a collection of sheets. The process begins with the creation of a simple solid known as an atomic or primitive body. Boolean operations can then be performed on an atomic body to create a new, distinct body. Advanced techniques such as shelling, offsetting, blending, stitching/unstitching, and drafting may then be used to create the final part model.

 Surface modeling is the process of creating sheets as the foundation for a model. Boolean operations can also be performed on sheets. The surface creation tools are primarily intended to be used with surface files that are imported rather than for creating complete part models using surface modeling techniques. The use of solid modeling tools is the recommended method for modeling a part.

Solids

A solid is an object that has volume. Solids can either be single bodies (lumps) or a collection of bodies (multi-lump body). You can view solids as wireframe drawings or rendered objects. However, you can only select rendered objects to perform modeling functions.

Solids are rendered in grey; however, selecting a solid changes it to a yellow body. Solids may also be defined as stock (blue) or a fixture (red). Selecting a body will show if it is composed of lumps or if it is a multi-lump body. Models containing lumps or multi-lump solids can be separated to create multiple bodies.

 Sheets

Sheets do not have any thickness or volume. A sheet has knowledge of the neighboring sheets that surround it. Sheets may have either one face or several faces. Similar to solids, sheets may also be defined as a part, stock, or fixture.

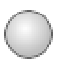
When surface files are imported, each surface entity is brought in as a single sheet (unless the Solidify option is chosen). These sheets can either be machined directly or modified as necessary to complete the part model.

Primitives/Atomic Solids

A primitive or atomic solid is a simple, non-divisible solid—a solid that is not built from other solids. Atomic solids do not have a history (or branches). All solid models originate from at least one atomic solid. Examples of an atomic solid include a sphere, a cube and a revolved/lofted/swept/extruded 2D shape.

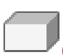
Atomic solids are created using the options in the Create Solid palette which is accessed by clicking on the Create Solid button in the Solid Modeling palette. The solidifying functions, which convert sheets to solids, are also contained in the Create Solid palette. Solidified sheets are considered atomic solids as they have no history associated with them.

1. Sphere
2. Cuboid
3. Extrude
4. Revolve
5. Loft
6. Sweep
7. Solidify



Sphere

A ball shaped solid.



Cuboid

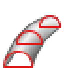
Any type of rectangular solid.

Extrude

A two dimensional closed shape extruded along the depth axis.

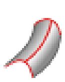
Revolve

A 2D shape, either open or closed, revolved around either the horizontal or vertical axis by a specified number of degrees.



Loft

Lofted solids are created by selecting a series of closed shapes that will be blended together using selected alignment points. Lofting is also referred to as skinning or blending.



Sweep

Swept solids are created by selecting base curve geometry and drive curve geometry. The base curve acts as the spine which determines the overall contour of the sweep and the drive curve(s) designate the location and shape of the solid.

Solidify

There are several options for solidifying sheets into solids. They include: Cap, Extrude, Offset and Solidify Closed Sheets.

Workspace

Active solids and sheets exist in the Workspace. The Workspace consists of the drawing window and the Body Bag (when visible). Bodies must be active in order to perform any modeling functions such as Boolean operations. Bodies that only exist in the History list are considered dormant bodies. Because the solid modeler creates single bodies from any operation by default, the component bodies are removed and made dormant to simplify the modeling process and control the file size. Dormant bodies can be retrieved using the History list.

Workgroups and Coordinate Systems

Bodies are not contained in workgroups. They are drawn in either the Body Bag or the Workspace, regardless of the current workgroup.

Bodies are assigned a coordinate system (CS) which is based on the current CS when the body was created. Some of the modeling functions (such as extrusions and revolved bodies) are CS-specific, meaning that the current coordinate system is used to create the body. Other modeling functions (such as lofting) are not dependent on the current CS.

Coordinate Systems can be modified into the proper orientation by selecting components of a solid or sheet. There are two categories of CS alignment, plane through geometry groups and plane normal geometry groups. Planes can either be aligned through selected geometry or normal (perpendicular) to selected geometry. With solids and sheets, the plane through geometry groups includes planar edges and planar faces, while the plane normal geometry groups includes an edge and a point, and a face (planar or non-planar) and a point. For more information, see the *Advanced CS* guide.

Boolean Operations

Boolean operations use two or more bodies (either solids or sheets or some combination of the two) to create a new single solid or sheet. The Boolean operations contained in the system are replace, swap, addition, subtraction, intersection and separation. The order of selection is important as the functions are not done simultaneously. The boolean process operates on the 1st and 2nd items. That result is then used in the boolean operation with the 3rd item, then the 4th and so on. You can use Add and Subtract on multi-lump bodies, but the Intersect function requires that all selected bodies are intersected at some common point.

Boolean operations are destructive, in that the initial bodies selected for the Boolean operation are deleted, and only the resulting body remains active in the Workspace. The deleted bodies used in the Boolean operation become dormant bodies and can be retrieved from the History list. Non-destructive Booleans can be performed by holding down the `Alt` key. Non-destructive Booleans generate the new body and place the original two bodies used in the Boolean operation in the `Body Bag`.

1. Replace
2. Swap
3. Addition (Union)
4. Subtraction (Difference)
5. Intersection
6. Separate

Replace:

Replaces one body with another. The first body selected completely replaces the other body.

Swap:

Switches the two selected bodies.

Addition (Union):

Combine the volume of bodies. The resulting body is a new single body composed of the selected bodies. If the bodies you select are disjunct (not touching), the resulting body is a multi-lump body.

Subtraction (Difference):

Removes the volume of one or more bodies from another body. Select the bodies and subtract them. The resulting body is the first body minus the second body. The first body selected is kept and the second body selected is subtracted out. When you select more than two bodies, the subtraction continues in the order of selection.

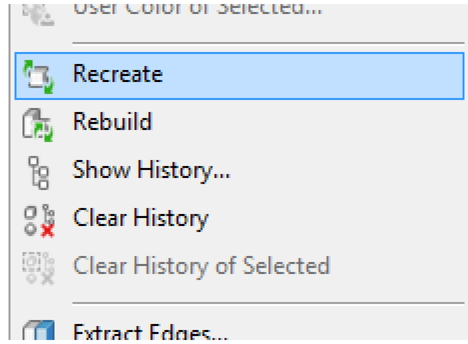
Intersection:

Keeps the common volume between bodies. Select two or more bodies and intersect them. The resulting body is the volume that is common to the bodies selected. The bodies must all have a common overlapping area.

Separate:

Splits a multi-lump body into individual bodies.

Recreate Mode



Rebuilding a solid incorporates any changes made to bodies that were used to create that solid. These bodies are referred to as parent bodies. Recreate allows the user to change a parent body, and must be used for changes that the user wishes to incorporate into the rebuilding of a solid.

Bodies can be modified using the Recreate function of the software, which is contained in the context menu (accessed by a `right-clicking` the body). When Recreate is selected, the system goes into Recreate mode.

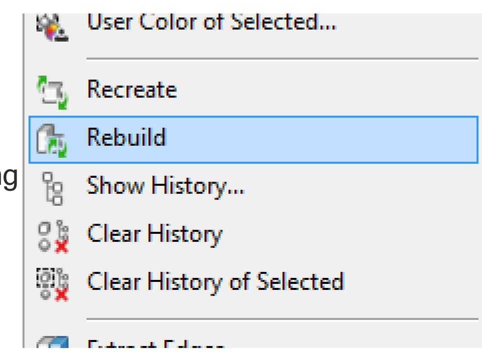
When in Recreate mode, the body selected to be recreated is drawn in red and placed in the Body Bag. Exit Recreate mode by selecting the body drawn in red or selecting the Exit Recreate item from the body context menu. If applicable, the parent body or bodies that were used to create the body to be recreated become active and appear in the drawing window. The function button (or buttons) originally used to create the body will be outlined in red. All original data will be restored.

The following are some examples of what can be done using the Recreate function:

- Change the data for an atomic body (for example, the radius of a sphere).
- Recreate an atomic body from different or modified geometry. The geometry should be changed prior to entering Recreate mode.
- Change the type of atomic body that was created (for example, a sphere can be made into a cuboid).
- Change the selected parents used in a Boolean operation.
- Change the type of Boolean operation that was performed.

Recreate only affects the creation of the selected body. To recreate its parents, they must be brought back from the History list.

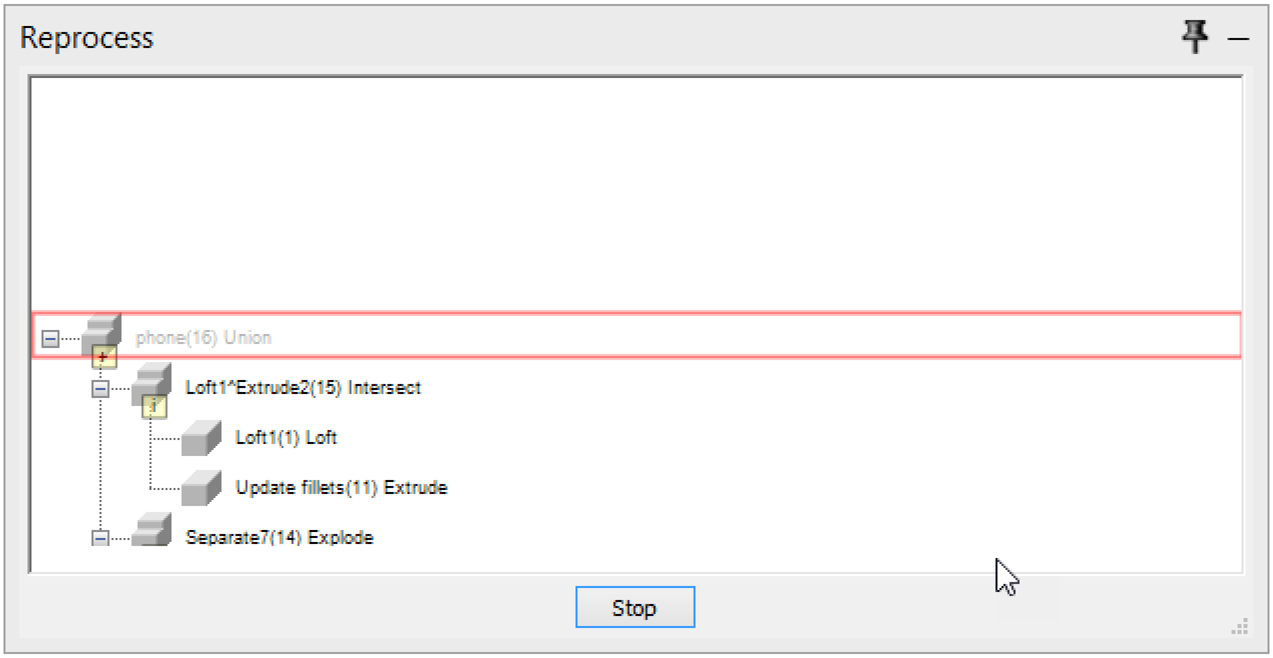
Rebuilding Solids



Because the system maintains the history of solids and sheets, it is possible to modify earlier steps during model creation and rebuild the final model. Any changes made can be incorporated into the final model using the Rebuild function. This level of associativity is very useful when working with complex models that would take considerable time to rebuild from scratch if changes were necessary.

To rebuild a model, follow the steps outlined below.

1. Click on a body from the History list.
2. Make modifications to that body using the Recreate function.
3. Rebuild the body by selecting Solids > Rebuild.



The Rebuild function reprocesses the History list. Any changes made to a solid or sheet in the tree will be re-incorporated into the model. The only way to make a modification to an existing body without changing the name and reference of that body is to use the Recreate function. The Rebuild function cannot be undone, which means that once a model has been rebuilt, it cannot be changed back to its previous state. You may reopen the model to retrieve the previous history if needed.

Rebuild is useful when working with imported solid models. As an example, let's say you start with an imported Parasolids file. You build a mold base body from this model. Changes are made to the original model provided. The new model is imported into the system. To incorporate the new changes, you would recreate the mold base by selecting the new body instead of the old one and performing the Boolean operation. Then select the final body and rebuild it.

Modeling Reference

This section describes the GibbsCAM modeling capabilities. For exercises that provide a practical application of modeling functions, see the *Tutorials*. We recommend starting with the modeling exercises and then referring to the reference information as necessary.

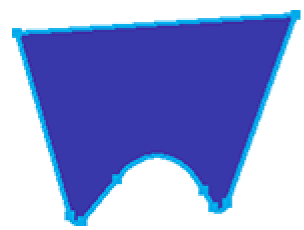
Surface Modeling Palette

Clicking the Surface Modeling button in the Command Toolbar opens the Surface Modeling palette, which is shown below. Surface modeling and sheets are primarily used when working with imported surface files, and are not generally required for building bodies.

The Surface Modeling palette includes various methods for creating sheets, including planes, revolved sheets, lofted sheets, Coons patch sheets, swept sheets, and sheets from faces as well as the ability to trim/untrim, stitch/unstitch, and extend sheets.

1. Plane
2. Revolve
3. Loft
4. Coons Patch
5. Sweep Sheet
6. Sheet from Face
7. Trim/Untrim Surfaces
8. Stitch Sheets
9. Unstitch Sheets
10. Untrim & Extend Surfaces

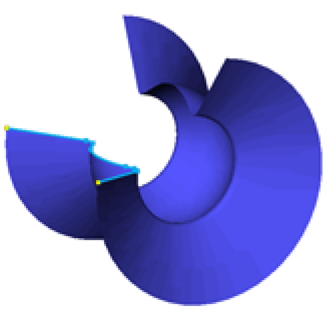
 Plane

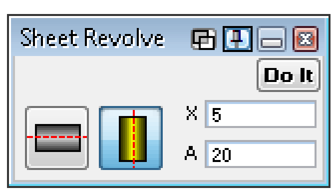


This button is used to create planes. When no geometry is selected, using this function will create a flat planar sheet based on the current coordinate system at a depth of zero. Triggering this function with a closed shape selected will create a sheet bounded by the selected geometry at a depth of zero in the coordinate system. If the closed shape is not planar, a plane will still be created by projecting the geometry to a depth of zero in the current coordinate system.

 Revolve

This button will open the Sheet Revolve dialog which allows users to revolve a shape around the horizontal or vertical axis by a specified number of degrees to create a sheet.



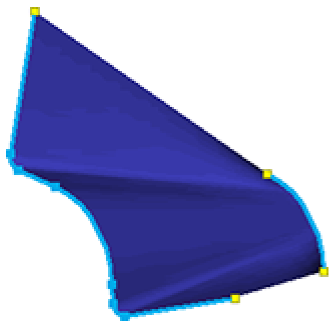


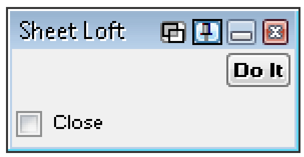
Sheet Revolve dialog

Select an open, terminated shape or a closed shape to be revolved. The axis buttons specify which axis the selected shape will be revolved about. If the horizontal axis is selected for the axis of revolution, a vertical value must be entered to specify the position of the revolution axis. Likewise, if the vertical axis is the axis of revolution, a horizontal value must be entered to specify the position of the revolution axis. The value entered in the A text box is the angle (specified in degrees) the selected shape will be revolved around the selected axis. A positive angle value will revolve the shape in a counter-clockwise direction and a negative angle in a clockwise direction based on the positive axis of revolution.

Loft

This button will open the Sheet Loft dialog which allows users to create a sheet through a series of open or closed shapes. The system will blend all the selected shapes into a smooth sheet. Sheet lofting produces ruled surfaces when only two shapes are selected and sculptured surfaces when three or more shapes are selected.

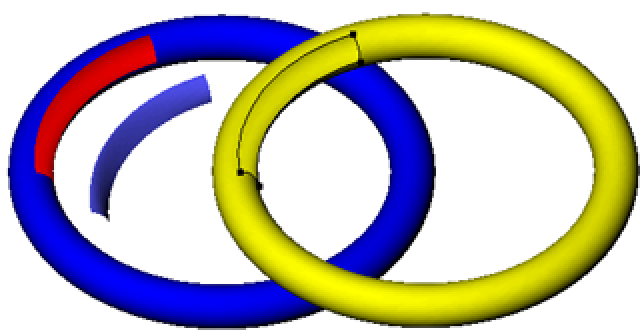




Sheet Loft dialog:

Select a series of shapes to be blended into a smooth sheet. These can be closed shapes or open, terminated shapes. The shapes act as the cross-sections through which the final sheet will be created. The system will blend the selected shapes into a sheet using C0 points (corners) as alignment points. If the Close checkbox is selected, the system will attempt to blend the first and last shapes together to form a closed sheet.

Coons Patch

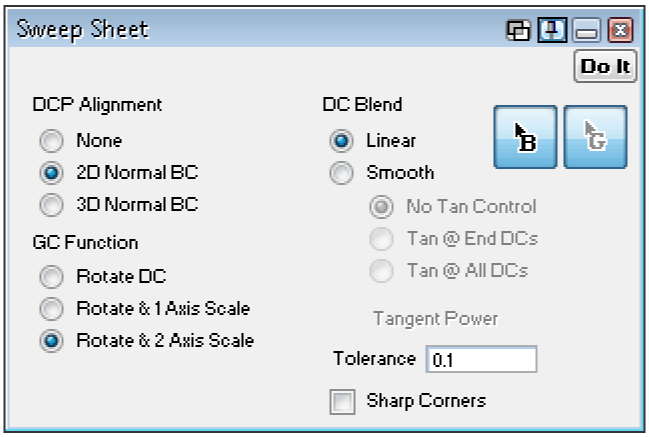


This button creates a sheet called a Coons patch through either three or four selected open, terminated shapes. A Coons patch is a surface type that uses boundary shapes (typically splines) and blends a smooth surface between them. Either three or four shapes must be designated as boundary shapes. Each shape can be any size or orientation as long as the endpoints are coincident (in the exact same location in X, Y and Z) and each shape is continuous and does not contain any sharp corners. The selected shapes represent the boundary of the sheet.

In some cases, connected splines or features can be selected to create a Coons patch. Also, if trimmed splines that do not have coincident points at the edges are imported, a Coons patch can be created provided that the ends of each trimmed spline are coincident. Often times, a Coons patch surface can be created if there are more than three or four line segments but the connected splines have three or four distinct corners.

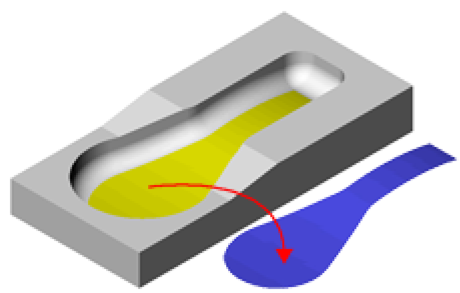
Sweep Sheet

The Sweep Sheet function is nearly identical to the Sweep Solid function described in Solid Modeling Palette. The only difference involves the alignment rules for the drive curves. Swept sheets do not use alignment or sync points selected on the drive curves to determine how the drive curves will be blended together. Only one alignment point per drive curve needs to be selected for the Sweep Sheet function. Refer to Solid Modeling Palette for additional information.

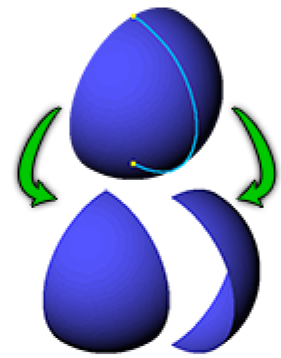


Sheet from Face

This option creates a sheet from the face of a solid or sheet. A face is one surface of a solid or sheet that is bound by an edge loop. Using the Face Selection mode accessed from the Floating Taskbar, users can select individual faces of a body. Selecting a face or faces and clicking on this button will create a sheet based on the face and bound by the edge loop of the selected face. Neighboring faces will produce stitched faces in the resulting sheet.



Trim/Untrim Surfaces

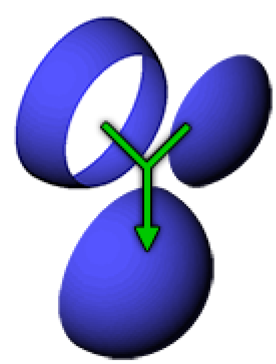


This button performs both the trim and untrim functions depending on the entities selected when the button is clicked. If a sheet and geometry are selected, the system will attempt to perform the trim operation. The trim function breaks a single sheet into two separate sheets at the selected trim geometry. The geometry selected for the trim operation must completely cut the selected sheet into two pieces. If the geometry does not lie on the selected sheet, the geometry will be projected onto the selected sheet and the trim operation will be performed. Holding down the `Alt` key while clicking the Trim/Untrim button will perform both the trim and untrim operations at once. The system will untrim the selected one-faced sheet and then trim that sheet to the selected geometry in one step, never attempting to create a valid face from the untrimmed surface.

If only a sheet is selected, the system will attempt to untrim. The Untrim function only works with single-faced sheets. The edge loop is what bounds the underlying surface definition into a finite bounded surface. The Untrim function removes the bounding edge loop so that the underlying surface definition replaces the selected surface. The untrimmed surface will be bound by the workspace stock size.

> ⓘ This is useful when working with imported IGES files that are not stitching or solidifying due to edge loops of neighboring surfaces not joining within the specified tolerance. If this is the case, the user can select the problem sheet, untrim it to create the underlying surface definition and then trim that surface with the extracted edges from neighboring sheets.

Stitch Sheets

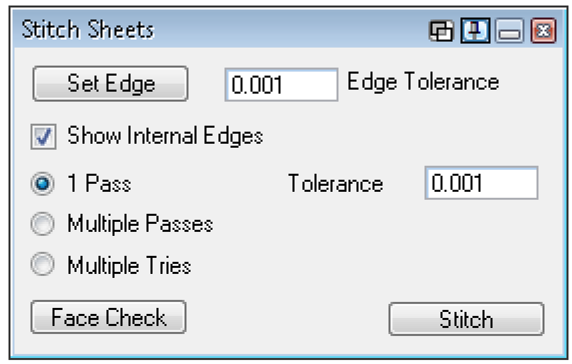


This button will open the Stitch Sheets dialog. This dialog provides different methods for stitching sheets together as well as tools to analyze stitched sheets. In order to stitch sheets, the user must select all the sheets to be stitched, choose a stitching method from the Stitch Sheets dialog and click the Stitch button.

Surfaces are stitched at their edges. When surface files are imported into the system, each surface is represented as a single-faced sheet. A face is a trimmed surface with an edge and knowledge of its neighboring surfaces. An edge is the trim curve that bounds a face. For sheets to successfully stitch to their neighbors, the edges of these sheets must align with each other within a specified tolerance; otherwise, there are holes (gaps) and adjoining sheets that are separated by a gap cannot be stitched.

The system must be in Edge Selection mode in order to view edges. The Show Internal Edges checkbox provides a method for only viewing the external edges of a part. The internal edges are edges that can be viewed from the inside of the model looking out, while the external edges are ones that can be viewed from the outside. The external edges are the edges that need to be stitched together. After performing a stitching operation, the only external edges that will be visible are the edges that could not be stitched together because of the tolerance gap. All stitched edges become internal edges.

Once the problem edges have been identified, if the gap is large, the user can build a sheet using Coons Patch or another Surface Modeling tool to fill the hole. Often times the gaps are small and can be fixed by applying a looser tolerance to select edges. This is accomplished by selecting the problem edges, entering the looser tolerance in the Edge Tolerance text box, and clicking on the Set Edge button. Applying a different edge tolerance to certain edges often aids the system in stitching together all the sheets together.



There are three stitching methods offered in this dialog 1 Pass, Multiple Passes and Multiple Tries. Each of these methods uses the Tolerance value entered in the dialog. The tolerance can be thought of as the maximum gap that can exist between the edges of two sheets that the system will still stitch together. For example, the edges of two adjoining sheets are 0.002mm apart. If the tolerance set is 0.002mm or greater, the two sheets will be stitched together and the result will be a single sheet. If the tolerance is less than 0.002mm, the two sheets will not be stitched together and remain two separate entities. The minimum tolerance is set by the system and is 0.00002mm or 0.00000079". The tolerance specified by the user cannot be less than this value.

1 Pass:

When the 1 Pass option is selected, the system will attempt to stitch all selected sheets at the given tolerance. The system will take one pass at the specified tolerance in this attempt. The system will analyze each sheet, its neighbors and its edges and if they fall within the tolerance, stitch the sheets together. If all edges stitch together at this tolerance into a single closed sheet, the system will solidify the sheets, thereby creating a solid. Otherwise, the result will be a multi-faced sheet composed of all of the sheets that could be stitched together.

Multiple Passes:

When this option is selected, the system will attempt to stitch together all selected sheets by performing a series of single passes. The system will begin at the minimum tolerance (0.00002 mm or 0.00000079") and attempt to stitch the sheets at that tolerance. The tolerance entered by the user provides the maximum tolerance that the system will go to in its attempt to stitch the sheets. Multiple passes will be taken at incremental tolerance steps ranging from the minimum tolerance (set by the system) to the maximum tolerance (set by the user). On each pass, the system will stitch together all the sheets that it can at that tolerance and then proceed to another pass at the next tolerance, attempting to stitch any remaining sheets. The progress bar, located at the bottom of the workspace, displays the number of sheets remaining to be stitched and the tolerance being used on the current pass. When all of the passes have been completed, if the sheets stitched together into a single closed sheet, the system will automatically solidify the sheets, resulting in a solid. Otherwise, the result will be a multi-faced sheet or sheets.

Multiple Tries:

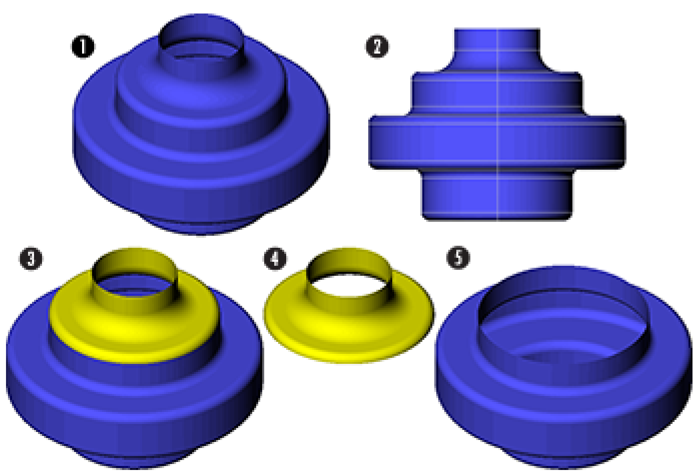
This option is similar to Multiple Passes in that it takes incremental passes ranging from the minimum tolerance (0.00002 mm or 0.00000079") to the maximum tolerance, which is the Tolerance value entered in the dialog. The system will attempt to stitch all the sheets together at each tolerance increment, starting over after each pass that does not stitch all the selected sheets together. The system is looking for the smallest single tolerance that will stitch the entire part. This is similar to taking a series of one-pass steps and undoing after each one. The stitching process will stop when all the selected sheets have been stitched together into a single sheet even if this occurs before the maximum tolerance is reached.

Face Check:

Clicking the Face Check button will perform a face validity check on the selected sheets. This is identical to the face validity check that is run when the Solids > Tools > Check Body Validity item is selected. The face check produces an error message for each invalid face and also deselects the problem faces. It is useful to run a face check if stitching has failed to identify problem areas before attempting to stitch again. When a face fails the check, it must be deleted and re-created in order for future stitching attempts to be successful.

Unstitch Sheets

This button will unstitch or detach faces of a sheet. This will also convert solids into sheets. The faces will be unstitched at the edge loop which bounds the selected face or faces. Example of unstitching a sheet illustrate an example of unstitching. The first row of pictures shows geometry revolved to create a single multi-faced sheet with the edges displayed to differentiate between the faces of the sheet. In the second row, multiple adjoining faces are selected to be unstitched.

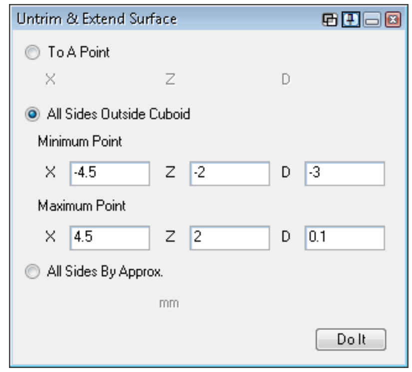


1. Original Sheet
2. Edges of sheet which separate the faces
3. Selected face to be unstitched
4. Resulting Sheet
5. Resulting Sheet

Example of unstitching a sheet

Untrim & Extend Surfaces

This function untrims and extends sheets to the user-specified point(s), effectively eliminating all trimmed loops and plugging in any holes that exist in the sheet. After selecting a single-faced sheet, click this button to open the Untrim & Extend Surface dialog. The system offers three methods for untrimming and extending sheets: To A Point, by All Sides Outside Cuboid specified by two points, and by All Sides By Approximate value. This will not change the shape of the surface, and is most useful when attempting to heal surfaces.



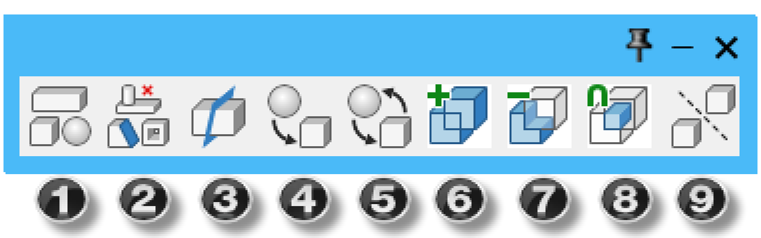
To A Point:
The surface will be untrimmed and extended to the specified point along the chosen axis. Note that multiple edges may be extended depending on their proximity to the specified point.

All Sides Outside Cuboid:
The surface will be modified (either reduced or enlarged) to fit within the cuboid, specified by entering its minimum and maximum point. The default coordinates for the cuboid coincide with the boundaries of the stock.

All Sides by Approximately:
All sides of the sheet will be extended along the current coordinate system by the length entered in the in./mm field. Note that this field only accepts positive values.

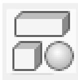
Solid Modeling Palette

To access the Solid Modeling palette, click the Solid Modeling button in the Command Toolbar. You access all of the solid modeling functions from the Solid Modeling palette. Click the first button to display the Create Solid palette, which provides options for creating atomic or primitive bodies. Click the second button to display the Advanced Solid Modeling palette, which provides for such functions as offsetting and rounding. The remaining buttons in the Solid Modeling palette provide various operations for solids and sheets, include slicing, replacing, swapping, adding, subtracting, intersecting, and separating.



1. Create Solid Palette
2. Advanced Solid Modeling Palette
3. Slice
4. Replace
6. Add
7. Subtract
8. Intersect
9. Separate

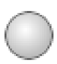
5. Swap
10.



Create Solid Palette

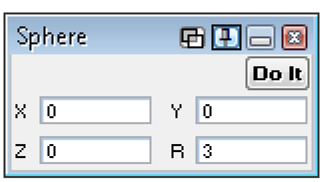
Clicking the Create Solid button opens the Create Solid palette, which provides various methods for creating atomic bodies and solidifying sheets into solids. Atomic or primitive solids are non-divisible bodies in that they were not created from any other bodies. The following sections describe the controls you can access for each button.

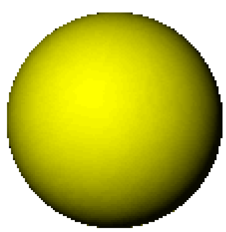
1. Sphere
2. Cuboid
3. Extrude
4. Revolve
5. Loft
6. Sweep Solid
7. Solidify

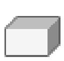


Sphere

Opens the Sphere dialog which you can use to create spherical bodies. Enter the H, V, D (horizontal, vertical, and depth) coordinates of the centerpoint of the sphere and a radius value. Click the Do It button to create the sphere.

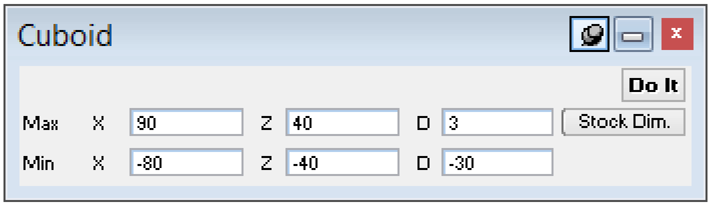


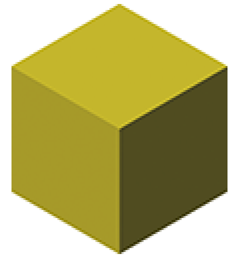




Cuboid

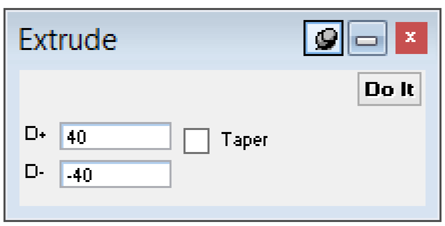
Opens the Cuboid dialog which you can use to create cubes and rectangular bodies. Enter a minimum and maximum horizontal, vertical and depth value to define the volume of the cuboid. These values are measured from the origin of the current coordinate system. The labels used in the dialog may vary when the current coordinate system aligns with one of the primary planes. The labels X, Y and Z will be used instead of H, V and D. Click the Do It button to create the cuboid. The Stock Dim. button loads the workspace stock definitions based on the XY coordinates.

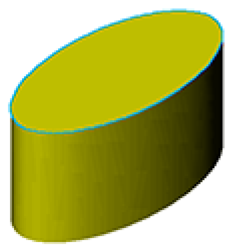




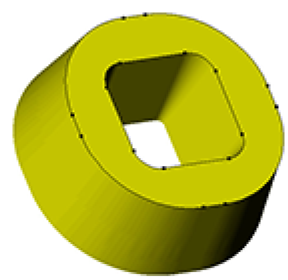
Extrude

Click this button to open the Extrude dialog which you can use to create a solid by selecting a closed shape and extruding it along the depth axis. A closed 2D shape can be extruded along the depth axis of the active CS in the positive and/or negative direction based on the values you enter. The extrusion starts at the depth location of the selected geometry. Click the Do It button to create the extruded solid.



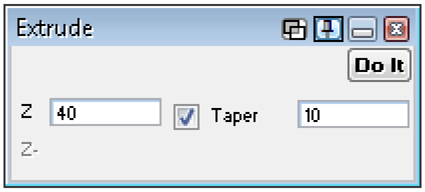


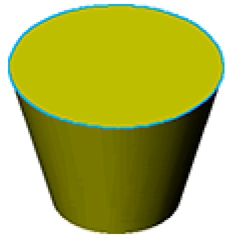
Straight and Tapered extrusions work with multiple geometry loops. The order of selection is important because the system uses the loop that is selected first as the outer profile.

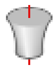


Tapered Extrusions:

You can taper an extruded solid. Type a value in the Taper box to specify the angle of the taper. You can only taper an extrusion in one direction along the depth axis in order to properly calculate the taper on the solid. When you enter a value in the Taper box, the negative depth specification is grayed out. A negative value can be entered for the Z+ value so that the shape can be extruded along the negative direction of the depth axis. You can also enter negative angle values for the taper amount. When extruding a solid, the shape selected to be extruded is duplicated along the depth axis by the amount specified. For an extrusion without a taper, the offset shape is an exact duplicate of the original shape. When creating an extrusion with a taper, the offset shape is going to be larger or smaller (depending on if the taper is positive or negative) than the original shape.

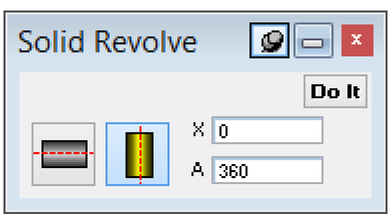


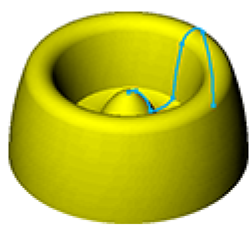


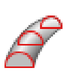


Revolve

This button opens the Solid Revolve dialog which you can use to revolve a shape a specified number of degrees around either the horizontal or vertical axis to create a solid. Select any terminated or closed shape to be revolved. The selected shape must be an open terminated shape, rather than a closed shape, in order to revolve 360° about an axis position of zero. There cannot be a line on the axis of revolution or else revolving will fail because it will produce self-intersecting edges. The axis buttons designate whether the shape is revolved around the horizontal or vertical axis of the current coordinate system. If the horizontal axis is selected for the axis of revolution, you must enter a vertical value to specify the position of the revolution axis. Likewise, if the vertical axis is the axis of revolution, you must enter a horizontal value to specify the position of the vertical axis that will be the revolution axis. The value entered in the A text box is the angle (specified in degrees) the selected shape is revolved around the selected axis. A positive angle value revolves the shape in a counter-clockwise direction and a negative angle in a clockwise direction based on the positive axis of revolution.

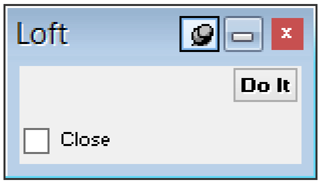


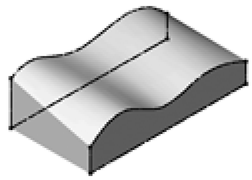




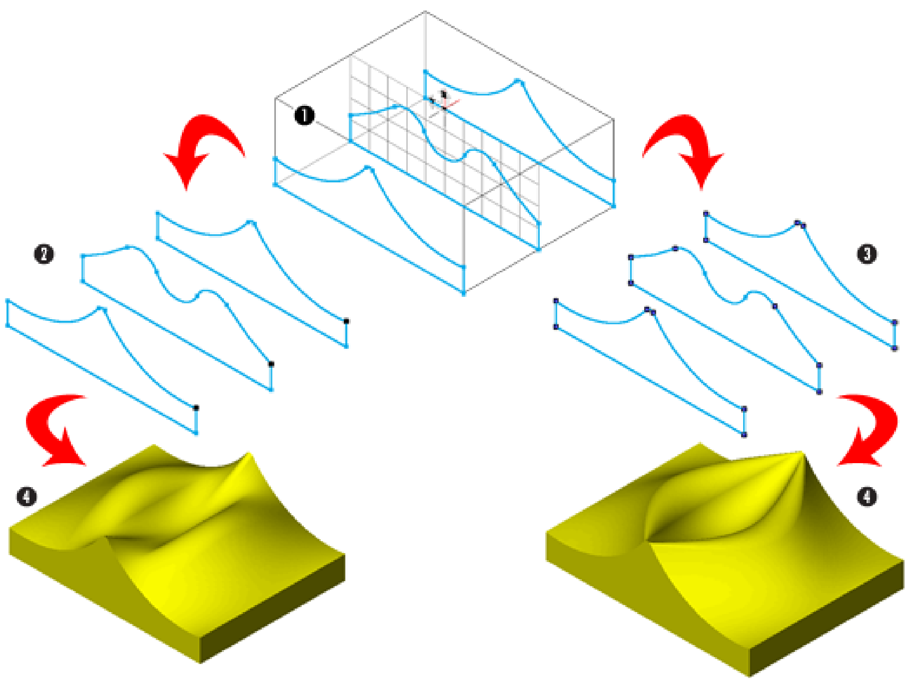
Loft

Click this button to open the Loft dialog. Lofting is also referred to as blending or skinning. Select a series of closed shapes to be blended into a solid. These selected shapes define the cross-sections of the resulting lofted solid. The shapes should be selected by choosing points on each shape that will act as alignment or synchronization points. The system will break up the section of the shape between alignment points into an equal number of segments and create a face (surface) by matching each segment. The alignment points on each shape match up in the finished lofted solid. To achieve the best results when lofting, select all relevant points to act as alignment points. If the shapes selected have the same number of corners, one alignment point per shape can be selected and the system blends the lofted solid using the corners as alignment points. A corner is defined as the non-tangent intersection between two features. When selecting alignment points on the shapes, either select all alignment points per shape in the same order or select the first alignment point on all shapes then the second, etc. The system looks at the selection order for each shape. If the Close box is checked, the system attempts to blend the first and last shapes together into a closed solid. Click the Do It button to create the lofted solid.



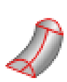


In Loft, three closed shapes are to be lofted. The top picture shows the shapes selected to be lofted. The second set of pictures show the alignment points selected–one has one alignment point per shape, the other six alignment points per shape. The third set of pictures show the lofted bodies created from the shapes. The solid created from the shapes with one alignment point selected blends the shapes using the four corners of each shape because the shapes have the same number of corners. The solid created from the shapes with six alignment points selected blends the solid based on all of the alignment points, allowing you more control over the solid that is generated.

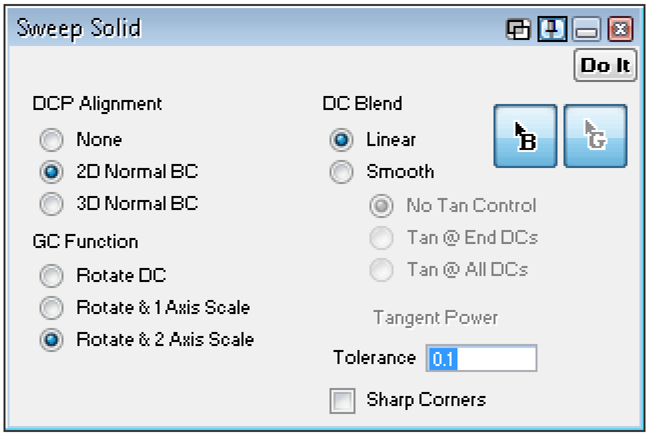


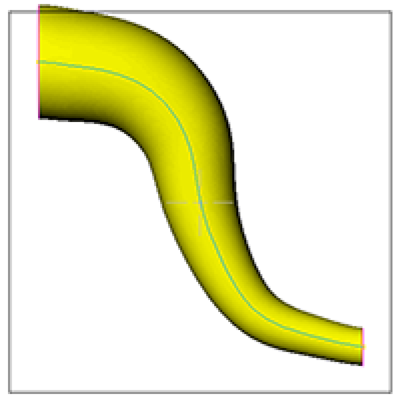
1. Curves to be lofted.
2. 1 alignment point is selected.
3. 6 alignment points are selected.
4. The resulting lofted solid

Example of the use of alignment points in lofting

 Sweep Solid

This function provides options for creating swept solids. A swept solid is created by selecting a drive curve(s) that define the basic shape of the sweep, defining the base curve, which defines the spine or primary edge of the sweep, and an optional secondary edge or guide curve. The tolerance value specifies how closely the generated swept solid will be to the “true” swept surface. For an explanation of terms that you use with the Sweep function, see “Swept Shape Terminology” on page 53.





Drive Curve Plane (DCP) Alignment:

The DCP Alignment setting determines the alignment of the drive curve plane in reference to the base curve, that is, how the drive curve is swept around the base curve.

For examples that illustrate the various alignment and sweeping options, see Swept Shape Examples. For an illustration of the differences between the None and 2D Normal BC alignment options, see Swept Shape Examples. For a comparison of the 2D and 3D Normal BC options, see Swept Shape Examples. For an illustration of one base curve, which is acting as a spine, and two drive curves, see Swept Shape Examples.

None:

The selected drive curve determines the orientation of the drive curve plane. All the frames used for the drive curve plane are parallel to the plane of the selected drive curve. The drive curve is simply translated along the base curve but not rotated in any way.

2D Normal BC:

The selected drive curves are rotated around the sweeping plane normal vector so that it is perpendicular to the base curve in the sweeping plane. However, the drive curve does not remain normal to the base curve as the base curve moves in Z (or the depth of the current CS). That is, the vertical axis of the drive curve plane always stays parallel to the depth axis of the sweeping plane. The alignment is locked.

3D Normal BC:

As in the case of 2D normal, the selected drive curve is rotated around the sweeping plane normal vector so that it is perpendicular to the base curve in the sweeping plane, and it follows the base curve as it changes in depth. That is, the horizontal axis of the drive curve plane lies in the sweeping plane and is normal to the base curve at every point.

GC Function:

You can control transitions between drive curves (used when there are more than two drive curves). The transitions are set using the GC (guide curve) Function. These options utilize the guide curve to “scale” the drive curve in one or more axes. The differences between these options are illustrated in Swept Shape Examples.

Rotate DC:

Selecting this option restricts the guide curve to aiding the base curve in rotating the drive curve only.

Rotate & 1 Axis Scale:

Allows the guide curve to aid the base curve in rotating the drive curve and, in addition, to scale the drive curve in one axis at a time as it transitions through both the base curve and guide curve.

Rotate & 2 Axis Scale:

Allows the guide curve to aid the base curve in rotating the drive curve and, in addition, to scale the drive curve in two axes at a time as it transitions through both the base curve and the guide curve.

DC Blend:

The DC (drive curve) Blend modes define if the transitions are Linear or Smooth.

Linear:

Linear blends will produce the same result as if the sweep was performed on the sections separately; in other words, as if the results of a sweep between DC1 and DC2 were added to the results of a sweep between DC2 and DC3.

Smooth:

A smooth continuous solid will be swept between the drive curves as specified by the option selected. The effects of the various tangency options are shown in Swept Shape Examples.

No Tangent Control:

There will simply be a smooth transition between drive curves. The user has no control over tangency.

Tangent at End DCs:

The swept solid is blended to be tangent to the base curve at the first and last drive curve. Transitions between any other drive curves will be smooth without any control, as with the No Tangent Control option.

Tangent at All DCs:

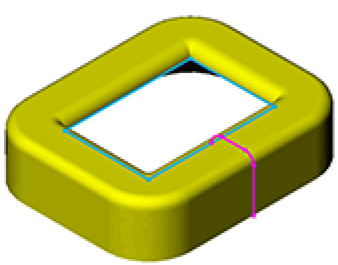
The swept solid is blended to be tangent at all drive curves.

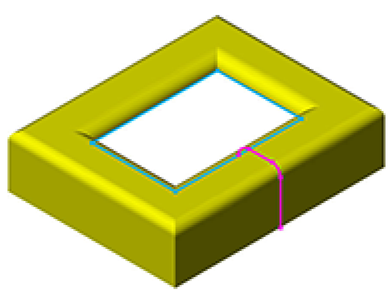
Tangent Power:

Tangent Power controls the strength of the tangent blending when specifying control for the ends of all drive curves. The range is from 0.0 (no control, works the same as No Tangent Control) to 1.0 (a very sharp transition with long straight parallel sections).

Sharp Corners:

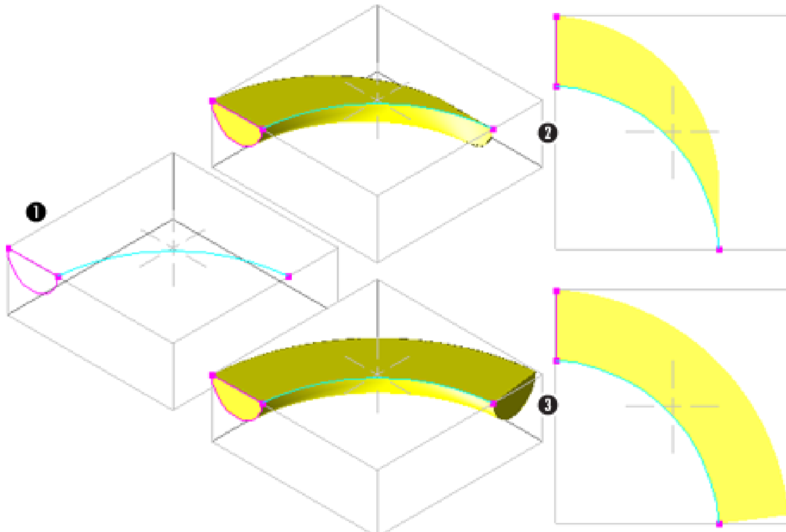
This checkbox determines whether corners should be smooth (rounded) or sharp (square). If Sharp Corners is checked, the system will extend a solid so the corners meet to have mitred corners, keeping the drive curve’s profile.





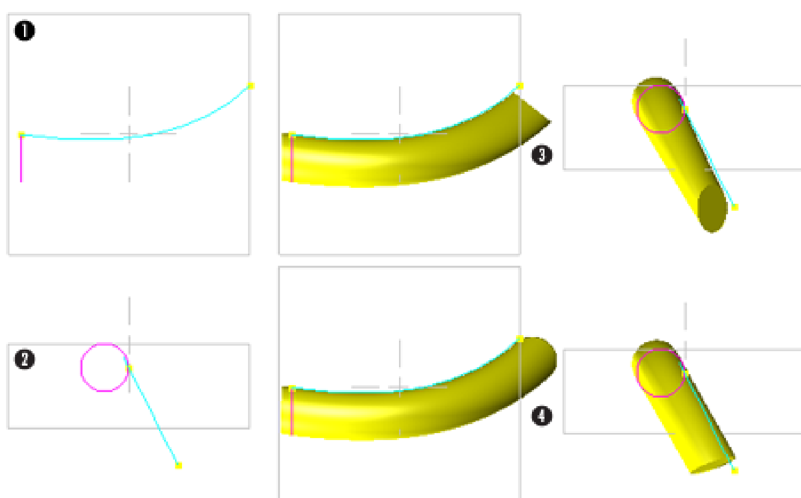
Swept Shape Examples

Following are examples of sweeping shapes to illustrate the differences between the functions and how selection can matter.



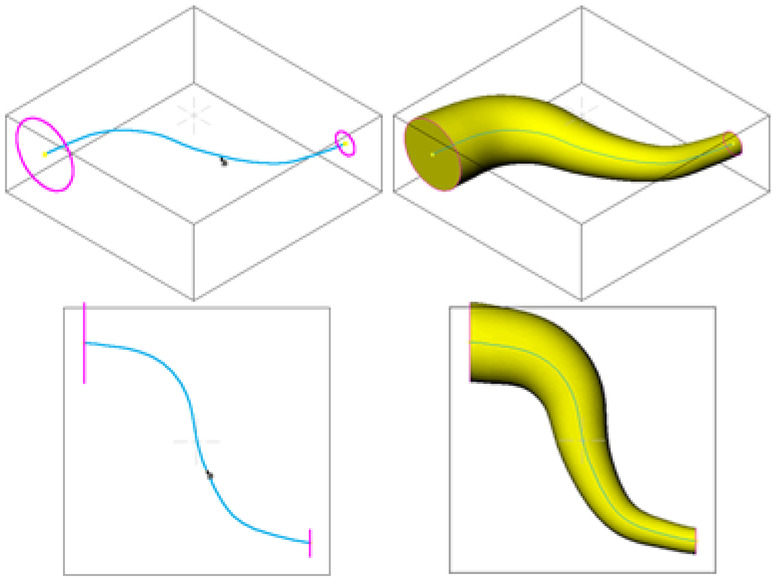
1. Top view cross section
2. DCP: None
3. 2D Normal BC

None vs. 2D Normal BC alignment

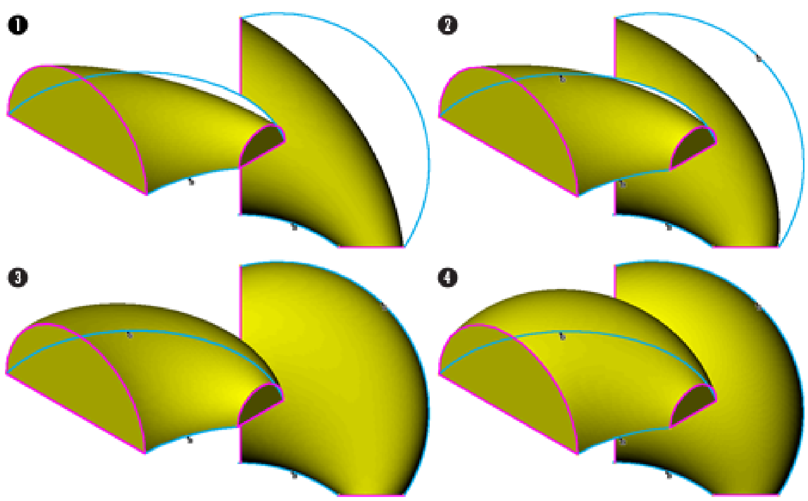


1. Isometric view
2. Top view
3. 2D Normal BC
4. 3D Normal BC

2D Normal BC vs. 3D Normal BC alignment from the Top and Right views

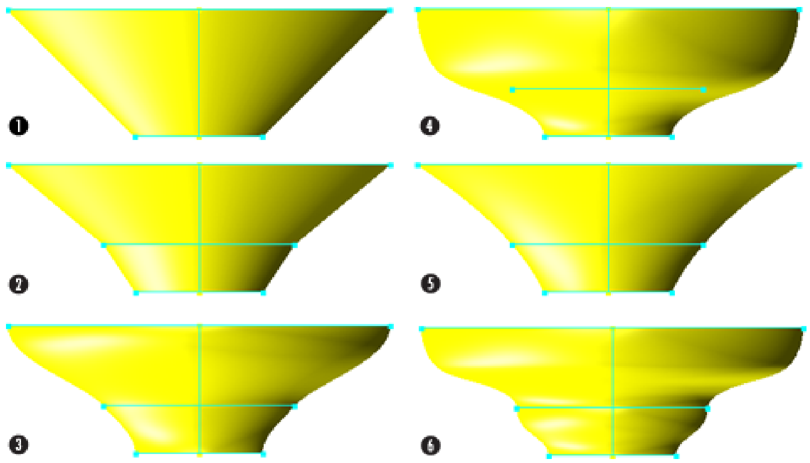


One base curve and two drive curves



1. Base Curve only
2. Guide Curve and Rotate DC
3. Guide Curve, Rotate & 1 Axis Scale
4. Guide Curve, Rotate & 2 Axis Scale

GC Function examples of 2 Drive Curve sweeps using 2D Normal alignment



1. 2 DC - Linear
2. 3 DC - Linear
3. 3 DC - No Tan Control
4. 2 DC - Tan @ End DCs
5. 3 DC - Tan @ End DCs
6. 3 DC - Tan @ All DCs

DC Blend examples of swept bodies with two and three Drive curves

Swept Shape Terminology

 Base Curve:

The base curve can be a 2D or 3D curve and must either be a closed shape or an open terminated shape. It should also be defined in the exact 3D location of the desired swept solid. The B-pointer marker , located in the Sweep Solid dialog, is used to designate the base curve. The B-pointer can be dragged from its box in the dialog and dropped on the geometry to be used for the base curve. It can be removed in the same manner, dragging it from geometry back to its box in the dialog. The location of the B-pointer marker on the base curve has no affect on the resulting swept solid.

Drive Curve:

The drive curve is a 2D curve that defines the cross-sections of the swept solid. The drive curve must be defined in the correct 3D location for the desired swept shape. The following is a list of rules regarding drive curve creation.

- Drive curves must be planar.
- All drive curves must be closed shapes. Open terminated shapes can be used for the drive curves only when these open shapes can be capped by a single plane.
- Select drive curves by choosing alignment points on each of the shapes.
  - Alignment points must be connectors or terminators on the drive curves.
  - If more than one alignment point is selected, all corner connectors (non-tangent intersections) must be selected.
  - If only one alignment point is selected for each shape, corner connectors will automatically be aligned. In this case, each shape must have the same number of corner connectors or else the sweeping operation will fail.
  - The same number of alignment points must be selected on each shape.
  - Full circles have a default alignment at 12:00 in their respective planes. This allows the user to select circles for the drive curves without needing to create and select

alignment points. If full circles are selected for the drive curve without alignment points selected and the resulting swept solid is not the desired result, create terminators or connectors on the circles in order to control alignment.

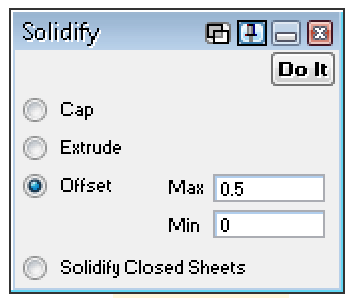
Guide Curve (GC):

This is an optional curve that you can use to control alignment of the drive plane, replacing the “normal-to-base-curve” standard. You can also use this option to “scale” the drive curve in one or more axes. Apply the G-pointer marker to a feature to be used as a Guide Curve.

Sweeping Plane:

The sweeping plane is the current coordinate system when the Sweeping function is performed. The sweeping plane affects the DCP Alignment options when 2D Normal BC or3D Normal BC is selected. The sweeping plane also determines the CS to which the resulting swept solid is assigned.

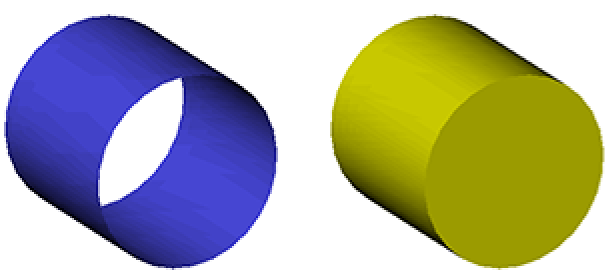
Solidify



This button accesses the Solidify dialog which provides options for creating solids from sheets. It is often useful to solidify sheets into solids to reduce part complexity and so that solid modeling functions can be performed. However, it is not necessary to solidify sheets in order to machine them. Surfaces can be machined directly without being solidified or stitched together. This dialog includes four options for converting sheets into solid bodies. To solidify a sheet, select the desired option in this dialog, select the sheet and click the Do It button. The first three options can only be used to change a single sheet into a solid. The Solidify Closed Sheets option can be used on multiple sheets. Each method is described below.

Cap:

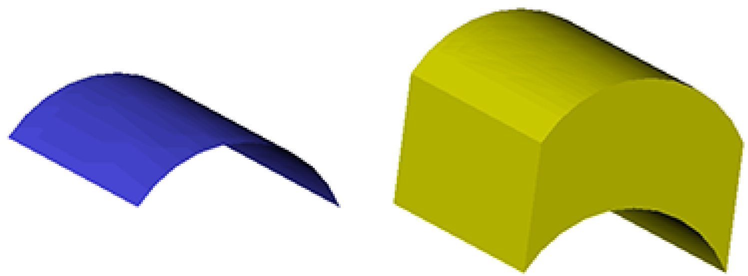
The Cap option creates a solid from an open sheet by creating planar surfaces at any open ends of the selected sheet. The enclosed area is filled in to create the solid. In order to use the cap option to solidify a sheet, the sheet selected to be capped must only require planar surfaces to provide closure of open ends of the sheet. Solidifying a sheet using Cap illustrates the capping function to solidify a sheet.



Solidifying a sheet using Cap

Extrude:

The Extrude option creates a solid by extruding a selected sheet along the depth axis of the current coordinate system. The sheet can be extruded in either the positive or negative direction along the depth axis. You enter a value which specifies how far along the depth axis to extrude the sheet. Entering a negative value extrudes the sheet along the negative direction of the depth axis. When performing an extrusion, the sheet selected to be extruded is duplicated along the depth axis by the amount specified and the area between these sheets is filled in to create a solid. To use the extrude option to solidify a sheet, the sheet selected to be extruded cannot overlap itself or fold over. Also, the extrude axis (depth axis of the current coordinate system) cannot intersect the sheet in more than one place and cannot be parallel to the edge of the sheet. Solidifying a sheet with Extrude illustrates extruding a sheet in order to solidify it.

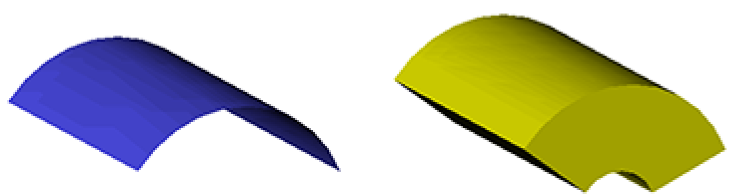


Solidifying a sheet with Extrude

Offset:

The Offset option creates a solid by creating an offset sheet of the sheet selected to be solidified at a specified distance and filling in the area between the original sheet and the offset sheet to create a solid. The definition of an offset is that every point on the offset sheet will be normal (perpendicular) to a point on the original sheet. Offsetting can be thought of as rolling a ball with a diameter the size of the offset amount along the sheet.

When using the Offset option, you specify a Max and/or Min value that acts as the offset amount. The sheet selected to be solidified acts as the zero reference point for the Max and Min values. These values can be positive or negative. The sheet selected to be solidified is offset in one direction by the Max value and in the other direction by the Min value. Solidifying a sheet with Offset illustrates the offsetting function to solidify a sheet.



Solidifying a sheet with Offset

Solidify Closed Sheets:

This option creates a solid by filling in the volume enclosed by separate adjoining sheets. The sheets selected do not need to be stitched together in order to use this option. There can be no holes or gaps between the sheets to be solidified. This option provides the same functionality as the stitching options accessed from the Surface Modeling palette.

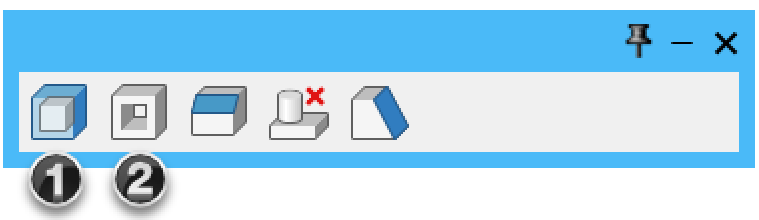
Advanced Solid Modeling Palette

Clicking the Advanced Solid Modeling button opens the Advanced Solid Modeling palette. This palette includes Shell/Offset, Blending, Unstitch Solid, and Draft functions. Each function is described below.

1. " Offset/Shell" on page 56
2. " Blending " on page 58
3. " Unstitch Solid" on page 60
4. " Draft" on page 62

Offset/Shell

This button is located in the Advanced Solid Modeling palette and accesses the Offset/Shell dialog. There is an Offset button and a Shell button within the dialog. Each is described below.

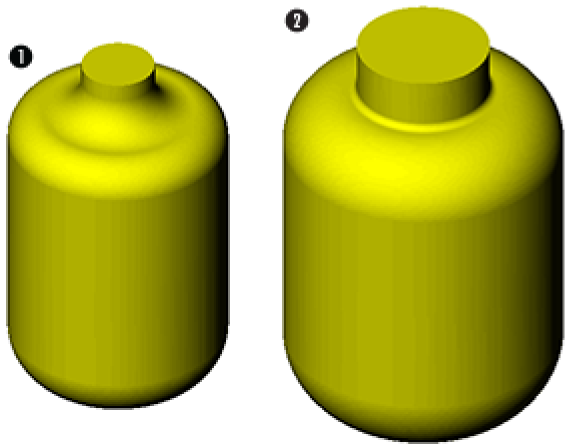


1. Offset
2. Shell

Offset:

The Offset function can be performed on solids, sheets and individual faces. This function will enlarge or shrink a solid or face by the specified Offset amount. Positive offset values will make the selected entities larger; negative values, smaller. Both bodies and faces can be offset and multiple bodies and faces can be offset at one time. To offset a solid or face, select the solid or face(s), enter an offset amount and click the Do It button. It should be noted that the original solid selected to be offset is replaced by the offset solid. The original solid can be restored from the History list, if necessary.

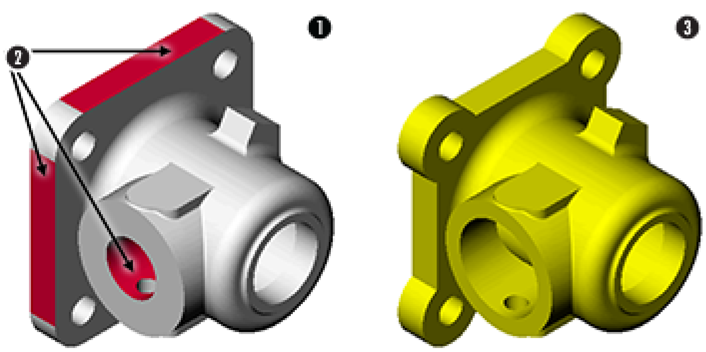
In Example of an Offset solid, the original solid is a canister and the offset is enlarged by a given amount. Note the change in the size of the fillet at the top of the bottle – it is a much smaller fillet than on the original solid. This is due to the fact the system must extend the other faces to intersect each other. This example would fail to offset if the offset amount was greater than the size of the fillet.



1. Original Solid
2. Offset Solid

Example of an Offset solid

In Offsetting selected faces of a body, only selected faces are offset and the unselected faces "grow" in order to provide for the offset.



1. Original body
2. Faces selected to be offset
3. Resulting body with offset faces

Offsetting selected faces of a body

In certain instances, the offset function will not succeed because the specified offset amount creates excessive topology changes. Topology is the term in solid modeling for the manner in which specific faces of a solid are positioned relative to each other. Modeling functions that change the shape of a face do not affect the topology unless the function requires a change to the way faces connect to one another along their edges. An example of offsetting that will require excessive topology changes, and therefore will fail, is if the offset amount is greater than an inside (concave) fillet of the face or solid to be offset. The offset function attempts to extend the unoffset neighboring faces to intersect with the faces that are being offset. When one or more of the unoffset neighboring faces is tangent to the offset face, no amount of extension will intersect with the offset face. Therefore the offset function will fail in this instance.

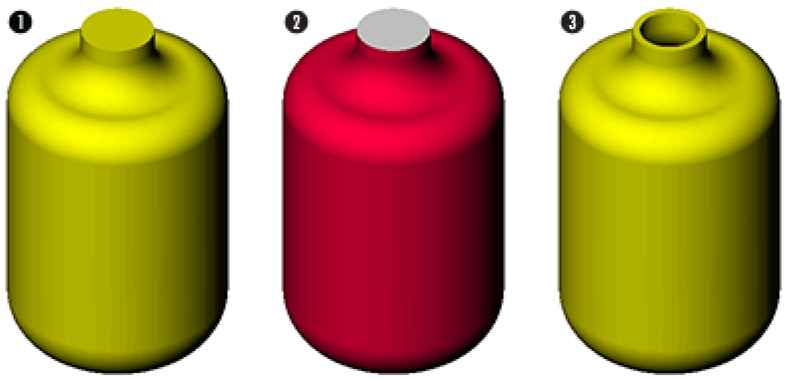
Offsetting Sheets:

You can perform the Offset function on sheets. A sheet has two sides, an inside and an outside. The outside of a sheet is defined as the side from which the positive direction of the surface normals are projecting outward. The negative direction of the surface normals projects to the inside of the sheet. When offsetting sheets, the location of the selected sheet will be moved in the direction of its positive surface normals by the specified offset amount. Surfaces are offset to the outside. The outside and inside of a sheet can be determined by turning on the Indicate

Sheet Side button in the Floating Toolbar. This button will display the outside of sheets as blue and the inside of sheets as red.

Shell:

The Offset amount specifies the amount the solid is shelled, which is equivalent to the wall thickness of the resulting hollow solid. Entering a negative value for the offset shells the solid to the inside, meaning that the outside face is not enlarged to account for the shell amount, and remains in its original position. A positive value offsets the solid to the outside and the solid itself becomes larger as a result. In this case, the inside of the face of the shelled solid is the same as the outside face of the original solid. Deselecting faces on the solid to be shelled creates entry holes at those faces. Select a solid, deselect the faces to be removed for entry holes while in Face Selection mode and click the Do It button to create the shelled solid. It is not necessary to create entry holes; however, if there are no entry holes on the shelled solid, it will need to be sliced or modified to see the results of the shell. Example of a shelling operation illustrates an example of a shelling operation which creates an entry hole by deselecting a face.

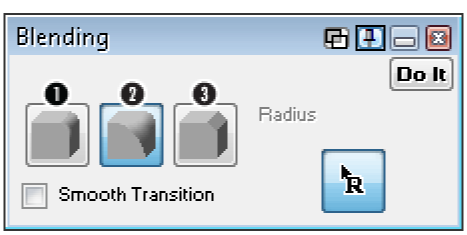


1. Original Solid
2. Solid selected with top face unselected
3. Result

Example of a shelling operation

Blending

The blending or rounding function contains options for blending edges of bodies. There are options for constant radius rounding, variable radius rounding and constant width chamfering. For an illustration of the Blending dialog options, see Blending dialog. The dialog options that appear depend on the blending type you select. To use the blending functions, you must select edges of solids or sheets. You can round multiple edges at one time.



1. Constant Radius
2. Variable Radius
3. Constant Chamfer

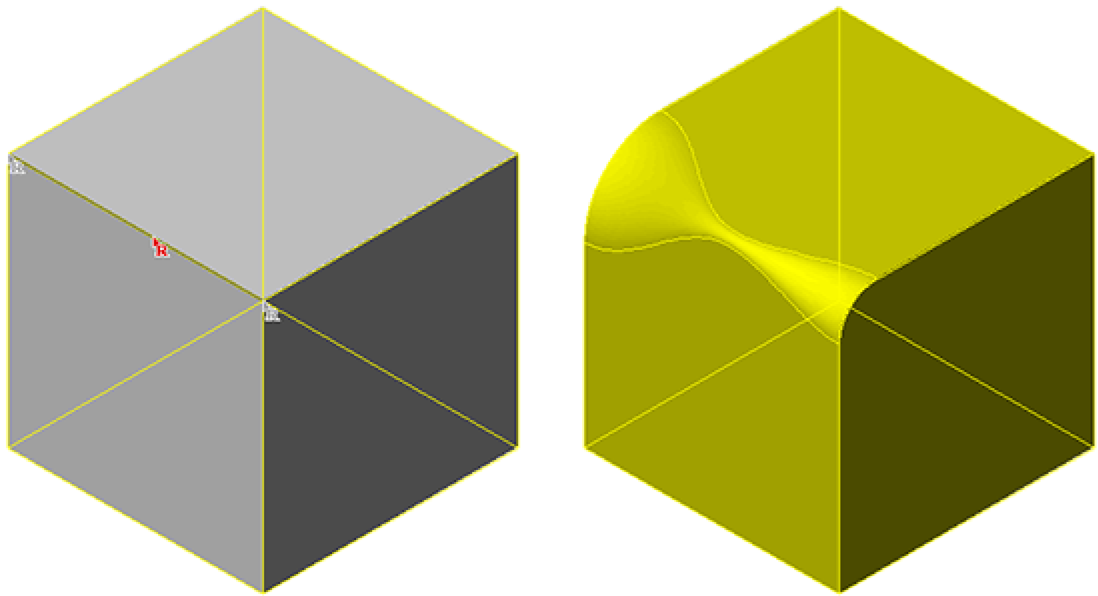
Blending dialog

Constant Radius:

When the Constant Radius button is clicked, selected edges are rounded according to the Radius value entered. When the Spherical Corners option is checked, rounding is applied down each sharp vertex at each corner.

Variable Radius:

The Variable Radius option allows users to specify different radii at different locations along a single edge. This is accomplished using the R-pointer markers which designate the locations along the edge(s) where the different radius values will be applied. For an illustration of a variable radius blend, see Example of a variable radius blend. There are R-pointers at each of the vertices (corners) of the selected edge that cannot be moved, and one additional R-pointer has been added at the midpoint of the edge. The radius values entered for each of the markers blend together to make a smooth transition.



Example of a variable radius blend

When an edge is selected to be variably rounded, R-pointer markers will snap to the vertices of the selected edge. Vertices are the points where two edges meet; they are the corners or endpoints of an edge. Variable Radius Rounding requires that the radius be specified at every vertex of an edge. Therefore, the R-pointer markers remain attached to the vertices and cannot be moved. Aside from the radius markers at the vertices, optional R-markers can be moved from the Blending dialog and placed at any location along an edge. The R-pointers at the vertices are drawn in gray, while the optional R-pointers are drawn in black. Black pointers can be moved; gray pointers cannot.

To apply a pointer, drag the R-pointer in the Blending dialog to a location along an edge. There is no limit to the number of optional R-pointers that can be placed along an edge. To remove an optional marker, drag it back to the Blending dialog. Once the markers are placed, the radius values must be entered. When one of the R-pointers is clicked on, it turns red and the radius value entered in the Blending dialog will be applied at that location. Different radius values can be entered for every R-pointer. Simply select the R-pointer by clicking on it, make sure it turns red, and enter the desired radius value. Once all of the radius values have been entered, the Do It button will create the rounded edge.

There is a Smooth Transition checkbox in the Blending dialog. If it is unchecked, the system will create a linear or constant rate of change in radius between R-pointer markers. This will not produce a smooth transition at the R-marker positions, but will produce a uniform change in radius along the edge. With the Smooth Transition option checked, the system will produce a variable rate of change in radius that starts slow and ends slow, providing for a smooth transition in the rate of radius change at each of the R-pointers.

Constant Chamfer:

When you click the Constant Chamfer button, selected edges are chamfered based on the Length value entered. The chamfer is calculated by offsetting both faces joined at the selected edge by the length value specified and then finding the intersection of these offset faces. From the intersection, normals are projected back onto the original faces. The points where the normals intersect the original faces are the start and end points of the chamfer.

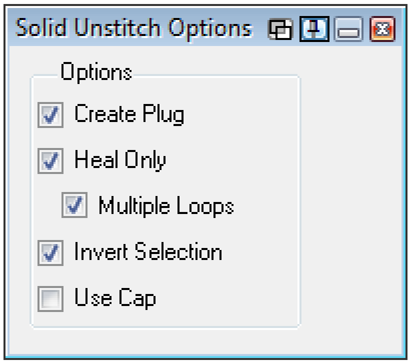
Unstitch Solid

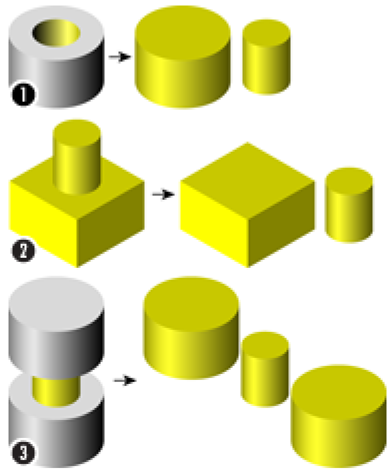
You can use the Unstitch function to separate solid models into component pieces or “healed” to remove holes. You can use Unstitch Solid to:

- Cap holes which are intended to be drilled and do not need to be machined along with the contour of the model.
- Remove fillets or chamfers that are unnecessary for machining, or would be more efficiently created using a tool.
- Remove blends on edges to simplify a model, which can provide faster and more efficient toolpath creation.
- Create core and cavity molds from hollow models.
- Calculate the volume that a bottle or any type of hollow container can hold by using the volume calculations in the Properties dialog.
- Use bodies created from the unstitching function to build EDM electrodes.

A solid can be thought of as a series of faces stitched together at their edges into a complete closed shape which is a solid (filled rather than hollow). Unstitching a solid provides a means of removing the stitching along selected edges to separate a solid into component bodies. The system extends faces along the selected edges to “heal” the components into valid solid bodies. The primary purpose of the unstitch solid function is in working with finished part bodies to create core and cavity bodies for mold work, or for removing holes and other details not needed for certain machining operations. This is particularly useful when working with models that were imported into the system. When using unstitch, either all of one group’s faces or edges must be selected which will separate all of the original solid’s faces into two unconnected groups. To select edges, the Edge Selection button in the Floating Toolbar must be clicked so that the edges of a selected solid are visible and able to be selected.

For greater control over the unstitching of solids, Right-click the Unstitch Solid button and select Options. This opens the Solid Unstitch Options dialog. This dialog offers multiple methods for removing holes or bosses in solids. These options are not mutually exclusive and can be used in conjunction with one another. You may select multiple options to use as a backup; one option may succeed where another has failed.





1. Create Plug
2. Heal Only
3. Multiple Loops

Examples of the Solid Unstitch Options.

Create Plug:

An additional solid or “plug” is always created instead of simply sealing up or removing a feature. For example, a hole in a solid becomes a plug and a boss becomes a separate solid.

Heal Only:

A plug is be created when unstitching. Note that Create Plug always takes precedence over Heal Only; only when Create Plug fails and Heal Only is checked will Heal Only be performed.

Multiple Loops:

The multiple loops option is an additional method to heal bodies. The unstitch option may or may not work on some bodies, the Multiple Loops option is simply another method to use and is particularly useful in a situation like the image to the right.

Invert Selection:

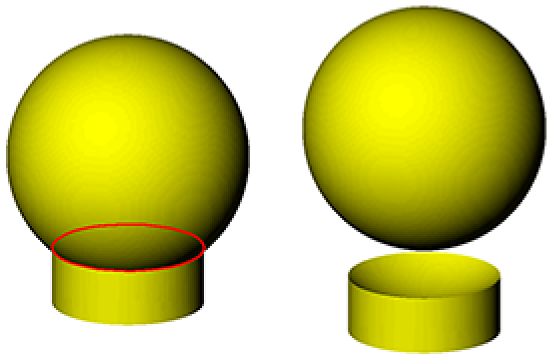
If unstitching cannot be done based on the selection, the selection will invert and reattempt the unstitch.

Use Cap:

During an unstitch, faces are typically extended to seal off a hole, which can result in an odd 3D shape. This option creates a 2D plate that patches the hole.

Unstitching Components

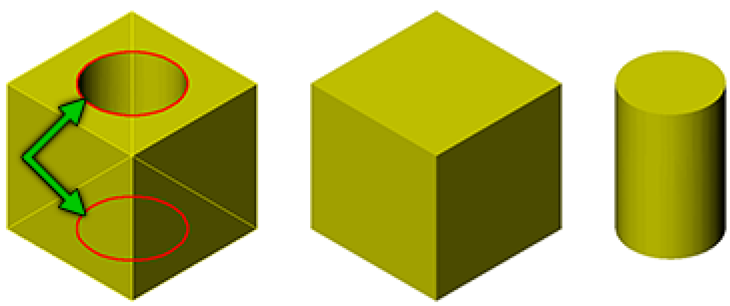
In Example of unstitching the original solid is a sphere with a cylindrical base. Selecting the intersecting edge to unstitch with Create Plug enabled will create two solids. One will be the sphere, which will not have a hole where the cylinder was previously attached. The other solid will be a cylinder: flat on one end and concave (spherical) on the end that was initially attached to the sphere.



Example of unstitching

Healing Components

Unstitching can also remove or “plug” holes. The following series of pictures (Another example of unstitching to “heal” a body) illustrate this aspect of Unstitch Solid. The original body is a cube with a through hole at its center. The hole has two edges, one at each exit. To unstitch this type of hole, select both the edges of the hole and `click` the Unstitch button. The two resulting bodies are a cylinder (the hole) and a solid cube with no holes. In this example, one of the bodies is completely empty. Unstitch will invert holes into a solid.



Another example of unstitching to “heal” a body

Draft

If you are licensed for SolidSurfacer, the Advanced Solid Modeling dialog offers a Draft button.

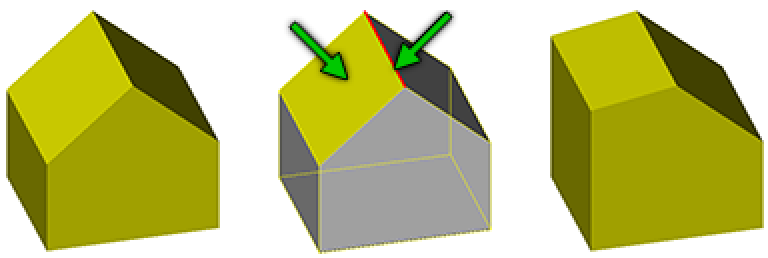
The Draft dialog applies a draft angle to selected faces of a solid. The draft function is primarily intended for mold work where draft angles are necessary in order to pull apart a mold and remove the part.

To draft a solid:

1. Select the faces to be drafted, including a reference entity indicating where the draft will begin, which is usually an edge.
2. Enter the draft angle.
3. `Click` the Do It button.

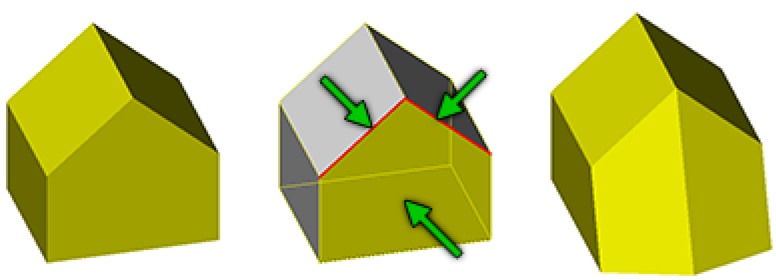
It is recommended that the draft be applied to the final core and cavity bodies rather than the part model that is subtracted or intersected to form the mold. To apply a draft angle, you must select the faces to be drafted and a reference entity, usually an edge, from which the draft will start. Multiple faces and edges can be selected and drafted at one time. To select faces and edges, the system must be in Face Selection and Edge Selection mode. The reference entity acts as the pivot point or origin for the draft angle. Often, the parting line curve acts as the reference entity. The angle is calculated based on the depth axis of the current coordinate system; the depth axis is the zero reference. Positive angles draft the face out in the direction of the positive surface normal of the face, while negative angles draft the face in the negative direction of the surface normal.

Drafting with a face and edge shows a solid to which a draft angle is applied from the top edge and left angled face. The draft angle applied is 80° (measured from the depth axis, which is Z in this case). All measurements for applying a draft are absolute, not incremental.



Drafting with a face and edge

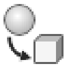
Drafting from a face and two edges shows the same solid, but in this case, the front top edges and the front face are selected for the draft. The draft angle applied is 20° (measured from the depth axis). Notice in this case, an additional edge and face have been created by the drafting function.



Drafting from a face and two edges

Slice

This function slices selected solids or sheets into separate entities. The slicing entity can either be the current CS or a selected sheet. When using a sheet as the slicing tool, the sheet must extend all the way through the target. If a solid and sheet are selected when this button is clicked, the body will be sliced into two separate bodies where the selected sheet intersects the body. Likewise, if two sheets are selected, the first sheet selected will be sliced where the second intersects the first. Slicing a solid with a sheet is a type of Boolean operation; therefore, the sheet will be destroyed or deleted once the slicing operation is complete. The slicing function also works only if a solid or sheet is selected. In that case, the solid or sheet will be sliced with the current coordinate system. It is recommended that slicing operations be performed as early in the modeling process as possible due to the fact that coordinate systems and planes act as very big (potentially infinite) knives when slicing and may unintentionally slice other entities.

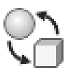


Replace

Substitutes a body with another body in any stage of the History list. You can replace atomic bodies or bodies that have been modified. The first body you select replaces the second body you select. You can then use the Rebuild function to update any affected body as necessary. This function is useful when you must make modifications to an object with a History list that contains imported or atomic bodies. You cannot use Replace on solids from the same tree.

To replace one body with another body:

1. Select the body you want to use as the replacement. If necessary, restore the body to the Workspace from its History list.
2. Select the body you want to replace.
3. Click the Replace button.



Swap

Switches two bodies in any stage of the tree. Select the two solids you want to swap. You can select the bodies in any order. Click the Swap button and then use the Rebuild function to update any affected solids as necessary. You cannot swap bodies from the same tree.

Add

The addition boolean operation provides for sheet to sheet and solid to solid combines. Adding two sheets produces a new single sheet composed of the two added sheets. The order of selection is not important when performing additions. Sheets must either be coincident or completely non-intersecting. Two surfaces are coincident when they overlap and all points on one surface also lie on the other surface within the area of overlap. Non-intersecting sheets and solids produce multi-lump sheets or solids.

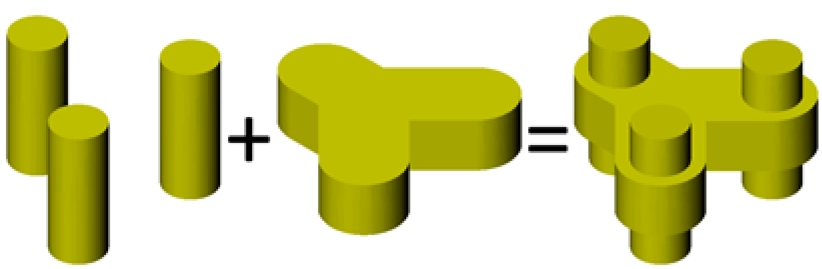


Figure 1: Solid + Solid addition

The figure below indicates the edges of the overlapped sheets.

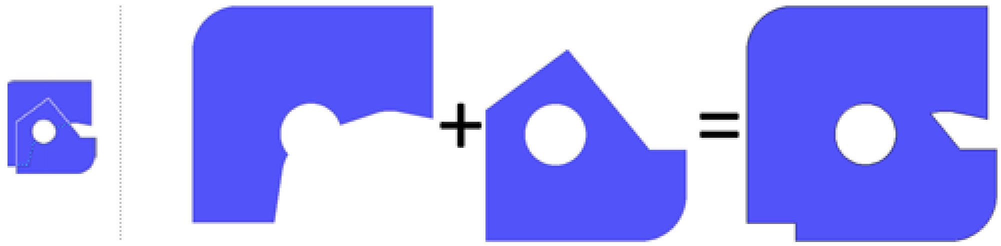
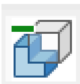
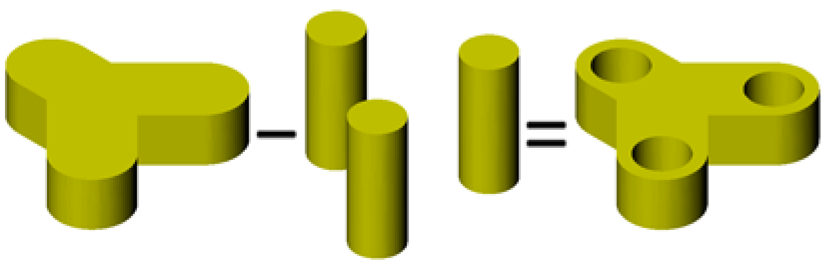


Figure 2: Sheet + Sheet addition.

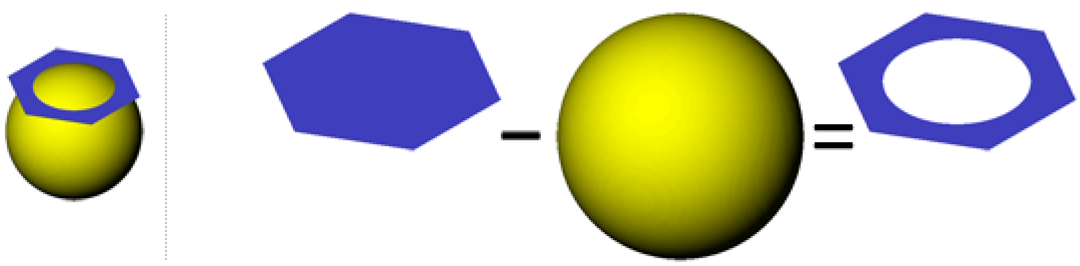


Subtract

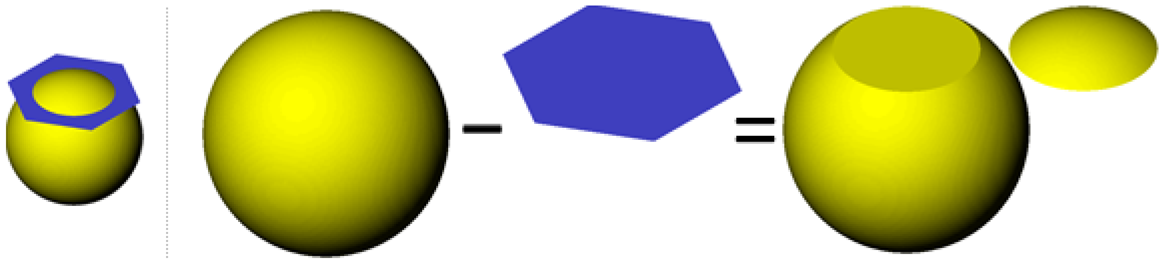
The subtraction boolean operation subtracts the common area of one body from another. The order in which you select the bodies is important because the second body you select is subtracted from the first body you select. The second body is deleted after the operation is complete. All bodies must be coincident or intersect in a way that would completely split the first selected sheet, or be non-intersecting. The following figures illustrate the different types of interactions between sheets and solids when the subtraction operation is performed.



Solid - Solid = a solid with the common volume of the second solid minus the first.



Sheet - Solid = a sheet trimmed by the solids boundary intersection of the sheet.

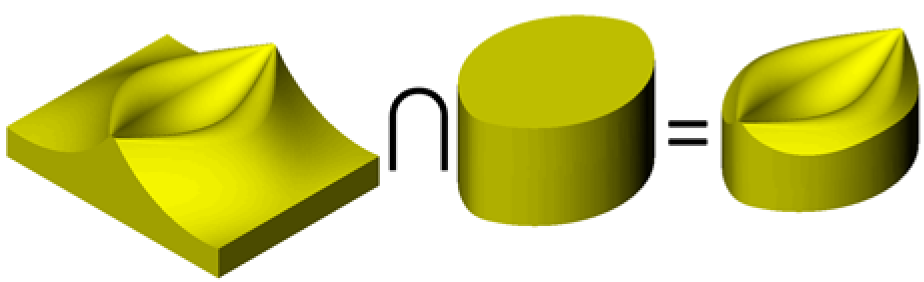


Solid - Sheet = a multi-lump body if separated becomes two individual bodies

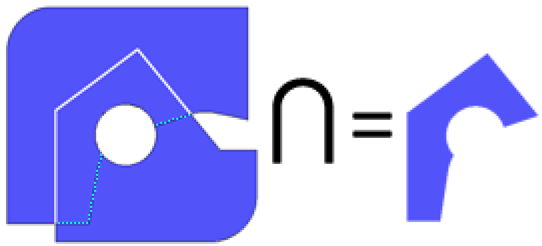
Sheet - Sheet = removing the common area of a sheet

Intersect

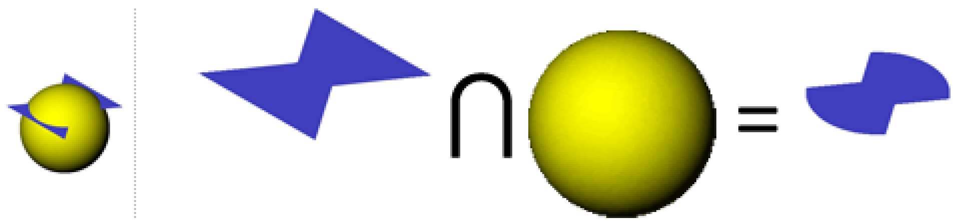
An intersection operation trims two bodies to the shared area between them in the workspace. Intersections can be made of any two bodies whether solid or sheet as illustrated in the following figures.



Solid intersect Solid = the common volume between the two bodies



Sheet intersect Sheet = the common area between the two sheets



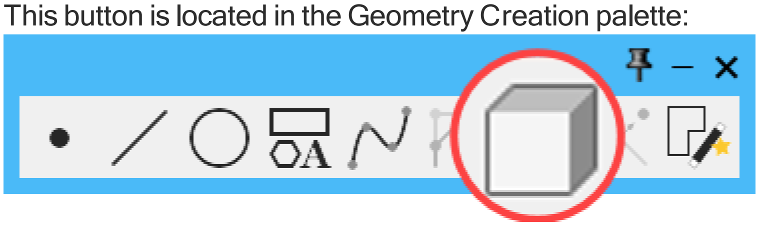
Sheet intersect Solid, resulting in a sheet trimmed by the solid

Separate

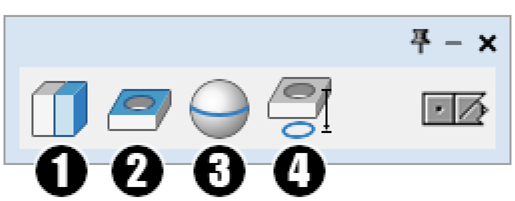
The separate operation divides multi-lump solids and sheets. That is, separate divides a multi-lump body into individual bodies. After you separate a multi-lump body, clicking on one of the bodies selects only the body you click on, instead of the entire multi-lump body.

Geometry Creation from Solids

This button is located in the Geometry Creation palette:



The palette that it accesses provides options for creating 2D geometry from solids and sheets. The options contained in the palette include: Geometry Extraction, Hole Extraction, Parting Line, and Outline. Each of these functions is briefly described below. For full information, see the *Geometry Creation* guide.



1. Geometry Extraction
2. Hole Extraction
3. Parting Line
4. Outline

Geometry Extraction:

Creates geometry from selected edges of solids and sheets. To view edges of a solid or sheet, the system must be in Edge Selection mode. Connected shapes are created if the selected edges create a closed loop. Typically, this function extracts the selected edges as splines or curves. However, if the resulting spline edge can be converted to lines or circles within the specified tolerance, the extracted geometry consists of lines and circles. A tolerance of zero is recommended when extracting geometry that is definitely a circle or a line.

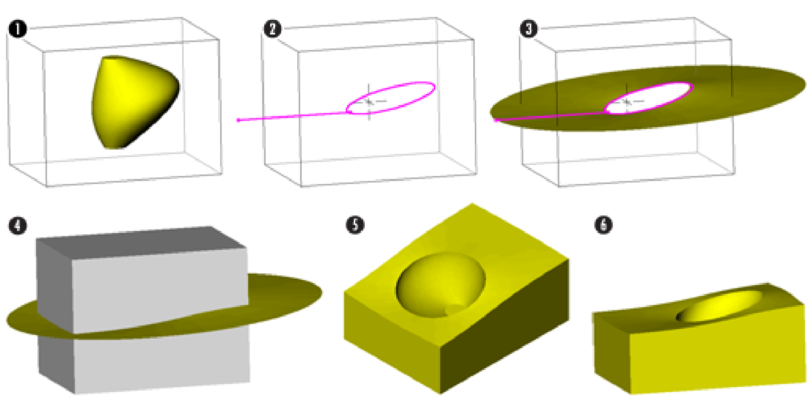
Hole Extraction:

Creates circles from holes in solids or sheets. You can extract circles from the existing holes on a model in order to have geometry to select for drilling operations. When using this function, either a solid or sheet can be selected. When hole extraction is performed, the resulting geometry will all be circles. The depth of the extracted geometry is based on the bottom of the hole(s), so that you can determine the depth for the drilling operation.

Parting Line:

Automates the creation of a parting line curve which can be used to create a parting line surface. To use this function, select all the faces that the parting line will be on or select the entire solid. The parting line function uses the depth axis of the current coordinate system as the draw axis. The draw axis is defined as the axis on which the mold is pulled apart. The parting line curve is the curve on which the surface normal vector is normal to the draw axis at every point. The parting line function creates geometry which can then be used to create a parting line surface. A good way to create a parting line surface from parting line geometry is to sweep a straight line along the parting line geometry, creating a sheet which will intersect the solid. The straight line is the drive curve and should intersect and slightly overlap the parting line geometry which is the base curve. Once the parting line surface is created, the part model can be subtracted from a cube to create the mold and then sliced with the parting line surface to create the two halves of the mold.

Example of creating a parting line surface to create a mold shows the process of creating a parting line surface by selecting a part model, creating parting line geometry, and then generating the mold.



1. Model
2. Parting line and Sweeping geometry
3. Parting line surface: swept shape
4. Mold block (with part model subtracted out)
5. Top half of mold cavity
6. Bottom half of mold cavity

Example of creating a parting line surface to create a mold

Outline:

This function will create geometry that is an outline of the selected faces on single or multiple solids and/or sheets. The geometry is created at a depth of 0 in the current CS.

History List

Understanding the History list can be crucial to extremely complicated modeling. The following information is intended to clarify the meaning of each icon and symbol within the History list. It is highly recommended that you name bodies when working with complex models.

Body Types

Atomic body:

An atomic or simple body is any body created in one operation such as those in the Create Solid palette.

Lump body:

A lumped or complex body is made up of two atomic bodies.

Multi-lump body:

A multi-lump body consists of at least two lumped bodies.

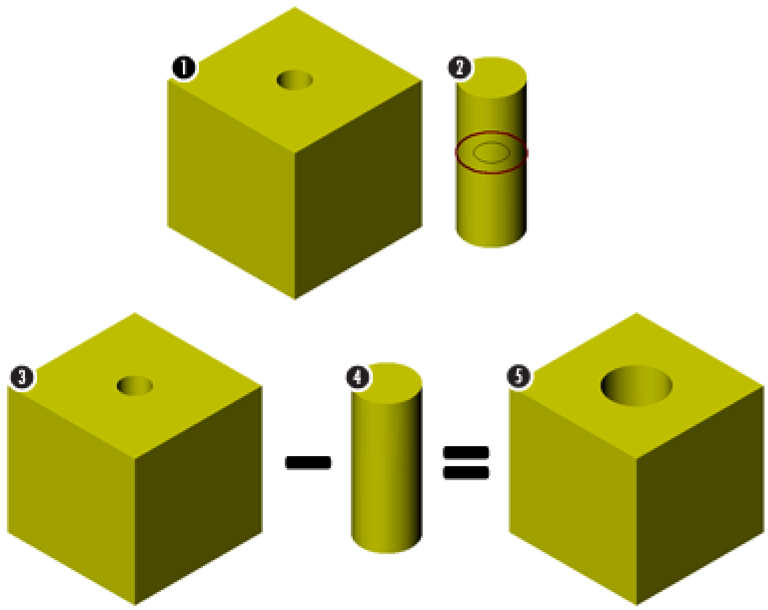
Rendered facet bodies, lump and multi-lump bodies contain a symbol on the icon to clarify what operation was performed in order to reach its state in the History list. The following is a list of characters that appear on lumped and multi-lump bodies.

Symbol	Function	Symbol	Function	Symbol	Function
+	addition	h	Stitch	t	Translate
–	Subtraction	i	Intersect	T	Duplicate And... Translate
–	Trim	k	Shrinkage	u	Untrim
|	Unstitch Sheet	m	Mirror	v	Variable Radius Round
/	Unstitch solid	M	Duplicate And... Mirror	w	Swept
!	Draft	o	Solidify	x	Explode
b	Blending Round	r	2d Rotate	X	Extract
c	Chamfer	R	Duplicate And... 2D Rotate	none	Absolute Rotate or Translate

Symbol	Function	Symbol	Function	Symbol	Function
f	Offset or Shell	s	Slice	none	
$	Facet Body				

Body Names

The history names can provide information on how bodies were created. There are three operations that combine two solids: Addition, Subtraction, and Intersection. When these operations are performed, the name of the history item indicates the operation by combining the two names and placing a character to separate them. So a name such as "Cube1-Extrude2" in the image shown, means an extruded body was subtracted from a cube. A name with "Cube1+Extrude2" indicates addition while "Cube1^Extrude2" represents an intersection.



Modifying, Recreating, and Rebuilding Bodies

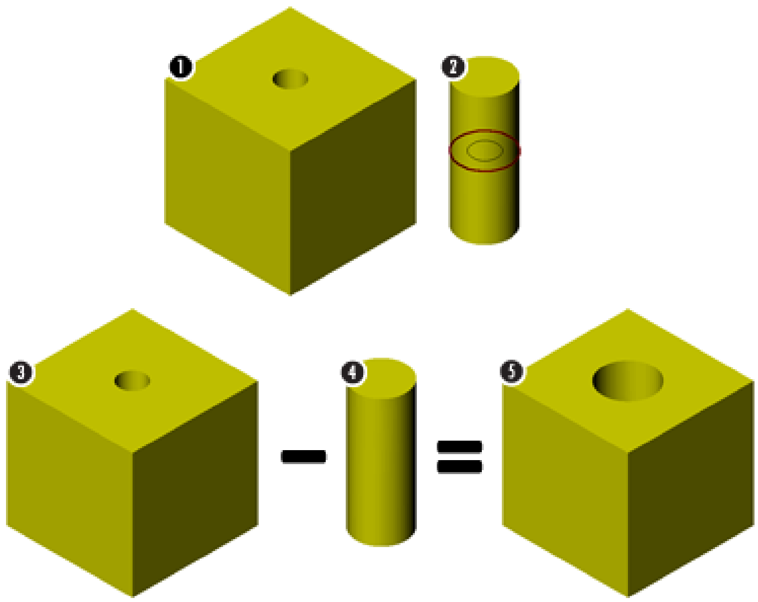
In general, there are four different methods for making modifications to bodies.

- Create a completely new solid (component) and manually recreate the final part to include the new component.
- Make a modification to an existing solid using one of the modeling functions, such as slice or offset, and manually recreate the final part including the modified component.
- Modify and/or recreate the component and then use the Replace Solid and Swap Solids functions to replace or swap the old component with the new one, and then use the Rebuild function to get the final solid.
- Bring back the solid that needs to be modified from the History list. Modify that solid using the Recreate function, and then use the Rebuild function on the final part to incorporate the recreated solid into the final part. The History, Recreate and Rebuild functions are all accessed from the body context menu.

The following examples provide a practical application of each of these methods. The final part model is a cube with a cylinder subtracted from the middle of it, creating a hole. The necessary modification is to enlarge the hole. Each of the methods described above for modifying a solid will be applied to facilitate the necessary change. See Method 1: Create a New Solid through Method 4: History, Recreate, and Rebuild.

Method 1: Create a New Solid

For this method, a new cylinder is extruded from a new circle with a larger radius. That new extrusion is then subtracted (Boolean operation) from a cube to create the final part, a cube with a larger hole.



1. Original model needs larger hole
2. New extrusion
3. Original cube
4. Larger cylinder
5. New model with larger hole

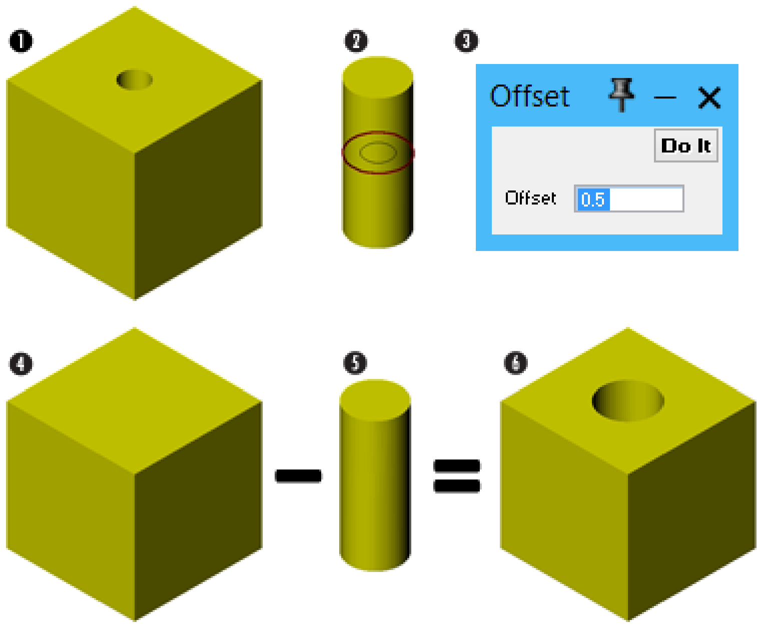
Creating a new solid to modify an existing solid

Method 2: “Locally” Edit an Existing Solid

You can edit certain faces of a solid without using the Boolean operations. Examples of functions which locally edit a solid are the offset function applied to selected faces of a solid or the Solid Unstitch function used to remove particular faces to “heal” a solid.

In this example, you can locally edit the cylinder that was initially used to create the part by offsetting the outside face. The original cylinder may be in the Body Bag or can be retrieved from the History list. To make the cylinder larger, you can offset the outside face of the cylinder by a given amount, effectively making the cylinder larger in diameter.

In this case, we did not create a new solid, but instead modified an existing solid using the Offset function. When a solid is modified using modeling functions, the name and reference of the solid are changed to signify the change that was made. According to the system, the modified solid is a completely new entity that has a new reference identity. For instance, in this example the original cylinder was named “Extrude#”. When the offset operation is performed, the new solid is named “Offset#”. The original solid labeled “Extrude#” still exists in the History list of this model. The cylinder with the offset faces could then be subtracted from the cube.

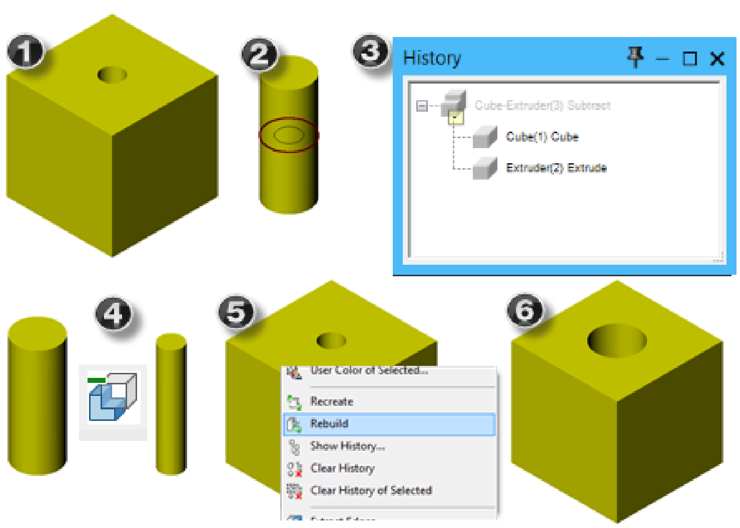


1. Original model needs a larger hole
2. Original cylinder
3. Value by which to offset the cylinder
4. Original cube
5. Offset cylinder
6. New model with larger hole

Editing an existing solid to modify the end result

Method 3: Replace/Swap and Rebuild

In this example, you create the new extrusion with the larger diameter and then replace the old smaller extrusion with the new larger extrusion by using Replace. Replace substitutes a solid with any other solid in any stage of the tree. Then, you use the Rebuild function to incorporate the new larger extrusion into the cuboid and generate the modified solid.

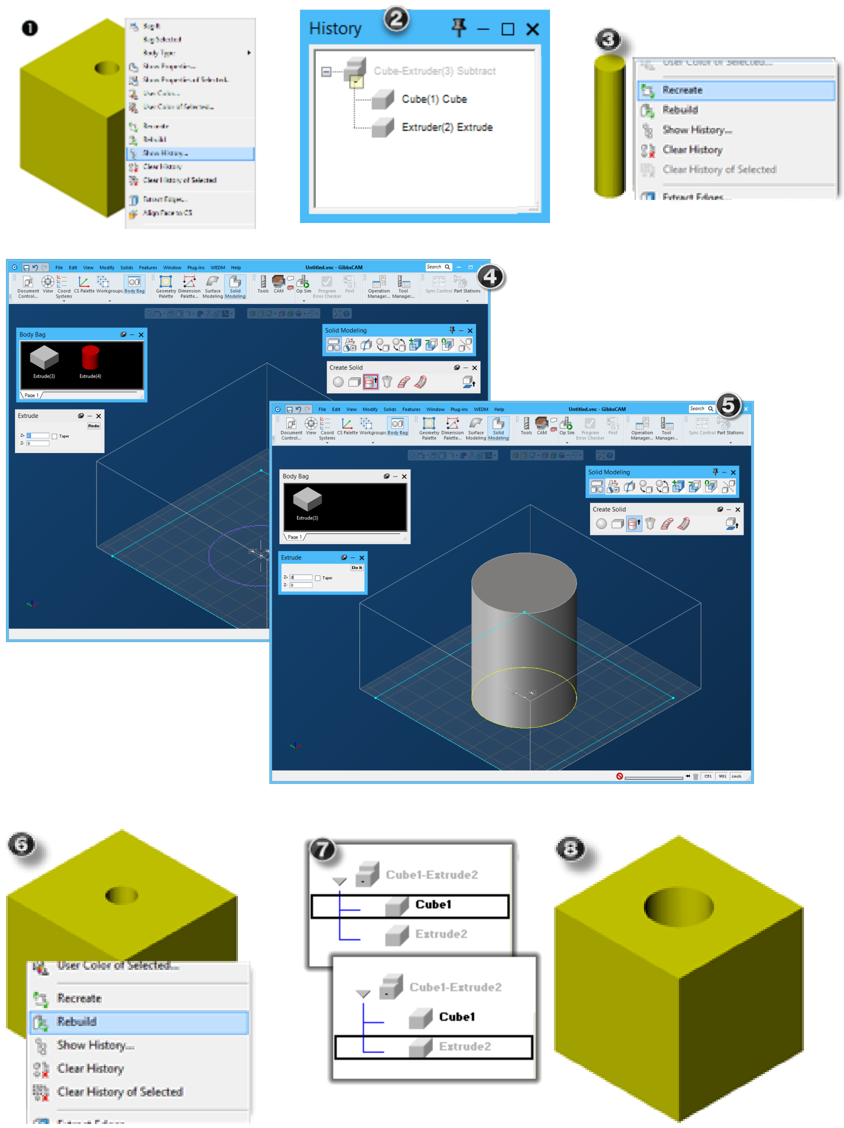


1. Original model needs a larger hole
2. New extrusion
3. Original extrusion is extracted from History list
4. History is Replaced by the new Extrusion
5. The model is Rebuilt
6. The new model

Using Replace and Rebuild to modify a solid.

Method 4: History, Recreate, and Rebuild

You use the History, Recreate, and Rebuild functions to make the necessary changes. The History lists maintain all of the bodies that are used to create any model. Any time a solid is created or modified, it is assigned a name and reference. If a modification is made to an existing solid, a new solid with a new reference is created. The Recreate function is the only exception to this; it allows you to make modifications to an existing solid without creating a new solid. Recreate changes the existing solid while maintaining the original name and reference. The solid that was recreated now exists in the History list, while the original solid that was modified no longer exists—it has effectively been deleted from the system. It cannot be retrieved. The Rebuild function simply reprocesses the History list of a model. The only way to change a model using the Rebuild function is by making modifications to a solid in the history of that model using the Recreate function.



1. History list is opened.
2. The body is extracted.
3. Recreate function is accessed.
4. The dialogs and geometry that created the cylinder are activated.
5. New geometry is created and extruded.
6. Rebuild the changes.
7. The History list is reprocessed.
8. The model after being rebuilt.

Modifying solid using recreate and rebuild.

Tips and Techniques

- Avoid congruent face modeling

  The system attempts to solve co-planar congruency problems (where the faces that are congruent are planes) and is often successful. However, as a general rule you should avoid Boolean operations with congruent faces. When possible, adjust one of the bodies (by offsetting faces, etc.) so as to not have congruent faces. A good practice is to always try to overlap bodies when possible.

- Avoid co-edge modeling

  There must be exactly two faces per edge in a solid.

- Slice simple solids

  A CS or plane acts as a big knife and slices every solid that it intersects. For purposes of accuracy, it is better to perform slicing operations on a simple solid.

- Blend corners last

  Rounded bodies have more faces, which can slow down several of the functions. Another reason to round last involves the Rebuild function. Bodies that have rounded edges can be rebuilt. However, there cannot be any significant changes to the topology in order for the rebuild to work.

- Do not round fillets that can be left by a tool

  The intersection surfacing option is designed to machine edges at the intersection of two surfaces. The intersection process can only be applied to edges that are not blended. So, if you want to create a necessary fillet using the radius of a ball or bullnose endmill, do not create the fillets on the part model.

- Minimize generations

  Be careful with Modify operations, such as Duplicate And..., as each time one is performed a new body is created. A good practice is to think carefully about the modeling operations in order to reduce the file size and make all future modeling more efficient. Another way to minimize generations of solids is to intersect bodies rather than perform two subtraction operations.

- Name your body

  Bodies should be given distinct, descriptive names to avoid confusion. You can name bodies using the Properties dialog or by changing the icon name when the solids are in the Body Bag.

- Minimize the use of non-destructive Boolean operations.

  Promote the use of the History list. By having fewer bodies, the file size is smaller and processing time is faster.

- Deselect the Body Bag items

If a body in the Body Bag is selected, it may be accidentally deleted. To avoid accidental deletion, keep the Body Bag closed when not in use.

- Use the Un-Bag It function for small items in the Body Bag

  Often a body is so small it cannot be seen when the Body Bag renders it to scale. The easiest way to select these small objects is to `right-click` the name and choose Un-Bag It from the context menu.

- Bag and Un-Bag many items in the Body Bag at once

  The title bar context menu for the Body Bag contains two important bagging techniques for bagging multiple selected items, Bag Selected and Un-Bag Selected, which are especially useful for surface file imports.

About Machining Solids

Introduction to 3-Axis Machining

This section contains reference information on the multi-surface machining functions in the system. The first part describes some of the terms and concepts for using the multi-surface machining capabilities. For information on the standard machining functions provided in the Production Milling module, see Mill.

Gen 3 Engine

GibbsCAM uses the Gen 3 solids toolpath engine. The Gen 3 engine provides many improvements over Gen 2, including smoother and more optimized toolpath for contouring operations. Gen 3 is optimized to produce toolpath consisting of lines and arcs.

Compatibility With Earlier Versions

If you open a part that was created in an older version or if a part needs to be saved to an earlier version, the Gen 3 Engine automatically converts toolpath to or from older versions of the system.

Older to new version

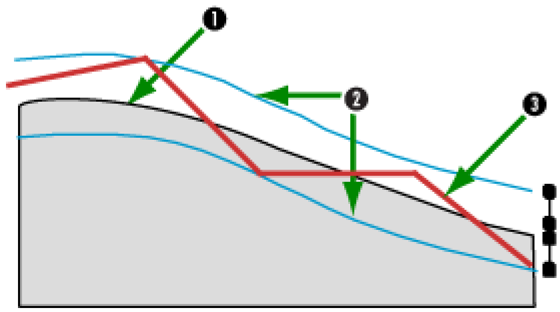
Data imported from earlier versions of the system does not have all of the functionality of the Gen 3 engine and the new features. To accommodate this, the process is rebuilt with whatever data can be gathered from the old process and the new options use default values.

New file being saved to an older version

When saving to an earlier version, all of the new functionality is lost, but valid toolpath is still generated. Saving back to versions 5.1 through 6.1 uses Gen 2 and even Gen 1, where appropriate. Saving back to versions prior to 5.1 uses the Gen 1 engine exclusively. Saving a file to prior versions does not result in identical toolpath, but the toolpath is valid.

Surface Tolerance

The Surface Tolerance affects how closely the toolpath approximates the surface to be machined. The tolerance value specifies an amount that the toolpath can deviate from the actual surface, either on the inside or the outside. The example illustrates a valid toolpath and displays the surface being machined and the surface tolerance region.



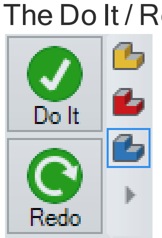
1. Machined Surface
2. Surface Tolerance
3. Toolpath

Because the toolpath can cut on the “inside” of the surface by the tolerance amount, specifying a surface stock greater than the tolerance amount ensures that the surface is not gouged by the toolpath.

The smaller the surface tolerance, the more closely the toolpath follows the actual surface. Coarser tolerances provide increased performance at the expense of processing time. It is recommended that coarser (larger) tolerance values be used on roughing operations and tighter (smaller) tolerance values be used on finishing operations to reduce processing time and the length of the generated code.

Selection Modes: Part, Constraint (Fixture), Stock

The Do It / Redo control contains three small buttons that specify selection modes.



1. Part
2. Constraint
3. Stock

Machining Selections:

You use the Part, Constraint, and Stock buttons when selecting the cut shape, constraints and stock for a set of processes. You can select what to cut, what not to cut, and what to use as the stock condition for each Process Group.

The Part button is selected by default. Any selections you make while the Part button is selected are used as the cut shape for the process list. All machining processes in the process list are applied to the selected cut shape, such as a solid, sheet, contour, and so forth.

You click the Constraint button to select solids, sheets, or faces as constraints for processes. Any bodies, faces, or sheets selected are not cut. By default, any unselected faces of a solid being machined are considered constraint faces and are not machined. Click the Stock button to designate solids and sheets as local stock, which means that the selected body acts as the starting stock condition only for the current process list.

Stock Definition

The stock shape is used to create machining operations and display the cut part rendered image of the part once those operations have been generated. Stock that defines a part may be set in one of three ways.

Workspace Stock

This is the initial set of values specified for every part in the Document Control dialog. This method of stock definition can be overridden by other methods; however, these values still define the Workspace and the area used by the Unzoom command.

Part Stock

Geometry in a Workgroup may be used to define the initial material condition. The shape may be extruded or revolved and may contain a single hole. A stock workgroup will override the stock cuboid as the initial material. There should only be one stock workgroup, as additional instances will be ignored.

Stock Body

The SolidSurfacer option allows any solid or sheet to be a designated as stock. The Properties dialog contains options that allow users to specify that a particular body is either a Part, Stock, or Fixture. Selecting the Stock option will cause the selected body to act as the initial stock condition for the part. This stock condition will be used for machining operations as well as in cut part rendering. This is considered a global stock specification as it will be used for the entire part. A stock body must completely enclose any bodies selected to be machined. This will override any workgroup or stock cuboid definitions. Only one stock body will be used, although it may be a multi-lump body.

Temporary Stock

A body may be defined as stock for a single set of processes. This will temporarily override any of the above three stock definitions. Create a temporary stock body by selecting the desired bodies to define the stock and press the stock button in the Machining palette. This can be useful for defining stock smaller than the part and to restrict the area being machined for a single set of processes. For roughing, the stock and part loops created at a single Z level/slice should not intersect.

Notes

Operation Stock Size

To generate proper toolpath, the workspace stock should encompass the part. The stock tolerance, part plus surface stock, is included in the stock condition within the workspace stock size for operations. Error messages may appear if the operation is invalid because the Surface Stock Allowance extends beyond the workspace stock size. Problems may occur when attempting to calculate remaining material.

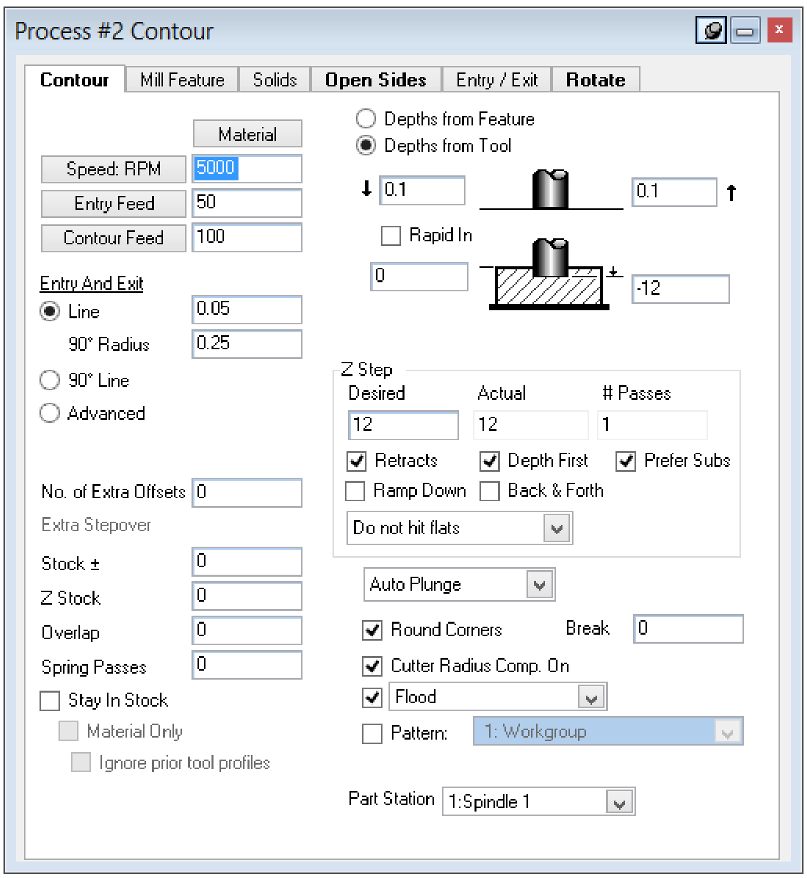
Fixtures

Roughing and contouring operations differ when working with fixtures. Roughing operations move around a fixture while contouring operations retract over fixtures.

Contour & Rough Machining

Contouring Process

Contouring operations are designed to take a single finishing pass along a selected cut shape. The cut shape for a contouring operation can be a solid, a sheet, selected faces of a solid or sheet, a contour (connected 2D shape) or some combination of the above.



- Selecting only a 2D contour will create a single pass around the selected shape based on the machining markers. This is no different than standard 2D machining.
- Selecting a 2D contour and a solid (or sheet) for the cut shape will create a toolpath based on the contour selected and the machining markers. That toolpath will then be projected up in Z

onto any selected body. The Z moves of the toolpath will only be modified where it would have gone into the body.

- Selecting only a body for the cut shape will create a toolpath that will take a single pass around the surfaces of the selected body at the specified Z depths. The system determines the contour based on the solid (or sheet) selected.

Using the Profiler

You can use the profiler to select faces for machining. You can use the Profiler to set machining markers for a Contour operation, like geometry. You can extend the start and end features. The profiler is automatically activated when you double-click an operation that uses the profiler. Machining sheets using this method is not supported.

Note: Extended moves are not gouge protected.

To extract a profile as geometry:

1. Select the CS you want to use.
2. Click Toggle Profiler to enable. A faint green grid appears parallel to the current CS.
3. As necessary, right-click the grid and set the Profile Depth to the position you want.
4. Select one or more bodies.
5. Right-click the Profiler grid and select one of the options: Select All Profiles, Select Faces From Selected Profiles, or Select Faces Inside Selected Profiles.

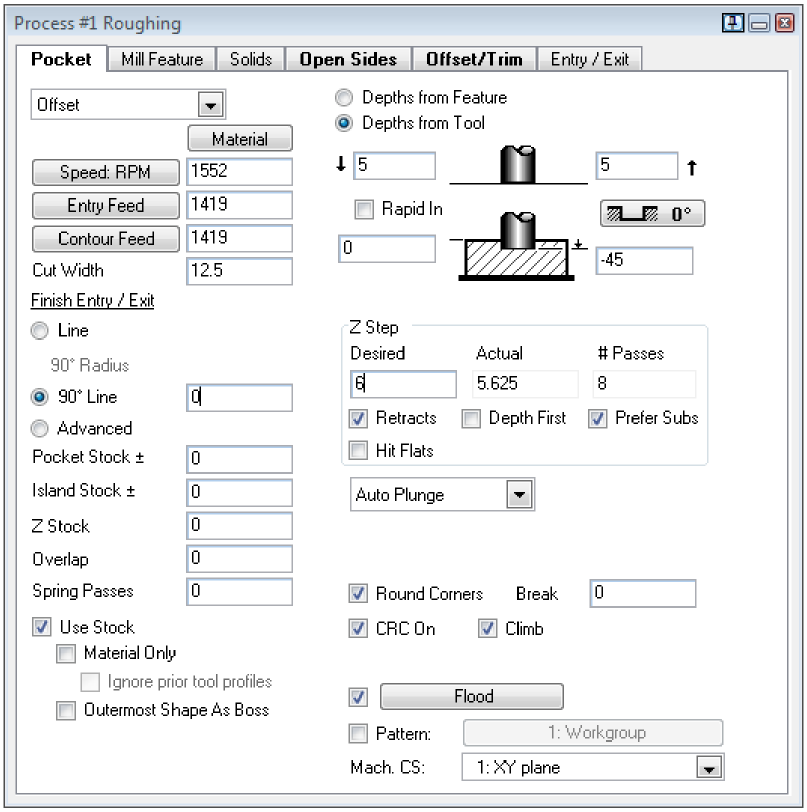
   A profile or profiles are generated and highlighted in blue.

6. Right-click the Profiler grid and click Extract.
7. In the Geometry Extraction dialog, provide a value for Tolerance and then click Do It.

   The profile is extracted as geometry.

Roughing Process

The Roughing process creates offset, zig zag, and face milling routines designed to remove material quickly. Cut shape selection for roughing is very similar to contouring.



- Selecting a closed 2D contour for the cut shape will create a roughing routine to remove material from the inside of the selected closed shape. This is no different than standard 2D machining.
- Selecting a 2D contour and a solid (or sheet) for the cut shape will create a toolpath based on the selected shape. That toolpath will then be projected down in Z onto the body. The Z moves of the toolpath will only be modified where it would have gone into the selected solid (or sheet).
- Selecting a body for the cut shape will create toolpath that will pocket out the body (or face) from the stock. The stock is used as the outer shape for the pocket.
- Individual faces on a model can also be selected for the cut shape; this allows for individual pockets to be machined. To machine selected pockets, select the bottom face of the pocket for the cut shape.

The toolpath at the final Z depth (specified by the floor Z) is calculated first. Each pass will be calculated from that depth and moved up in Z by the specified Z step amount. If the pass at the floor Z depth cuts into a selected solid or sheet, that pass will not be created, and the next pass (a step above) will be the final pass. The system will continue creating steps up in Z until the surface Z level is encountered. No pass will be created above the surface Z.

When Use Stock is unchecked, the stock definition is ignored. Roughing a solid will machine all selected faces, meaning that a pocket can be machined by simply selecting the floor (if the pocket's floor is flat).

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. Note that any passes above the stock will be omitted but passes below the stock will still be generated to the final Z depth.

If a fully-selected solid is being roughed, the tool will machine inward from the stock definition to remove material. The term “fully-selected” refers to all the faces the tool can see being selected. This does not include faces on the backside. A partially-selected solid will not use the stock to create a larger area to rough but will trim the pocket to stay inside of the stock definition.

Material Only

Material Only calculates toolpath for all remaining material left on walls 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 may be used as a single operation or as part of a multiple process group.

When the Material Only option is selected, the system tracks the areas where material is left during an operation by creating closed shapes with both “wall” and “air” features or a combination shape for each occurrence of remaining material. During subsequent operations, the system generates toolpath to remove only the material within these shapes. Toolpath generated in these areas is based upon an open-sided pocket configuration.

Machining preferences

The Allow Mill Material Only checkbox in the Machining Prefs tab of the Preferences must be active in order to track and store the condition of remaining materials. It is strongly recommended that this option be deselected if the Material Only option is not going to be used in operations.

When this option is active, the system performs the necessary calculations for a Material Only operation, even if the calculations are not applied. This information is also saved with the part file.

Material Only Pockets

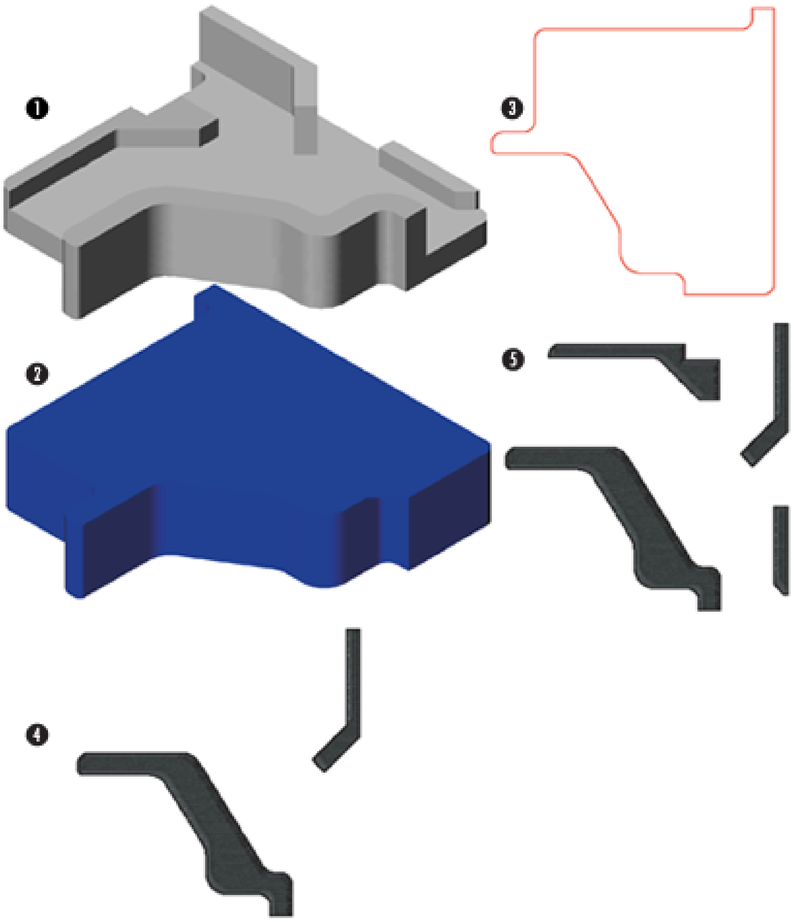
There are two different types of pockets when calculating toolpath for Material Only machining operations–closed and open. When generating toolpath for a solid that has closed and/or open pockets, SolidSurfacer uses the Multiple Shapes method described below. For more information on Material Only and cutting geometry, see the *Mill* guide.

Multiple Shapes Method:

This is the recommended method for assuring the best toolpath when generating toolpath for Material Only machining. This method requires at least two shapes. The first is an all “air” shape, which represents the stock, and another shape representing the pocket as an island. This second shape is an all “wall” shape. Using this method, the system treats the pocket as an island inside the stock.

To generate these shapes, SolidSurfacer performs a horizontal slice of the solid at each Z-level cut depth defined in the process dialog box. The all “air” shape is based on the stock condition at each Z-level step and the all “wall” shape(s) is based on the part condition at each Z-level step.

The following part shows what the “air” and “wall” shapes would look like at two different Z-level steps for the part. The part consists of a floor at Z0 and four walls with the tallest at Z25.



1. Part model
2. Custom Stock
3. Air geometry for stock
4. Material at Z20
5. Material at Z10

Material Only—Multiple Shapes - air and wall shapes

Optimizing Material Only for Solids

- Avoid full solid selections. Only select the area (faces) to be cut.
- Use Create 2D toolpath. This produces better toolpath (not just G1s but G2s and G3s) and also allows for a tighter Surface Tolerance setting. Avoid undercuts when using the Create 2D toolpath feature.
- Select Ignore Tool Profile when permissible. For more information on Ignore Tool Profile, see the *Mill* guide.

Material Only Limitations:

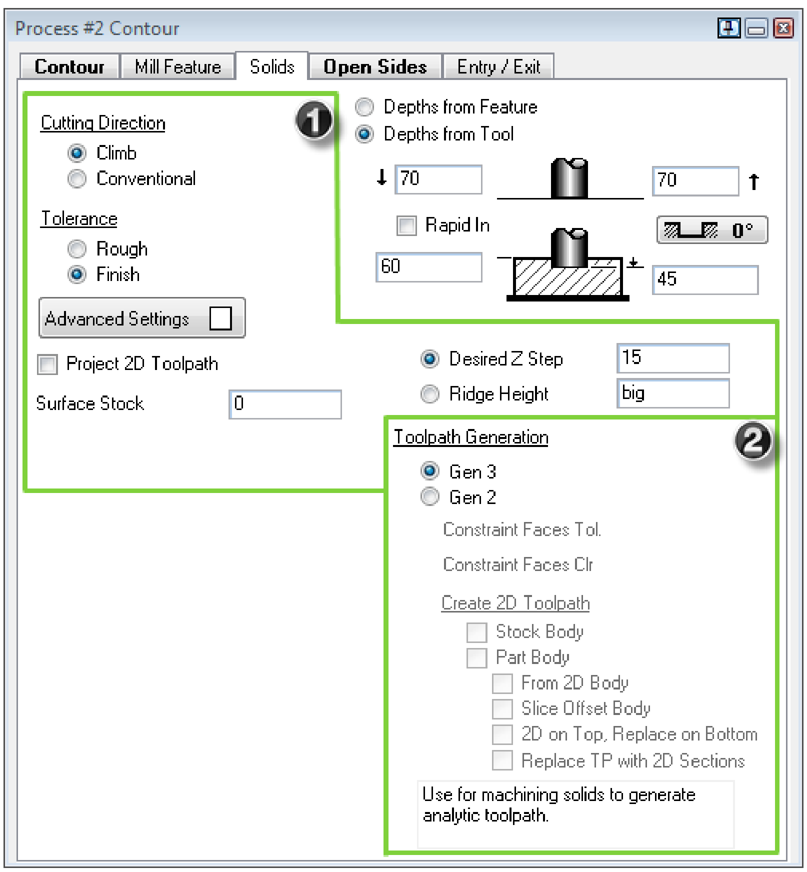
- Undercutting tools
- Custom Stock with undercuts
- Depth First

Troubleshooting

- If wasteful toolpath is generated, the Past Stock value for this operation may be too large. The recommended value for Past Stock is the tool diameter minus 2.5 times the maximum surface tolerance of the previous operation.
- If no toolpath is generated, the final cut depth may be below the stock bottom. Redefine the stock definition for this operation, then move the stock bottom to the desired final cut's Z-depth.
- If all else fails, extract edge geometry and machine as geometry. When extracting the edge geometry, specify a small tolerance so that the edges will be extracted as lines, arcs, and circles (analytics). Then use the Multiple Shapes method described in Material Only–Multiple Shapes - air and wall shapes.

Solids Tab

The Contouring and Roughing process dialogs have a Solids tab which contains information specific to machining solids and sheets. The tab is bolded when a solid is selected and the settings found here will only be effective if a solid is used in the operation.



1. Toolpath Control
2. Toolpath Generation

Toolpath Control

Cutting Direction

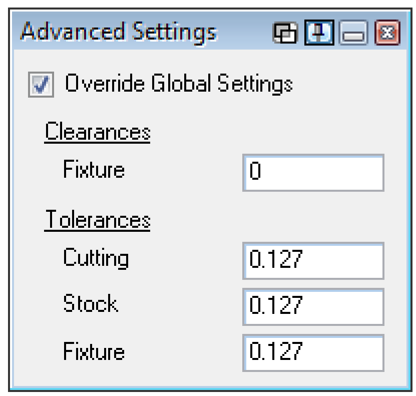
Note that the user must select the Cutting Direction only in the Contouring Process dialog. The selection made for the cutting direction determines whether the tool will climb cut or conventional cut during a contouring operation. When geometry, profile or solid is selected for a contouring operation, machining markers appear on the selected geometry, allowing the user to indicate the direction of the cut by selecting the appropriate arrow. If the cut direction is indicated with the machining marker arrows, the setting for the cutting direction contained in the Contouring dialog will match the selection indicated by the arrows. Likewise, if a selection is made for the cutting direction, the machining markers will be updated to match that selection. One does not supersede the other; the system uses the last selection made before the operation is processed. These options are especially useful when only a solid or sheet is selected for a contouring operation, because when that is the case, machining markers do not come up on the screen, providing the user with a method to designate the direction of the cut.

Tolerance

When Use Global Settings for Solids is checked in the Document Control dialog, use these radio buttons to toggle between a Rough and Finish tolerance (applicable only to the specific process). Using this setting speeds up toolpath and minimizes G-code.

Advanced Settings

See “Advanced Settings” on page 95.



Use the Advanced Settings to override the tolerances set in the Document Control dialog on a process-by-process basis. Click the Advanced Settings button to access the Advanced Settings dialog and then select the Override Global Settings checkbox to apply the clearance and tolerance values to the process. A blue checkmark will appear on the Advanced Settings button if the global settings are being overridden.

Clearances

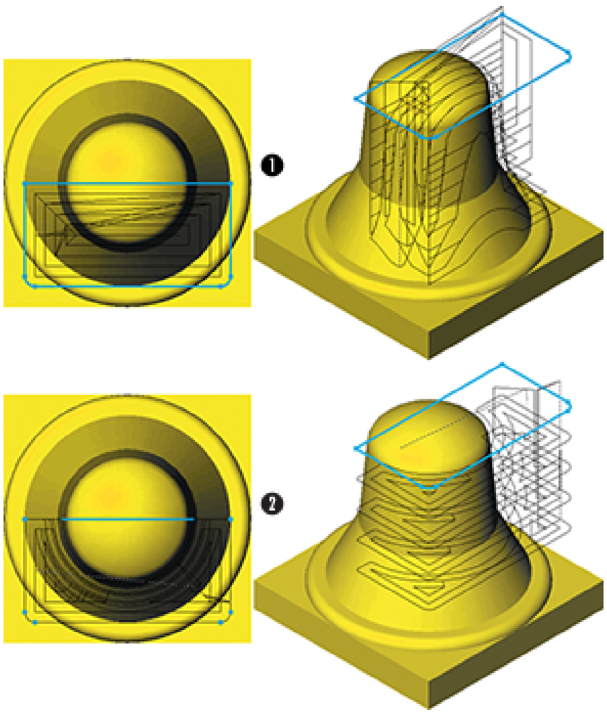
This section allows the user to set the interaction between toolpath and fixtures that are to be avoided. A fixture may be defined as a sheet or a solid designated as a fixture. There is a text box for the clearance value from a Fixture. This value is the additional distance the toolpath will be offset from the object.

Tolerances

These are the machining tolerances for the toolpath, or the margin of error. The toolpath may deviate by up to these amounts. A looser tolerance will require less memory and creates shorter output. To provide as much flexibility as possible, there are separate settings for Cutting, Stock, and `Fixture`. The Cutting tolerance is the tolerance of the toolpath over the selected face or faces—the area to be cut. The Stock tolerance is the accuracy of the toolpath's interaction with the stock definition. The Fixture tolerance specifies the accuracy of the toolpath's interaction with areas that are to be avoided. The default value for all selections is 0.005" or 0.127mm.

Project 2D Toolpath

The system will trim toolpath to a specific area when a solid and 2D geometry are selected for contouring. The toolpath will be bound within the selected geometry and will not go beyond the bounds of the stock if the geometry overlaps the defined stock. The behavior of the toolpath within the geometry is optional based on the Project 2D Toolpath option.



1. Project 2D Toolpath is enabled
2. Project 2D Toolpath is disabled

Example of using Project 2D toolpath.

When this option is disabled, selected geometry acts as a boundary that the toolpath will not cross. The tool will take successive 2D passes in Z, using the solid as a shape to follow and the geometry as a boundary. When this option is on, the toolpath will be projected over the solid, creating 3D toolpath while following the shape of the geometry (that is, the tool will take a pass around the geometry). If viewed from the top, the toolpath will look like toolpath for a regular 2D pocket. If viewed from another angle, the difference is apparent. By projecting the toolpath, an adequate finish is left on the part and the tool always moves in the same direction. However, this toolpath creates extra cut time and can re-machine the surface of the part on multiple passes. An example of using Project 2D Toolpath is illustrated in Example of using Project 2D toolpath..

Surface Stock

The Surface Stock setting 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. The Stock± amount entered in the Contour tab only adds stock in the cutting plane (machining CS X,Y). If both Stock± and Surface Stock are entered, they will be added together; one does not override the other. Surface Stock can be less negative up to -0.00005 less than the corner radius of the tool.

Z Step

If Desired Z Step is selected, the step in Z will be constant based on the value entered. The Ridge Height selection will create variable steps in Z resulting in a uniform ridge height on the cut part, generating a smoother finish on the part. The Ridge Height (also called scallop height) is calculated from the tool's corner radius cutting a flat surface. It is an approximate value.

Toolpath Generation

Use these radio buttons to toggle between using the Gen 3 or the Gen 2 Engine. The system is set to use Gen 3 by default for contouring operations. You must specify a tolerance for the constraint faces as well as the Create 2D Toolpath settings if using Gen 2.

Constraint Faces Tolerance
: This value specifies the tolerance for constraint faces. Note that this value should be smaller than the Constraint Faces Clearance value to avoid gouging.

Constraint Faces Clearance
: This value specifies the clearance for constraint faces, or the distance by which you want tools to clear these faces.

Create 2D Toolpath

The purpose of the Create 2D Toolpath settings is to produce toolpath from what might otherwise be a 3D toolpath. The system has multiple options on how to achieve this toolpath for contouring operations. This allows more control over the results of toolpath generation.

The term “2D toolpath” is used to identify a toolpath of the type desired for machining a 2D prismatic part and “3D toolpath” to identify a toolpath typical of machining a complex surface. Strictly speaking, the system’s 3D toolpath methods frequently produce toolpaths that are mathematically 2D, as they only move in X and Y. These are not, however, optimal for the machining of 2D prismatic parts. “Prismatic” refers to parts that can be constructed by extruding XY shapes along the Z-axis.

A 2D toolpath contains lines and arcs and does not vary with surface tolerance. A 3D toolpath is usually a large number of small line moves that vary from the true surfaces by the surface tolerance. A 3D toolpath is created when solids and surfaces are being machined.

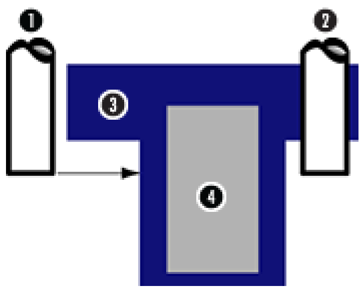
Create 2D toolpath is useful when a solid or single surface is being machined and most useful when the solid being machined has 2D elements such as planes and cylinders. The selected faces of a feature must be stitched together into a single surface. Create 2D toolpath is recommended primarily for use on solids, but if sheets must be machined, then they should only be used to machine a collection of surfaces if each surface is a single feature (for example, if each of the surfaces was a single pocket).

None of the choices available from Create 2D toolpath will approximate complex surfaces with arc moves. Any of the options may fail to produce a toolpath. This is why multiple choices may be selected. They are attempted in the order in which they are listed. When one fails, the next is attempted. If they all fail, a 3D toolpath is created. An informational message box will appear listing the status of each of the 2D methods attempted. Even if a method creates a toolpath, it may nevertheless be an invalid toolpath. Several of these methods have protection limitations that are different from the standard 3D toolpath. For more information, see “Limitations of Create 2D Toolpath” on page 91.

Stock Body:

By activating this option, the system will attempt to create 2D toolpath from a stock body for the outermost loops of a roughing operation. The 2D toolpath can come from geometry, solids, and stock definitions. Solid stock body definitions do not inherently produce 2D toolpath but instead produce a large number of small line moves. Selecting the Stock Body option will apply the Slice Offset Body only to the stock body. Since the stock can be used as the outer loop of pocket, a 2D toolpath here will improve all the roughing passes in a pocket. This function will produce better toolpath with 2D and 2.5D prismatic shapes, and will work better with the Material Only option.

There is no undercut protection if the Create 2D Toolpath option is selected for stock. If a stock definition gets smaller as you go down in -D, then the area to be machined at D= -2 can be smaller than the area at D= -1. We only see the area at the level being machined; this can lead to a rapid Z move into an area we think is clear, only to discover that uncut material from a higher level is bigger, resulting in a crash. To avoid this, you can either avoid using the Create 2D Toolpath options for your stock, or else you must be sure to visually check your entry plunge plunge moves.



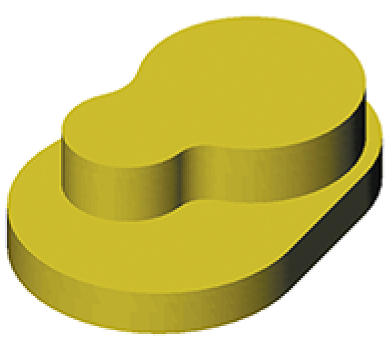
1. Clear Entry with undercut protection
2. Clear Entry without undercut warning
3. Stock
4. Part

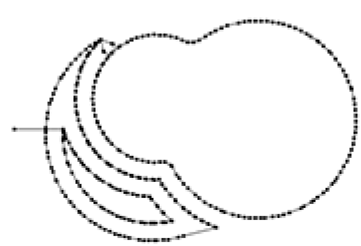
Undercut protection in a T-shaped part

Part Body:

This option allows the system to generate optimized toolpath based on the selected body. There are four Part Body options for how the toolpath is generated. Any combination of these options may be selected. The system will start with the simplest, quickest one, try to generate toolpath, then move down the list of selected items to the next option if the current option fails. When this option is inactive, the system produces 3D toolpath from all solids.

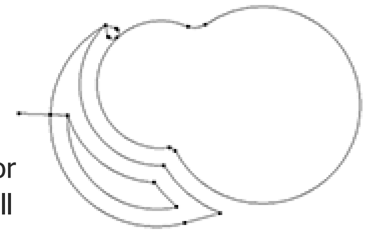
The model below is ideally suited for 2D toolpath. The next four examples will show the toolpath generated using the various Part Body options on the same model. Without Create 2D Toolpath, standard 3D toolpath is created as shown. A similar image will be used for each of the four options.





Model that is ideal for 2D toolpath	Standard 3D Toolpath

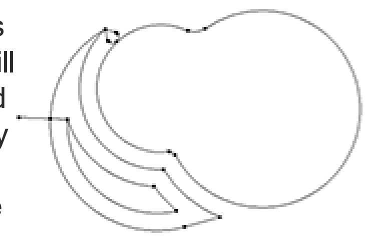
From 2D Body:



This option will generate 2D toolpath (lines and circles) without the surface tolerance deviation, provided that all of the faces selected are 2D. It makes high-quality 2D toolpath very quickly. Note that a horizontal chamfer or fillet is not 2D.

In order for the From 2D Body option to work, either the entire body or all faces of a pocket must be selected. No passes above the part will be generated and all Z steps will be uniform—there will not be a variable step. The From 2D Body has limited undercut protection, no constraint face protection, no fixture protection and can fail due to complex face edges. If a partial body selection is made, (faces are selected instead of the entire body), it is recommended that Use Stock be turned off.

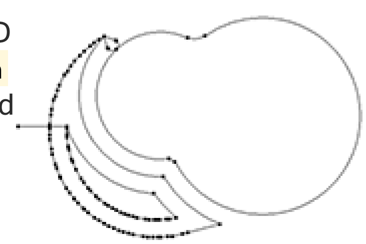
Slice Offset Body:



This option will accept any shape body with 2D or 3D elements. It is relatively fast and produces high-quality 2D toolpath. This option will work on all selected faces including 2D, 2.5D and 3D but only 2 and 2.5D faces will produce optimized toolpath. Note that this is the only choice that produces 2D toolpath from 2D and 2.5D faces. A 2.5D face is a face that can result in a 2D toolpath from an XY plane slice at a specific Z level. This 2D toolpath may be different at each Z level. Examples include spheres, cones, Z-axis revolved bodies, and some swept bodies.

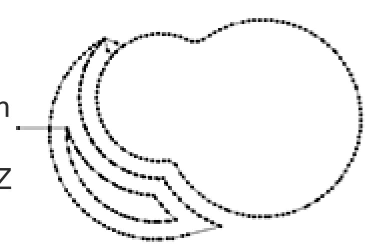
In order for the Slice Offset Body option to work, the entire body or all faces of a pocket must be selected. Additionally, all selected faces must be able to offset by the tool's corner radius amount. If the selected faces fail to be offset by the tool's corner radius, Slice Offset Body will not work. The probable cause is that faces at concave corners are smaller than the offset amount. If Slice Offset Body succeeds in its offset calculation, it will generate all toolpath and will skip any other Create 2D Toolpath choices. Slice Offset Body does not protect against undercutting, constraint faces or fixtures. If a partial body selection is made (faces are selected instead of the entire body), it is recommended that Use Stock is off.

2D on Top, Replace on Bottom:



This is intended for bodies that have a top Z range that is entirely 2D but then transition to 3D below this range. The 2D on Top, Replace on Bottom option will use From 2D Body methods for the top Z range and Replace TP with 2D Sections below that. This option is intended to improve performance in pockets that are primarily 2D with a complex floor. This option does a good job of cleaning up slow 3D toolpath where 2D and 2.5D faces have failed.

Replace TP with 2D Sections:



This will produce a combination of 2D and 3D toolpath. There are no restrictions on the shapes the option can work from.

The Replace TP with 2D Sections option will produce a 2D range down to a depth where 3D toolpath will be needed. The From 2D Body optimization option will be used to within the tool's corner radius in Z of the start of the 3D range. 3D toolpath will be generated from this

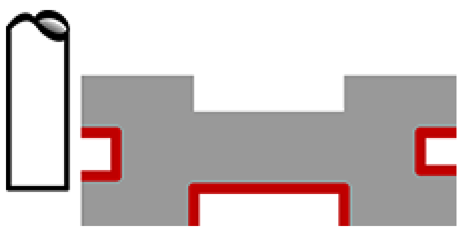
Z level down. This may produce some 3D toolpath on 2D faces near the transition area in Z, but is safer than gouging the part.
This option works best with single pockets as opposed to a large and complex group of faces that may transition from 2D to 3D at different Z depths in different areas. This option can significantly reduce toolpath generation time as the From 2D Body option is extremely fast but slowed on toolpath requiring many moves.

Limitations of Create 2D Toolpath

The following 2D toolpath limitations are noted as applicable in the function descriptions. Despite these limitations, there are many parts that do not need this protection, and the advantages of a 2D toolpath for prismatic solids are significant.

Undercut protection:

3D toolpath has undercut protection and will not allow the tool to cut a section of the part if doing so violates a higher area of the part. This includes a wall with grooves, a mushroom-shaped part, or blind features such as a pocket on the backside. This is why it is a good idea not to select backside faces when using Create 2D toolpath options. Some of the 2D methods do not have this protection. Undercut protection on a stock body has a different effect than undercut protection on a part body. On a part body, undercutting will gouge the part. Undercutting on a stock body may fool a tool into plunging into overhanging material, because it thinks there is no material at the Z level being machined; this does not cause a part gouge. Undercut protection eliminates both potential problems.



Areas of a part that may cause undercut problems

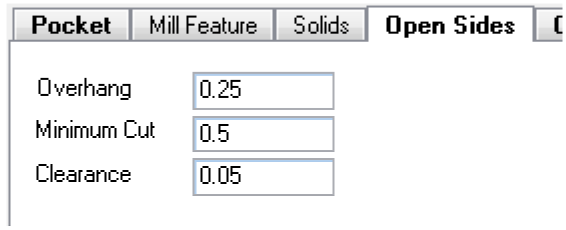
Constraint Face Protection:

3D toolpath will not gouge an unselected face on the same body. Some of the 2D methods do not have this protection. Without this capability, you cannot cut one face of a square pocket, as starting at the face’s edge will cut into its unselected neighbor.

Fixture Protection:

3D toolpath will not cut into a fixture body or face. Some of the 2D methods do not have this protection and will ignore fixtures.

Open Sides Tab



The system has an enhanced ability to machine open-sided pockets. This ability as it relates to geometry is fully detailed in the *Mill* guide. With SolidSurfacer, geometry does not necessarily need to be created or defined as "air" for this function to work. The part's stock will function as "air" geometry, and bodies will function as "wall" geometry.

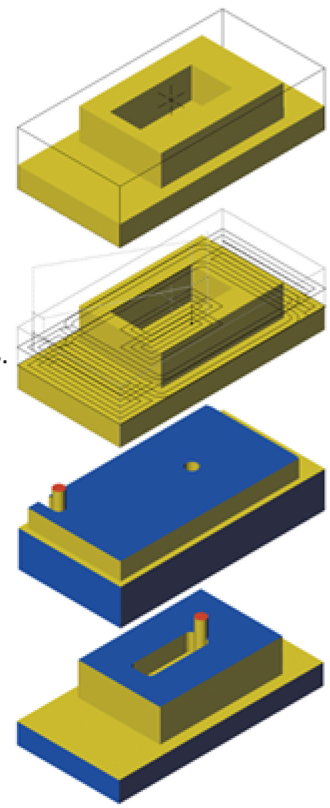
Both the Contouring and Roughing (with the exception of Face Milling) dialogs includes an Open Sides tab to specify parameters for open sides.

Overhang

This option only applies while roughing, and allows users to specify an amount that a tool will overlap the part's stock to clean up edges that might otherwise have a ridge. If this field is left empty, the system will automatically overhang the tool on the stock by the tool's cutting radius. This is also the maximum allowable value. Small values are best for normal roughing; large values can leave small ribbons of material for the last pass to clean up.

Minimum Cut

This option determines the smallest amount of material to remove along the outside of the material definition to complete a toolpath. The minimum allowable value is the tool's cutting radius.



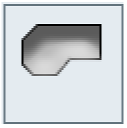
Clearance

This field allows users to specify the distance from an open-sided pocket from which a tool will enter.

To machine this model, a roughing process will be created along with drilled entry holes. This will all be done from one routine. The routine will consist of three operations: a hole operation and two roughing operations. Note that the toolpath extends to and rides on the stock diagram.

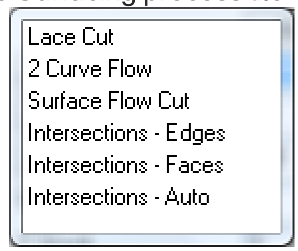
When the operation is rendered, we see that only one entry hole is drilled for the pocket that is bounded by the model. The model acts as a "wall" to the operation. Thus, the tool will start at the center and work its way outward. In this image we also see that the outer pocket, which has no boundary, only "air," has been started from an edge and the tool is working inward.

Once the open-sided pocket is complete, the system moves on to the bounded pocket. Note that this operation is machining outward.



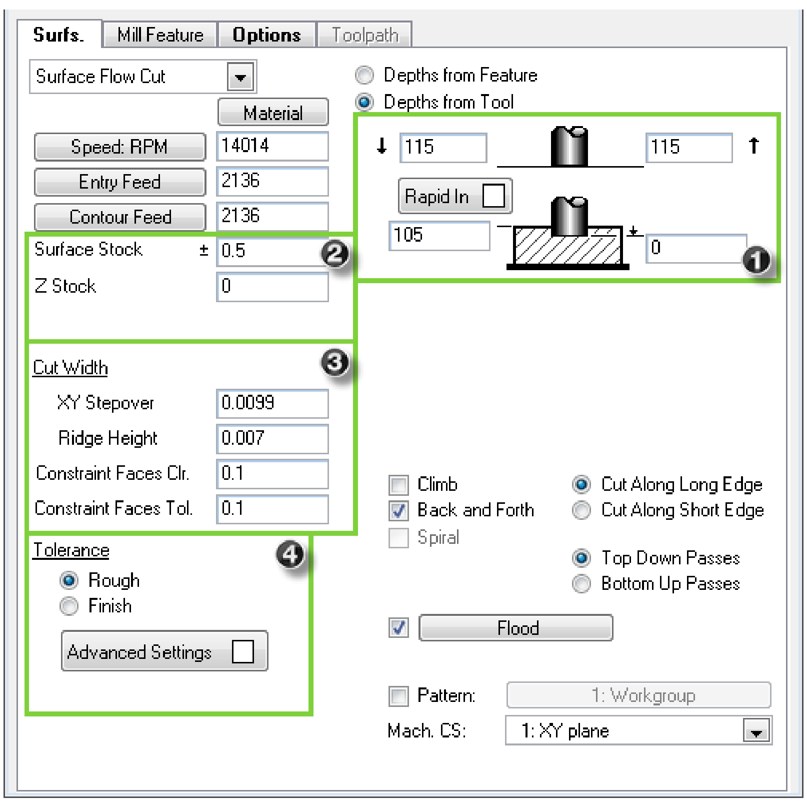
Surfacing Process

You use surfacing processes to create full 3 Axis toolpaths. You combine the Surfacing process tile with a tool tile to create a Surfacing process. The system supports surfacing with flat, bullnose, and ball endmills. You must select solids or sheets as the cut shape for surfacing processes. The Surfacing dialog always contains three tabs: Surfs., Options, and Toolpath. Other tabs, such as Rotate, appear if the machine type supports them. The specifications on each of these tabs change depending on the type of toolpath selected for the process. The six surfacing options are Lace Cut, 2 Curve Flow, Surface Flow Cut, Intersection-Edges, Intersections-Faces, and Intersections-Auto. For dialog items that are not described in this section, see the *Mill* and Advanced CS guides.



Common Surfacing Process Data

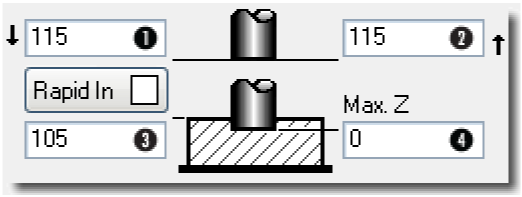
Several sections of the Surfacing process are common to all surfacing types.



1. Depths and Clearances
2. Stock
3. Cutting Control
4. Tolerance

Depths and Clearances

The clearances for SolidSurfacer processes are the same as for roughing or contouring.



1. Entry Clearance Plane
2. Exit Clearance Plane
3. Surface Z
4. Floor Z

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.

Exit Clearance Plane

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

Surface Z

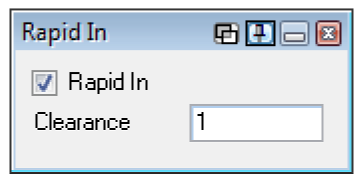
The Surface Z specifies the top level of the material.

Floor Z

The Floor Z specifies the finished depth of the pocket.

Rapid In

The Causes the tool to rapid from the clearance plane of the operation to the Z depth of the cut. You can also specify an incremental distance above the start point in the Clearance field (the default is zero). If the Rapid In checkbox is selected, a blue checkmark will appear on the Rapid In button. The Rapid In option should be used with caution, as it can create rapid moves directly into the part material.



Stock

Surface Stock

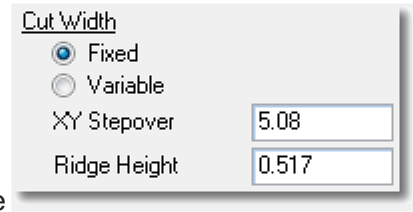
Surface Stock specifies the amount of material that will be left by the toolpath on any solid or sheet machined by the process. The toolpath will be offset by the Surface Stock amount in X, Y and Z. A negative value is valid for Surface Stock, provided that it is not greater than the corner radius on the tool being used for the process.

Z Stock

This value is the amount of extra material that the operation will leave or remove in the depth axis. Positive values leave material on floor Z and while negative values cut deeper than floor Z. A value of zero cuts to the depth specified.

Cutting Control

Cut Width:



XY Stepover creates the specified actual cut width for each pass, and calculates an approximate Ridge Height average. When you specify a Ridge Height , the specified toolpath is generated so that the cut width varies to maintain the specified Ridge Height on all areas of the machined surfaces. The toolpath is calculated so that no area on the part has a greater Ridge Height than the value entered. The XY Stepover is an approximation. For more information on Lace cutting, see “Cut Width” on page 96.

Constraint Faces Clearance:

The tool maintains this distance from any constraint face.

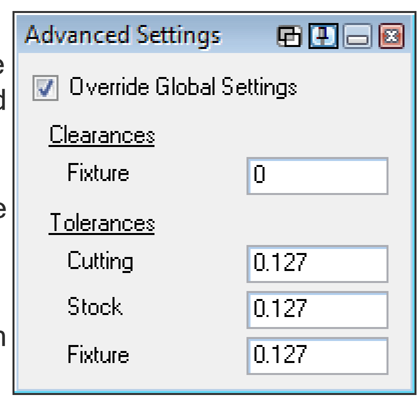
Constraint Faces Tolerance:

This is the tolerance for the Constraint Face Clearance. A tight tolerance means more precise toolpath at the cost of potential efficiency. This value should be less than the Constraint Face Clearance. For more information on constraint faces, see “ Machining Selections: ” on page 77.

Tolerance

When Use Global Settings for Solids is checked in the Document Control dialog, use these radio buttons to toggle between a Rough or Finish tolerance on the part. The rough or finish tolerances are set in the Document Control dialog.

Advanced Settings



Use the Advanced Settings to override the tolerances set in the Document Control dialog on a process-by-process basis. Click the Advanced Settings button to access the Advanced Settings dialog and then select the Override Global Settings checkbox to apply the clearance and tolerance values to the process. A blue checkmark will appear on the Advanced Settings button if the global settings are being overridden.

Clearances

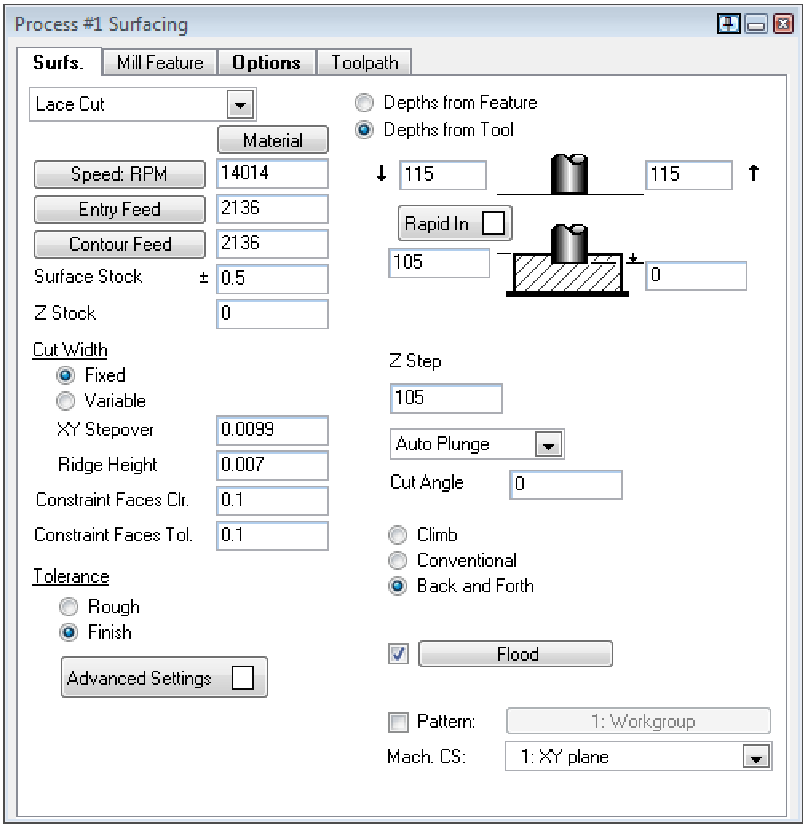
This section allows the user to set the interaction between toolpath and fixtures that are to be avoided. A fixture may be defined as a sheet or a solid designated as a fixture. There is a text box for the clearance value from a Fixture. This value is the additional distance the toolpath will be offset from the object.

Tolerances

These are the machining tolerances for the toolpath, or the margin of error. The toolpath may deviate by up to these amounts. A looser tolerance will require less memory and creates shorter output. To provide as much flexibility as possible, there are separate settings for Cutting, Stock, and Fixture. The Cutting tolerance is the tolerance of the toolpath over the selected face or faces–the area to be cut. The Stock tolerance is the accuracy of the toolpath's interaction with the stock definition. The Fixture tolerance specifies the accuracy of the toolpath's interaction with areas that are to be avoided. The default value for all selections is 0.005" or 0.127mm.

Lace Cut

Lace cutting is designed to cut surfaces at a specified angle and width.

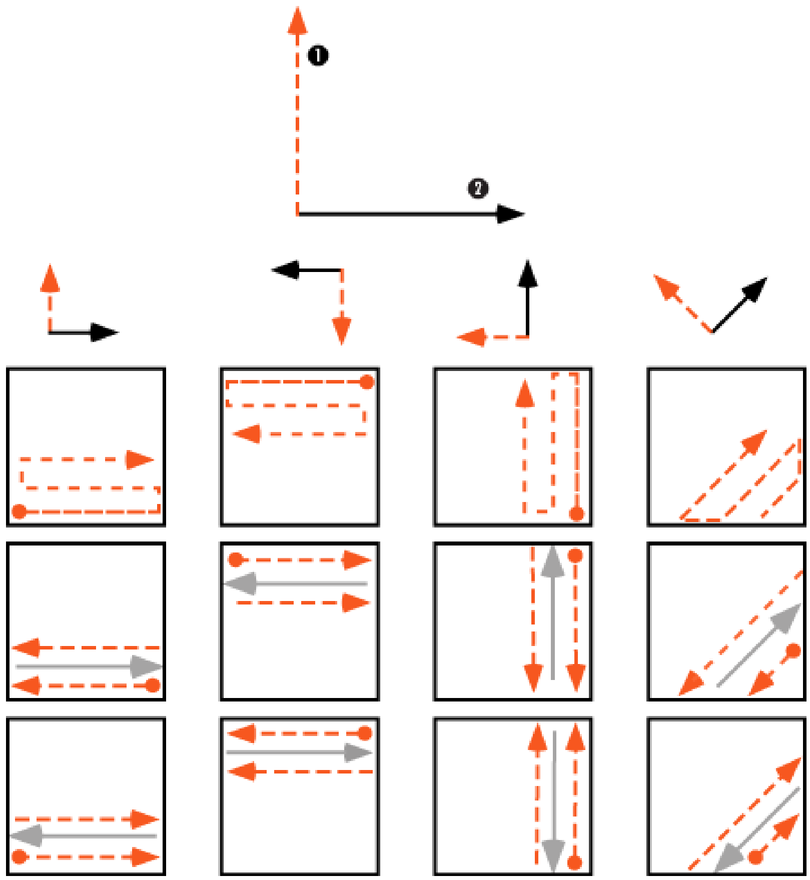


Cut Width

The cut width on lace cutting toolpaths can be fixed or variable. When Fixed is selected, the XY stepover specification is used to calculate the actual cut width for each pass, and the Ridge Height is an approximation. When Variable is selected, the Ridge Height specification is used to calculate the cut width for each pass, and the XY Stepover is an approximation. The cut width will vary to maintain the specified Ridge Height on all areas of the machined surfaces. The toolpath is calculated so that no area on the part has a greater Ridge Height than the value entered.

Cut Angle

The Cut Angle specifies the angle at which the tool will cut across the part. The Cut Angle also designates the direction and progress of the toolpath across the part. The diagram shows the relationship between the Cut Angle and toolpath progression. The cut angle and toolpath progression always maintain a normal vector relationship. If Climb is selected, the tool will always climb cut, retract and make rapid moves to begin the next cut. Likewise, if the Conventional option is selected, the tool will always conventional cut and retract. If the Back and Forth option is selected, the tool will not retract between cuts and will instead alternate between climb and conventional cutting, making a feed move between cuts. In the diagram, dashed lines indicate feed moves, solid lines indicate rapid moves, and the dot indicates the start point of the toolpath. The arrows indicate the direction in which the tool is cutting.



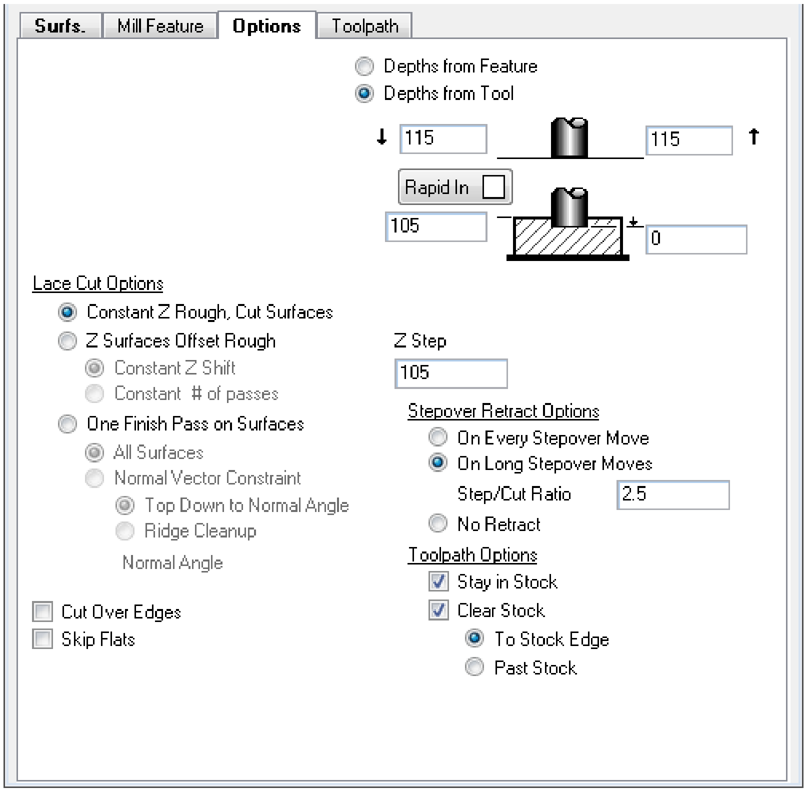
1. Toolpath
2. Cut Angle

Cut Angle

Mach. CS
For more information about this feature, see the *Advanced CS* guide.

Lace Cut Options Tab

The Options tab is enabled when the Lace Cut option is selected. It contains more detailed cut, stepover and retract options as well as controls for how to work with the edge of the part or selection.

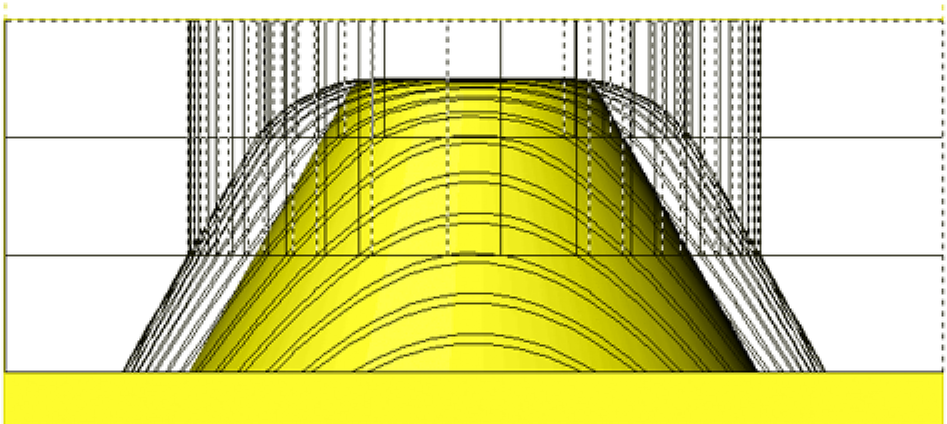


Lace Cut Options

The three options for lace cuts are Constant Z Rough, Cut Surfaces; Z Surfaces Offset Rough; and One Finish Pass on Surfaces. Two of the options contain variations; each generates a different type of lace cut toolpath. The first two are designed to create roughing and semi-roughing surfacing toolpaths. The last (finishing pass) option allows users to designate only certain portions of the selected surfaces to be cut/finished. Each type of lace cut is described below. There are also example toolpaths for each of the lace cut options. These pictures are shown from the side view.

Constant Z Rough, Cut Surfaces

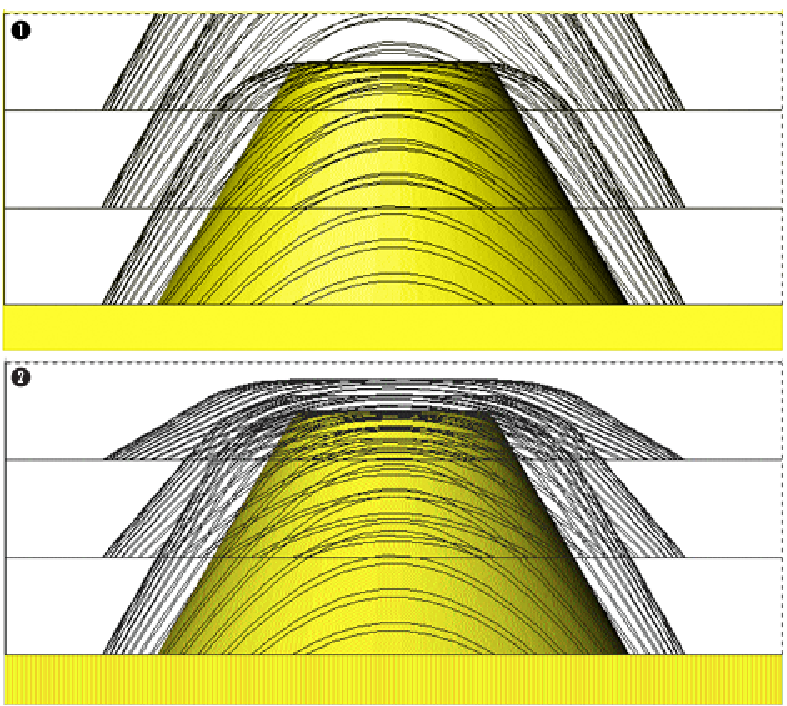
When this option is selected, the toolpath will cut along all surfaces that are part of the selected cut shape and that lie within the stock boundary. The system will first calculate the pass at the lowest Z depth specified by the Max. Z and create any subsequent passes up from there. No pass will be created above the Surface Z. Each pass will only cut surfaces not machined by previous passes in Z. When a portion of the surface that has already been machined by a previous pass is encountered, the tool will retract and move to any other surfaces that need to be machined at that Z depth. Each pass always steps down a fixed amount in Z.



Constant Z Rough, Cut Surfaces

Z Surface Offset Rough

This lace cut option creates a toolpath that repeats the finished toolpath (toolpath at final Z depth) up in Z. The final toolpath gets shifted up in Z by the desired Z Step, cutting off the pass when it hits the surface Z. The tool does not retract as it does with the Z Surface Offset Rough option; everything is cut. The tool rapids when outside the material, so the tool will plunge frequently. This option works best on gently undulating shapes. Constant Z Shift exactly duplicates the finished toolpath up in Z and retracts when it hits the surface Z. Constant # of passes breaks up any area that needs to be cut into the specified number of passes and then creates a step by connecting all of the passes. Each pass is a different shape; in addition, each pass cuts over the entire cut shape. There are no retract moves, as each pass cuts the entire part.



1. Constant Z Shifts
2. Constant # of Passes

Z Surfaces Offset Rough, Constant Z Shift

One Finish Pass on Surfaces

This lace cut option creates a toolpath that does not make any steps down in Z. The tool will only take one finishing pass over surfaces at the max Z depth. The different options allow users to control the surfaces that will be machined by the toolpath without having to select faces or designate constraints.

All Surfaces

With the All Surfaces option, the toolpath will machine all surfaces on the model at the Z depth entered for the Max Z that lie within the stock boundaries.

Normal Vector Constraint

This option is a process used to limit the area to be machined for a finish pass. Normal Vector Constraint specifies that toolpath will only be generated in areas where a surface is perpendicular to the tool within a given angular tolerance. The `Normal Angle` specifies this tolerance.

Top Down to Normal Angle vs. Ridge Cleanup.

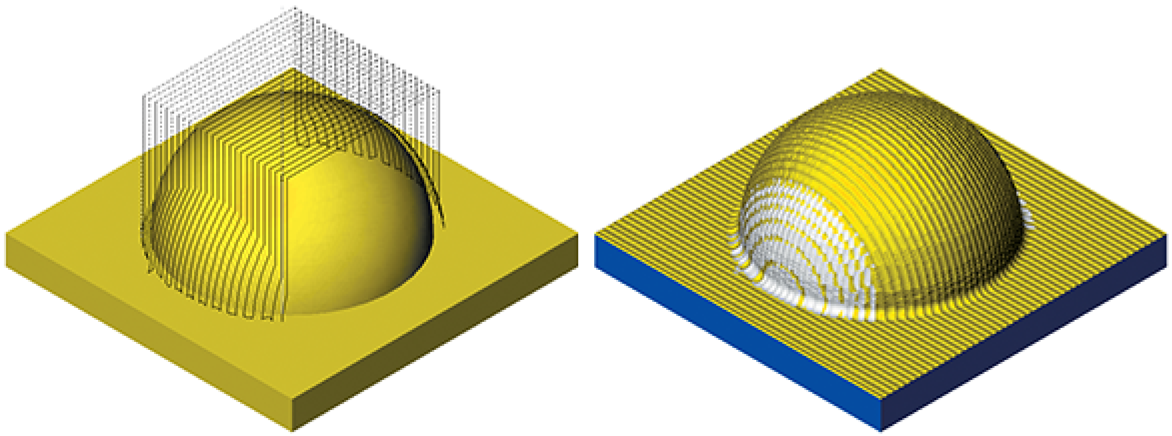
Top Down to Normal Angle

This will calculate this area from the top of a part. The normal vector of the area cut is effectively less than the Normal Angle value. This is useful for cleaning up where large ridges have been left by flat or bullnose endmills.

Ridge Cleanup

Ridge Cleanup is designed to work opposite from Top Down to Normal Angle. The toolpath generated by Ridge Cleanup is limited to surfaces whose normal vector is greater than the Normal Angle.

Ridge Cleanup toolpath illustrates how Ridge Cleanup should be used and the results. The previous operation left ridges on the sides of the sphere. The cut direction for the Ridge Cleanup is perpendicular to the previous operation. When rendered, the sphere is much smoother.



Ridge Cleanup toolpath

Normal Angle

The Normal angle specifies the angle toolpath will only be generated in areas where a surface is perpendicular to the angular tolerance set. For instance, if a value of 45 is entered as the Normal Angle value, the toolpath will be confined to areas of the part where the surface of the part is up to and including 45° off of perpendicular from the tool.

Cut over Edges

When this option is enabled, all generated toolpath will automatically cut past all mixed edges on a part (a mixed edge is the edge between a selected and unselected face). This option is off by default.

Skip Flats

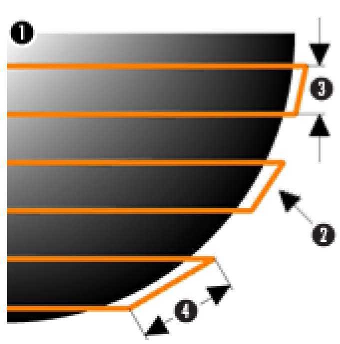
By default, toolpath is created that will machine the flat (perfectly horizontal) sections. Skip Flats will generate toolpath that will only follow the desired Z Step or Ridge Height.

Stepover Retract Options

These options only affect retract on the stepover moves of lace cuts; they determine how the tool will handle the stepover move between cuts.

On Every Stepover Move

This option will have the tool retract on every stepover move between cuts.



1. Machined Surface
2. Toolpath
3. Cut Width
4. Stepover Move

Stepover retract

On Long Stepover Moves

This feature provides for gouge protection when machining surfaces. If the ratio between the stepover move and the cut width (Stepover/Cut Width) is greater than the value entered for the Step/Cut Ratio, the system will retract for the stepover move. A larger ratio value will create fewer retract moves, while a smaller value will produce more retract moves. The default value of 2.5 is recommended to provide for the most efficient machining while still sustaining an adequate level of gouge protection.

No Retracts

A retract will not occur on stepover moves when the No Retract option is selected.

Toolpath Options

Stay In Stock

If Stay In Stock is checked, the toolpath will remain confined to the stock boundary, even if the selected face or body goes beyond the defined stock. This option is off by default so the toolpath will cover the entire selected body or face. Passes above the stock will be omitted but passes below the stock will be made to the final Z depth.

Clear Stock

This option will cause the toolpath to go beyond the bounds of the selected body or face in order to clear out the stock. The toolpath will only go as far as the Finish Z depth. Clear Stock functions in one of two ways, either

To Stock Edge

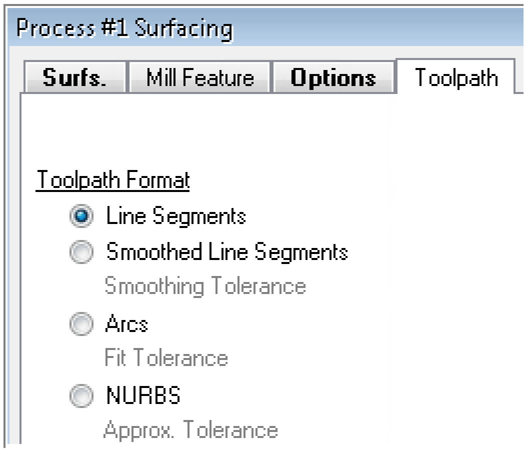
To Stock Edge will cause the toolpath to cut over the edge of the stock by the tool's radius.

Past Stock

Past Stock will cause the toolpath to fully clear the stock, extending the toolpath beyond the stock's edge by the tool's diameter.

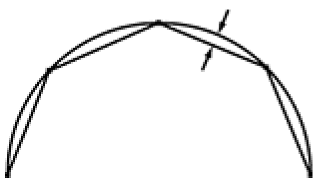
Toolpath Tab

When a lace cut is performed with a ball endmill, the Toolpath tab provides options to specify the method for generating the toolpath. Specifically, the options determine the type of geometric elements used in creating the toolpath. The geometric elements include lines, arcs and splines (also referred to as curves). If the toolpath is converted to geometry, the elements are visible.



Surfacing dialog, Toolpath Tab

Output options includeLine Segments, Smoothed Line Segments, Arcs and NURBS. Each output type is geared towards certain machines and CNCs based on their strengths. Some of these options provide extremely accurate toolpath (Toolpath Accuracy) at the expense of cut time. Other options provide better cut time with slightly reduced tolerance. Further options provide high tolerance and optimal cut time, depending on machine and CNC capabilities.



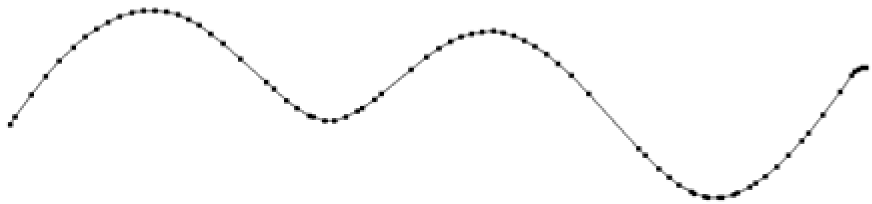
Toolpath Accuracy

Smoothed Line Segments, Arcs and NURBS each have a tolerance setting. The tolerance setting determines the allowable accuracy deviation from the surface to create toolpath. All of the tolerances are in addition to the standard tolerances found in the Surfs. tab.

The Smoothing Tolerance for Smoothed Line Segments controls the deviation from a smooth NURBS path. Setting the tolerance to a significantly smaller value than the standard tolerance results in a smoother finish. The Fit Tolerance for Arcs output controls the deviation of the arcs from a smooth NURBS path. The Approximate Tolerance for NURBS output is used only if the toolpath is output with a post that does not have NURBS support. If this happens, then the NURBS path is segmented at this tolerance.

Line Segments

This is the standard output and is the default setting. The output consists of non-uniform line lengths resulting in G01 moves.



Line Segment output (lines create the surface)

Smoothed Line Segments

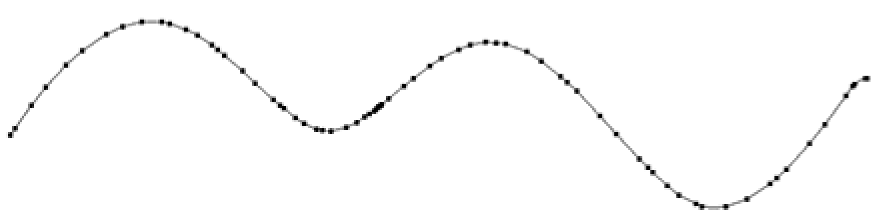
This option is similar to Line Segments, but results in a finer finish because its toolpath consists of more line segments. The system first builds and then segments a spline, which can result in the greater number of regular segments. These are also G01 moves.

Smoothed Line Segment output (many lines)

The Smoothing Tolerance is the accuracy of the toolpath to the surface to be machined. It is recommended that if the toolpath tolerance is to be 0.001” (0.0254mm), then the Smoothing Tolerance should be 0.0001” (0.0025mm).

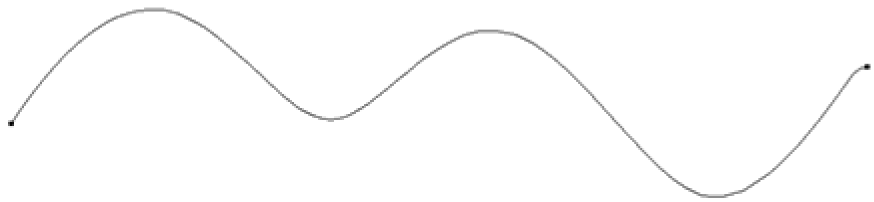
Arcs

This option outputs toolpath consisting of arcs. Toolpath is first created with NURBS and converted to arcs. Because many lines can be defined with relatively few arcs, there is a significant data reduction in the G-Code. Cutting arcs creates a smoother finish with more optimal machining times. This option is recommended for post processors that do not have NURBS, but that can perform circular interpolation. Note that machines that are limited to XY, YZ and XZ moves may have limited use for this option.



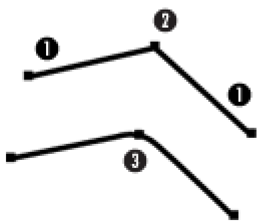
NURBS

NURBS toolpath is created directly from a surface or multiple surfaces of a part. This is the fastest (in terms of machining) and most accurate of the options. The NURBS option is designed for post processors with NURBS support. The Approx. Tolerance setting is available for machines that do not have NURBS interpolation. Smoothed Line Segments and NURBS are essentially the same output for machines without NURBS, so that parts can be interchanged between machines.



NURBS output (single curve)

A curve can be machined much faster because the machine does not need to stop at the vertex of two lines. This makes the NURBS output option ideal for high-speed machining.

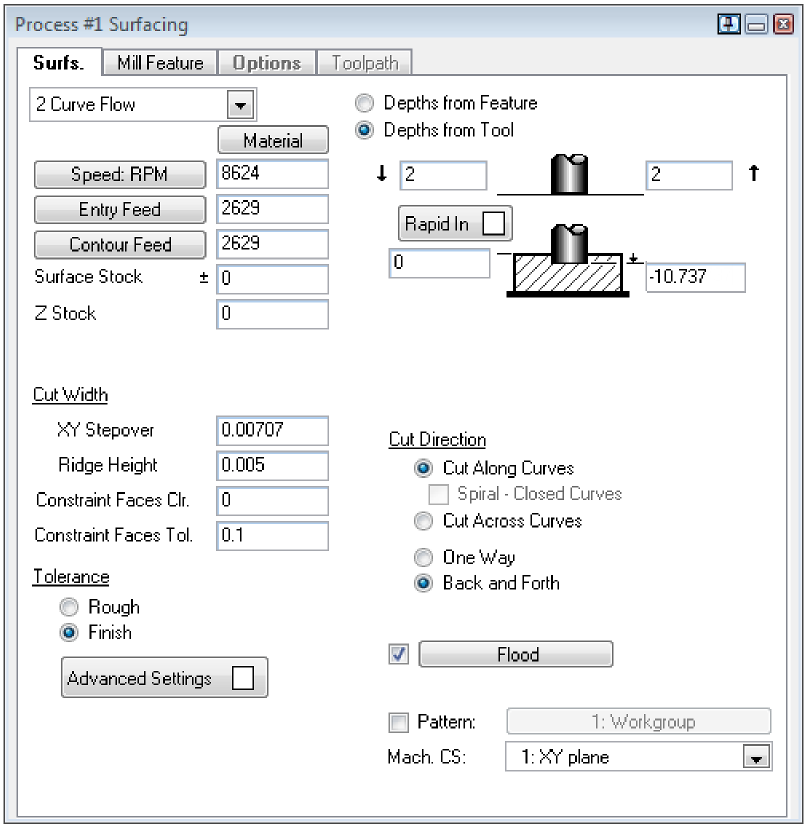


1. Line—machine will slow down
2. Stop
3. Tolerance curve

NURBS output can speed up machining.

2 Curve Flow

The 2 Curve Flow option creates toolpath that runs along or between two selected pieces of geometry and a solid that you can use for the cut shape.

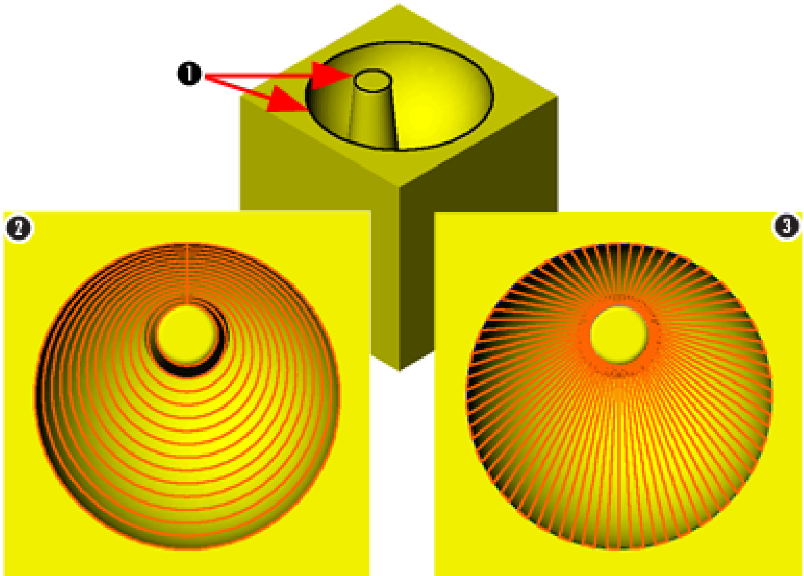


Cut Width:

The cut width is always constant on 2 Curve Flow toolpath and is based on the amount entered for the XY Stepover.

Cut Direction:

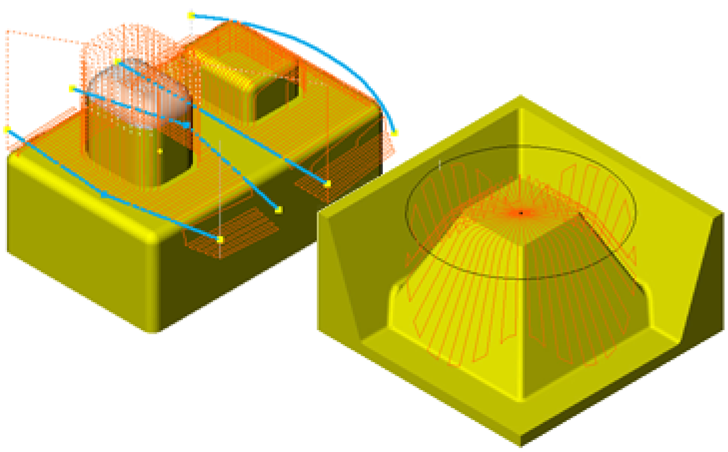
The Cut Along Curves and Cut Across Curves options indicate how the toolpath will flow between the two selected curves. The One Way and Back and Forth options determine whether the tool will retract after each cut. The Back and Forth option designates that the tool will feed from one cut to the next, while the tool will retract after each cut when the One Way option is selected.



1. Selected Curves
2. Cut Along Curves
3. Cut Across Curves

Example of 2 Curve Flow Cut Direction options

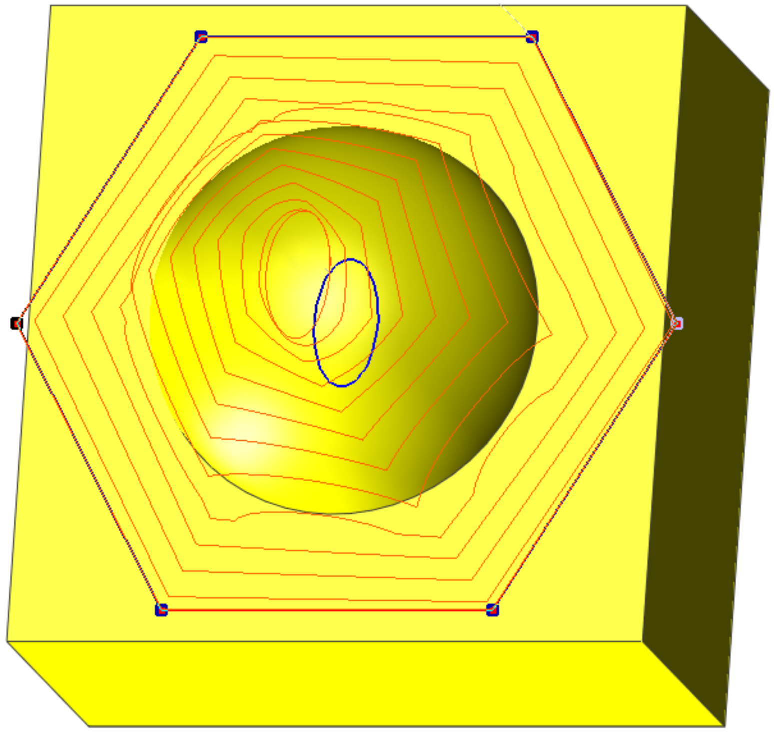
The system can produce 2 Curve Flow operations based on multiple pairs of geometry. There is no limit on the number of pairs of curves that can be used as a constraint for toolpath. An example is shown below. Note that the top of the taller boss is deselected, causing the toolpath to rapid over that section of the part. The 2 Curve Flow option also works with any selected geometry and a point. Please note that 2 Curve Flow recognizes fixtures but ignores stock conditions.



2 Curve Flow and geometry as a restraint for toolpath

Spiral - Closed Curves:

The Spiral - Closed Curves checkbox is available only for Cut Along Curves using One Way, discussed above. This form of 2 Curve Flow lets you define toolpath that spirals from one closed curve to another one that surrounds or is surrounded by the first. Either of the closed curves can be a circle, ellipse, polygon, or a free-form closed spline, and you can specify a point as the inner "curve". The spiral of the toolpath gradually morphs between the two curves.



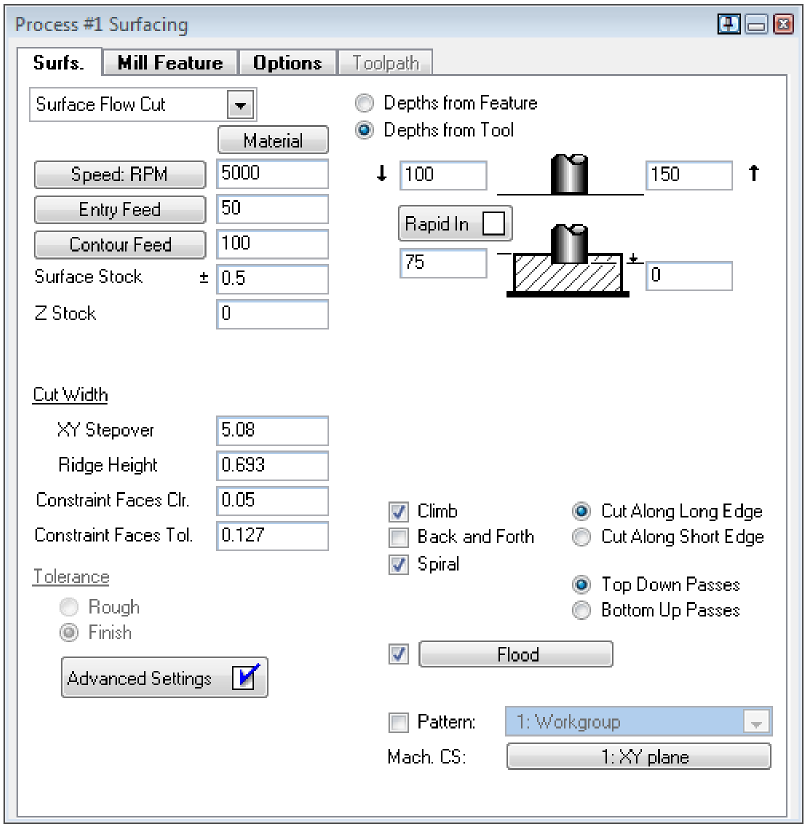
2 Curve Flow using Spiral - Closed Curves

Surface Flow Cut

Surface Flow Cut machines the entire area of a selected face. As such, Surface Flow Cut is designed for machining single faces, and works best when machining simple chamfers and fillets. Please note that Surface Flow Cuts ignore stock conditions as well as fixtures and geometry.

The two important items you use to define for a Surface Flow operation are:

- "Surface Flow Start Point" on page 109
- "Surface Flow Options Tab" on page 110



Surface Flow Start Point

The start point for Surface Flow Cut operations is controlled by the checkboxes: Climb, and Back and Forth or Spiral.

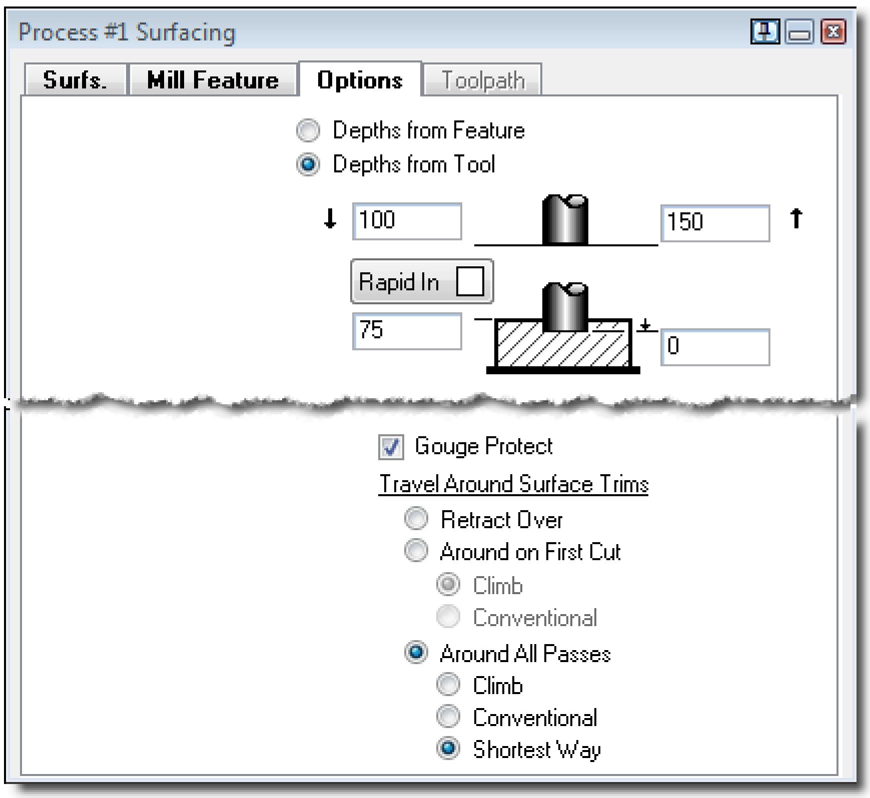
- When the Climb checkbox is selected, the process will be a “Climb” cut–the tool will cut in one direction starting on the right side of the part and climb cut across the selected surface. After each cut, the tool will retract and rapid back to the start of the next cut. If the checkbox is unselected, the system will default to a “Conventional” cut and the tool will start cutting on the left side of the part.
- When Back And Forth is selected, the tool will alternate between Climb and Conventional cuts, making a feed move between cuts.
- Alternatively, when Spiral is selected instead, the tool will follow a continuous spiral pass. However, Spiral can only be used on blended surfaces such as fillets. If you are unsure what kind of surface you are machining, you can use a plug-in: [Solids >] Solid Inquiry to query the surface.

The Cut Along Long Edge / Cut Along Short Edge options determine the flow direction of the toolpath. All surfaces have a U and V axis of flow lines. The system analyzes the selected surfaces and determines whether the U or V flow lines are longer for each face. Depending on the selection made here, the toolpath will either flow along the U or V axis, depending on which is longer or shorter.

TheTop Down Passes or Bottom Up Passes options determine how the toolpath will proceed along the depth axis of the machining CS. If the Top Down Passes option is selected, the toolpath will be traveling down the surface along the depth axis. If the Bottom Up Passes option is selected, the toolpath will travel up the selected surfaces along the depth axis. The tool approaches the path along the depth axis of the machining CS, so the depth axis can also be thought of as the tool axis.

Surface Flow Options Tab

A surface trim is the edge of an island or a cavity that is in the faces selected to be cut. These options only affect the tool moves made when a surface trim is encountered on the surfaces to be machined.

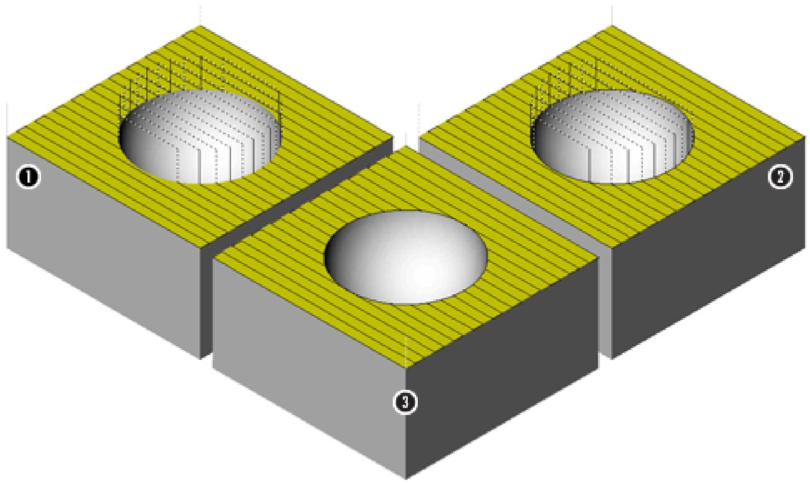


Gouge Protect

When enabled, the system will check for gouges with the tool on the selected surface against any selected constraint faces. For information on setting constraint faces, see “ Machining Selections: ” on page 77.

Retract Over

This option designates that the tool will retract over the surface trim when it encounters one.



1. Retract Over
2. Around on First Cut
3. Around on All Passes

Example of Travel Around Surface Trim options

Around on First Cut

When the tool encounters a surface trim, it will make a cut around the edge, but only on the first cut that encounters the surface trim. The user can designate this edge cut to be a Climb or Conventional cut.

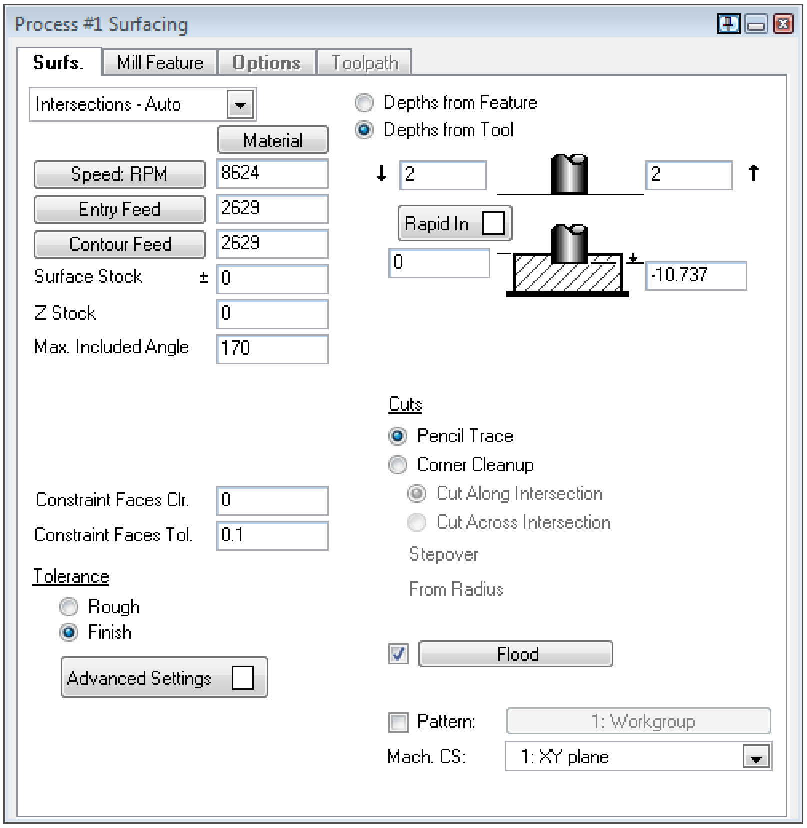
Around All Passes

The tool will cut around the edge of a surface trim on every pass that is made that encounters the surface trim. The user can choose whether this cut around the surface trim be a Climb or Conventional cut or designate that the cut travel the Shortest Way (whether it be Climb or Conventional).

- On horizontal or near-horizontal faces, the system may not give the expected result. Try changing the cut direction setting.
- The first cut of a Surface Flow process is not specifically guaranteed to be a Climb or Conventional cut. These machining parameters are only used for designating the start location of an operation.

Intersections

This surfacing option creates toolpath on selected intersections (either edges or faces of solids and sheets). The Intersections process is not restricted by stock definitions and does not recognize fixtures, but is a gouge-free process that avoids red constraint faces. This is because the toolpath machines the border or edge that is selected, thus overlapping to be tangent to an unselected face. This option works with constraint geometry. You can access the three types of intersections from a pull-down menu. For some tips on how to work with Intersections toolpath, see “Notes on Intersections Toolpath” on page 113.



Intersection - Edges:

Machines the area along a selected edge or edges. Selected edges must only be between two faces of a solid or sheet. This option works best when the faces attached to a selected edge are not vertical. To check whether a face is vertical, select Solids > Tools > Machining Face Check).

Intersection - Faces:

Machines the border of selected faces. The tool remains in contact with the selected face and the surrounding faces but avoids constraint faces. Note that Corner Cleanup cannot be performed on a fillet with a tool that is smaller than the fillet.

Intersection - Auto:

Machine all edges and all borders of faces on selected faces, an entire solid, or a sheet. Machines only edges or borders within the Max. Included Angle.

Max. Included Angle:

Toolpath is only generated where the angle between the intersecting faces is lesser than or equal to the angle specified here.

Cuts:

These options control how the tool cuts intersections.

Pencil Trace:

Drives a tool along selected edges or borders of selected faces in a single pass within the Max. Included Angle while maintaining contact with any face attached to the selected edges or faces.

Corner Cleanup:

Drives a tool along or across selected edges or faces within the Max. Included Angle in multiple cuts while maintaining contact with any face attached to the edges or faces. You can select Cut Along Intersectionto cut along an edge or face or Cut Across Intersectionto cut back and forth across an edge or face.

Stepover:

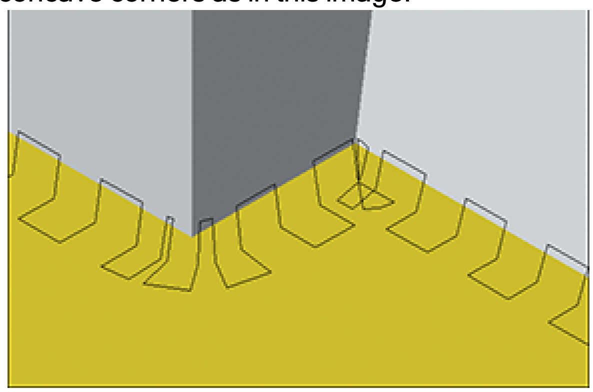
This specifies the tool's stepover for the operation.

From Radius:

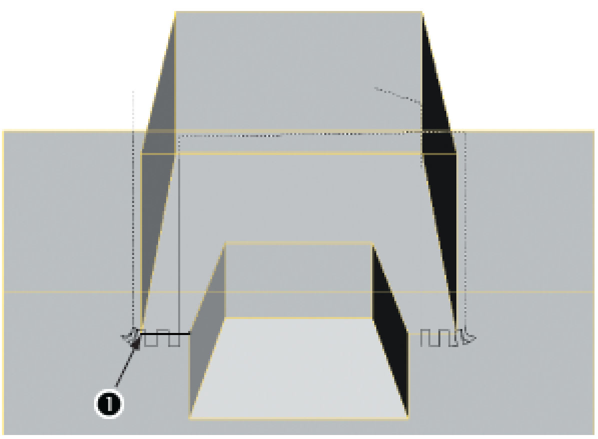
Enter the size of the tool to previously machine this area of the part. This allows the system to properly calculate the optimal area to machine.

Notes on Intersections Toolpath

In Corner Cleanup, the toolpath may overlap in concave corners as in this image.



- You may experience larger bottom connection moves and/or wavy upper connection moves (the move is projected and is travelling over the corner) in the toolpath during Corner Cleanup—Cut Across Intersection when cutting convex corners as shown in the following example.
- Toolpath generated along the faces of a selected edge automatically machines all other areas of these faces that share a common edge. To avoid this, use geometry to restrict the machining area. In the following example, one edge was selected (on the left side) but we see toolpath at all the places where these two faces have common edges.



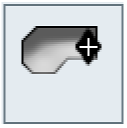
1. Selected Edge

Machining Tips

- Remember the stock hierarchy: temporary stock (Machining palette), stock body, workgroup stock, workspace stock.
- Deselect undercut faces for 2D toolpath.
- When Use Stock is active, a pocket in a solid can be machined by simply selecting

the face that comprises the floor of the pocket.

- 2 Curve Flow ignores stock conditions but recognizes fixtures.
- A Surface Flow Cut is designed to machine all of a selected face or faces. It does not recognize stock or fixtures. Be sure to inspect your toolpath carefully.
- If a part with Corner Cleanup is saved to an earlier version of the system, the operation can be lost. This is because there is no compatible operation in earlier versions. The Corner Cleanup operation may be viewed, output to geometry and posted but if the operation is redone or if Redo All Ops is selected, the operation will be deleted.
- When performing a Roughing, Contouring or Intersections–Corner Cleanup operation, there is a very small chance that the actual tolerance may be off from the machining tolerance by up to 70%. This is most likely to occur at sharp corners. If this is the case, try tightening the machining tolerance by 50%. If a finishing operation remains to be performed, there is nothing to do as the remaining stock is greater than any deviation in the tolerance.



Advanced 3D Machining

This section includes the following topics:

About this Process

The Advanced 3D process includes 12 kinds of 3-axis milling capabilities. Advanced 3D expands the 3D surface and solid body machining of SolidSurfacer in many ways. Some of the key differences between this and the basic solid machining process are:

- Support for facet body machining
- Support for 3D Material Only
- Support for more advanced tool shapes (tapered tools)
- New types of finishing processes (e.g. Surface Stepover Cut and Steep/Shallow Combination)
- Improved toolpath quality
- Options to provide toolpath that better supports High Speed Machining
- Improved use of boundaries
- Enhanced entry/exit control
- Ability to directly machine faceted bodies, including imported STL files
- Automatic Core/Cavity detection for inside out or outside in determination
- Operation splitting for tool wear and for optimal length out of holder
- Multi-threaded for multiple CPU machines and to support batch toolpath generation
- Automatic filleting of a surface to avoid sharp concave corners

Save a Copy – Warning

For parts with Advanced 3D toolpath at this release:

WARNING: We recommend against using Save a Copy to GibbsCAM 12 or earlier. This type of toolpath from the current release may not be preserved by Save a Copy, and so you may be unable to Redo, render, or post the part in the earlier release.

What is Different?

The Interface

The interface is essentially the same as the Pocketing and Solids processes. Many of the items share the same terminology and functions. Dialog box tabs provide access to various types of controls. See "Surfaces Tab" on page 120, "Options Tab" on page 157, "Entry/Exit Tab" on page 161, and "Boundary Tab" on page 166. The Advanced 3D process types are accessed from a menu, similar to the Pocketing options. See "Toolpath Cut Types" on page 121. A distinct visual difference is the addition of the Task Manager. See "What is Different?" on page 116. However, most of the differences are conceptual or how the toolpath is calculated. See "Toolpath Calculation" on page 117.

Processes

Default Values

The default values generally work like other GibbsCAM processes, where the last used values define the default for a new process. Some of the parameters are shared across multiple sub-process types, and some are unique to a sub-process type. Also, a number of parameter defaults are calculated from tool and default stock definitions. These tool-dependent defaults include the various stepover settings (for example, stepover, XY stepover, min. and max stepover), entry and exit arc radii, horizontal approach clearance, some of the trim to stock and rest material settings, retract style settings, boundary settings, and various other parameters. These default values are recalculated whenever a new tool is dropped on the process tile.

In the case where you have an existing operation and you need to change any tool parameters, it is highly recommended that the changes be made ***before*** loading the operation to the Process list – that is, before double-clicking on the operation. This is due to Advanced 3D Machining dependencies and automatic values. If a tool is modified while the process is open, the dependencies may not be updated.

Process Tiles

Because the Advanced 3D toolpath calculation can involve large amounts of time to calculate large toolpaths, the calculation software is multi-threaded. The calculation is performed in a separate system thread from the standard GibbsCAM thread. This allows you to perform many other GibbsCAM functions while the toolpath is being calculated. If the computer has more than one CPU, the calculation creates one additional thread for each available CPU whenever possible, which can speed up calculation time. To maintain data integrity of the GibbsCAM operation data, you cannot delete a tile while that operation's toolpath is being calculated. To delete an operation, you must first stop the calculation and then delete the operation. You can move an operation that is being calculated. In this case, the toolpath is added to the moved operation after the toolpath calculation finishes.

Base Process and Dependent Process

There are two classes of Advanced 3D processes:

- Base processes
- Dependent processes

A base process is any process that can exist independently. They are the normal processes like Lace Cut, Surface Stepover Cut, or Curve Projection. A dependent process is one that can only attach to a base process, and it is dependent on the base process. The following dependent process types are available: What is Different?, Toolpath Splitter, and Tool Holder Gouge Checker. Dependent processes must always follow a base process in a process list (although there may be blank tiles or other non-dependent processes in between). There can be multiple dependent processes in a row, all dependent on the same base process, and there are no limits to the number of dependent processes in a sequence. The Material Only dependent process is used to pass the leftover material from a base process to become the input stock for another process. See "Material Only process example: " on page 170. The Toolpath Splitter process is used to split a single process into multiple operations based on tool path length, usually to change a dull tool for a sharp one. See "Toolpath Splitter process example" on page 155. The Tool Holder Gouge Check process is used to split a process into multiple operations, using a shorter tool (i.e. shorter tool stick out length from the holder) to rough the process, and then a longer tool (i.e. longer tool stick out length) to finish the process. See "Tool Holder Gouge Checker process example" on page 156. There is no limit to the number of dependent processes in a single process list. Also, you can combine material only, Tool Path Splitter, and Tool Holder Gouge Check processes referring to the same Base Process. See "Example of Toolpath Splitter and Tool Holder Gouge Checker used together" on page 157.

Toolpath Calculation

Several inputs can affect the toolpath calculation, including: What is Different?, What is Different?, What is Different?, What is Different?, What is Different?, What is Different?, and What is Different?.

Boundaries

Advanced 3D is boundary based. That means each and every operation needs a boundary to be defined, even if it is just the default Bounding Box. The five boundary types are: Bounding Box (default), Silhouette, Shallow Areas, Cutter Contact Areas, and Selected Curves. For more information, see "Boundary Tab" on page 166.

Stock

The four stock types are: Bounding Box, Part with Offset, Solid, and Rest Material. If there is no stock selected, the default is Bounding Box. You can select a stock of Solid Body or Facet Body. For a casting part, the part itself with a certain offset can also be used as stock. This is done by using "Part with Offset" as the stock type, and defining parameters for the offset value, tolerance, and other values used by the system to calculate the appropriate casting size and shape. For more information, see “Stock Management” on page 168.

Facet Bodies

Facet bodies can be available as stock or as part to be machined. Just select the facet body as the selection to be machined. To use as stock, change the solid property of the facet body to stock, or select the stock selection mode icon on the CAM palette and select a facet body to be used as stock.

Fixture bodies

Fixture bodies are not supported at this time by the Advanced 3D software. Other methods such as creating Geometry to be applied as a boundary may be able to compensate for this.

Material Only

The three different technologies available for performing Material Only machining are: 2D material, 3D material, and machining with a stock body. Machining with a stock body is more general and can define the rest material from any number of operations. For an example of using Material Only, see “Material Only process example: ” on page 170.

To use 2D or 3D stock you specify, the stock type to be “rest material.” See the Boundary Tab. For a series of dependent “rest material” processes to use the 2D material only calculations, all of the processes in the series must be an Advanced 3D waterline (or “constant Z”) cutting method. This means that they must be an Advanced 3D contour or a pocket process. Any other combination of Advanced 3D dependent “rest material” processes will use the 3D material only calculations. Both types of calculations produce similar results, but the 2D method takes less time to calculate.

To perform material only processes using a stock body, you need a stock body that represents the material condition as it is just before the process is performed. You can either create a solid body or a facet body. The most automated way to create this stock body is to use Flash CPR on all the prior operations, and then stop CPR and create a facet body. You can set the solid property of the facet body to “stock” (that is, make it “global stock”), or you can select the facet body while the “stock” icon is depressed in the CAM palette for an individual Advanced 3D process (that is, “local to the process stock"). When you create an Advanced 3D process, select “solid” for the stock type, and the toolpath will be trimmed to the stock body defined.

Curve selection

Selected curves work as user-defined containment boundaries by default. The system interprets these curves as “2D,” meaning that it creates vertical surfaces, parallel to the tool axis, by extruding the selected curves. The selected curves must form one or more closed boundaries. Any toolpath that is outside of these selected curves is trimmed out.

You can select drive curves and trim curves by using the drive curves and trim curves buttons on the dialog for N Curve Flow. When you do this, you will see that the curves are displayed with an arrow showing the direction, and a wide colored line over the curve to show the full curve to be used. Note that all the drive curves must go in the same general direction, and both trim curves must go in the same general direction, linking the drive curves together. The drive curves must terminate at their intersection points with the trim curves. You must have, at a minimum, two drive curves and zero trim curves. There is no maximum number of drive or trim curves. All drive and trim curves must be open curves.

For Curve Projection sub-processes, you can select the projection curves by selecting the projection curve button on the dialog. The selected projection curves display with an arrow marker to show the machining direction, and a wide colored line to show the full curve that is used.

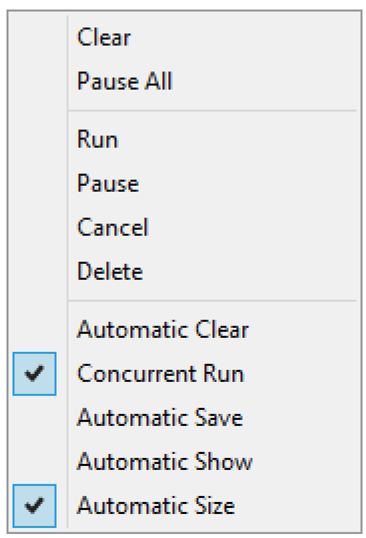
Automatic Fillet Surfaces

The Advanced 3D process can automatically create fillet surfaces along any concave intersection of two faces in the machining face or body selection. This allows the resulting toolpath to contain curved sections instead of sharp inside corners, which makes for better high speed toolpaths. The created fillet surfaces are tessellated surfaces that are created and added to the surface/body selection just prior to the toolpath being calculated. These surfaces are discarded once the toolpath is calculated. To activate this feature, select the “Add fillet” checkbox in the Fillet section of the Options Tab, and define a fillet radius to use.

Task Manager

Advanced 3D is multi-threaded. That means you can run multiple independent processes simultaneously, depending on the number of CPUs or cores (real or virtual) configured on your computer. When the Do It button is clicked, the processes in the process group are added to the Task Manager. Each task's status in the Task Manager can have one of these status values: Queued, Running, Blocked, Paused, Canceled, or Failed. If processes in the Task Manager are not paused, the tasks are executed immediately. When completed, the toolpath is added to the corresponding operation tiles in the Operation Panel.

Task Manager right-click menu.



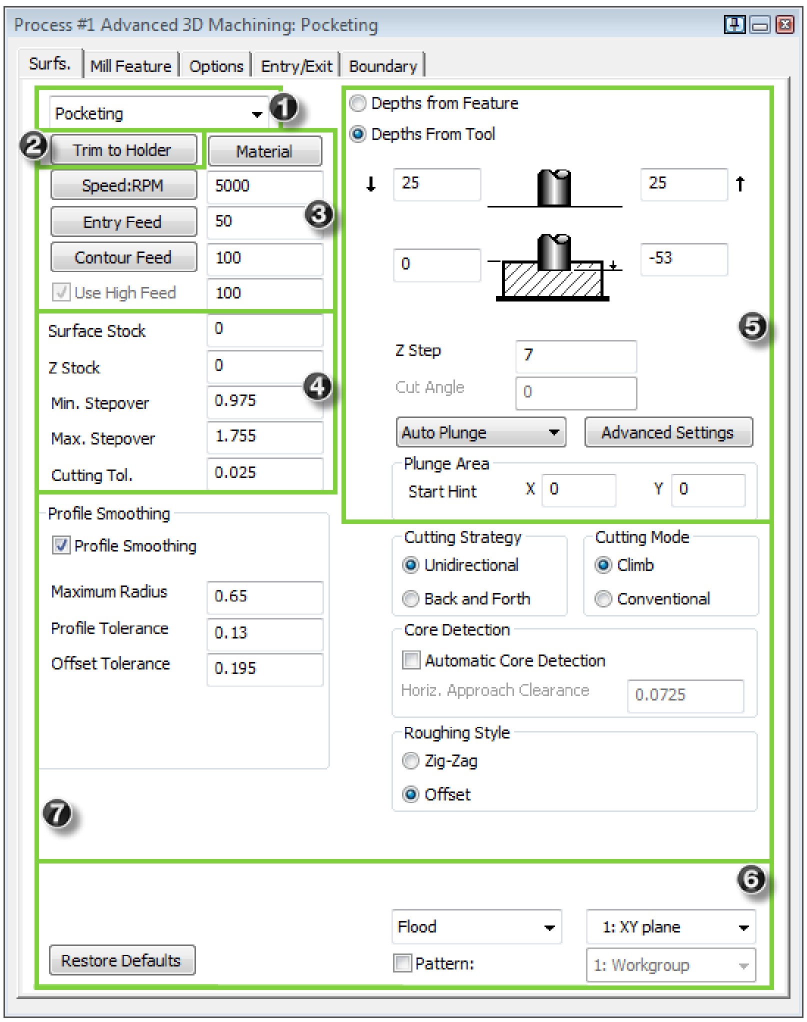
Use the Pin symbol on the Progress bar to display/hide the Task Manager.

Surfaces Tab

For the Advanced 3D Machining process, you use the Surfs. (Surfaces) tab to set basic information such as:

- Type of cut. See Toolpath Cut Types.
- Speeds and feeds. See Material, Feeds, and Speeds.
- Basic parameters including stock to leave, stepovers, and tolerances. See Basic Parameters.
- Settings such as Clearances, Cut Depth, and Plunging.
- Options that are common to all processes. See Common Options.
- Controls that are specific to the toolpath cut type you select.

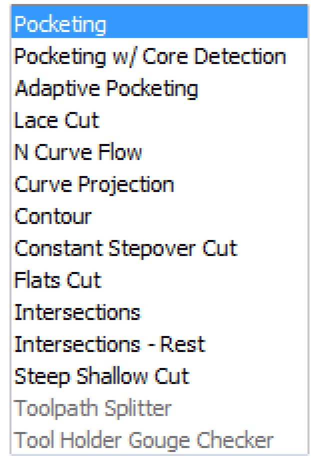
This section describes these options.



1. Toolpath Cut Types
2. Trim to Holder
3. Material, Feeds, and Speeds
4. Basic Parameters
5. Clearances, Cut Depth, and Plunging
6. Common Options
7. Controls specific to a process

Elements of the Surfaces tab.

Toolpath Cut Types

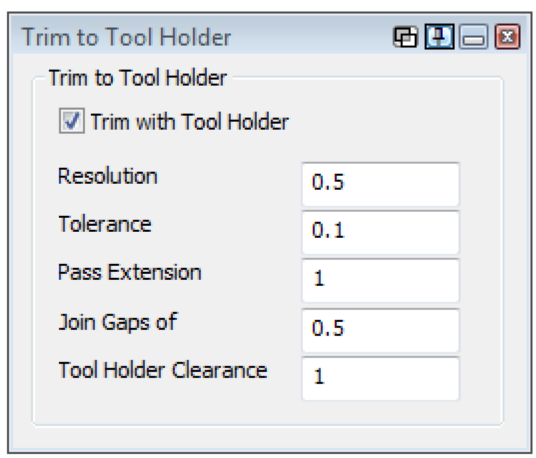


The Advanced 3D Machining process offers many different types of toolpath. When you start making a process, you choose what type of toolpath to use. The options include:

- "Pocketing" on page 129 and "Pocketing With Core Detection" on page 130
- "Adaptive Pocketing" on page 131
- "Lace Cut" on page 133
- "N Curve Flow" on page 136
- "Curve Projection" on page 138
- "Contour" on page 139
- "Constant Stepover Cut" on page 140
- "Flats Cut" on page 142
- "Intersections" on page 143
- "Intersections - Rest" on page 145
- "Steep Shallow Cut" on page 151
- "Toolpath Splitter" on page 154
- "Tool Holder Gouge Checker" on page 156.

When you select a toolpath type, the **Surfaces** tab displays the corresponding options.

Trim to Holder



When you click the Trim to Holder button in the Advanced 3D process dialog, the Trim to Tool Holder dialog appears. It lets you supply values to trim the toolpath in such a way as to prevent the tool holder from hitting the finished part geometry.

Select the Trim with Tool Holder checkbox to supply values for the following parameters:

Resolution

The distance between points at which the toolpath is checked. At each point the cutter is checked to see if it is above or below stock.

Tolerance

The accuracy to which the point at which the cutter enters or leaves the stock is found.

Pass Extension

When editing passes to surfaces, it is often desirable to extend the pass away from the material to give better cutting conditions when linking. This parameter controls the amount of extension applied.

Join Gaps of

When trimming passes to stock, it is possible for the resulting passes to become fragmented with many small gaps. This parameter allows gaps which are smaller than the distance specified to be “un-trimmed” and thereby avoiding fragmentation.

Tool Holder Clearance

An additional thickness, which must be a positive value, that defines the minimum distance the holder must clear the part and stock. The system will preserve all passes that are within the specified distance from the stock.

Material, Feeds, and Speeds

The items in this section of the process dialog are identical to all other processes. You can click on the Material button to select or edit a part material. The Speed: RPM, Entry Feed and Contour Feed buttons will suggest a speed based on the material or you may manually enter a value. For more information, see *Mill*.

Use High Feed

When this option is active all rapid movements will be replaced with high feedrate movements.

Basic Parameters

The basic parameters vary, depending upon the toolpath cut type selected. Each of the cut types have stock settings, stepover settings, and a tolerance setting.

Surface Stock:

This item specifies how much material is left on a surface. Any positive amount can be left, but negative values must be less than the corner radius of the tool.

Z Stock:

This item specifies how much stock is left on the depth axis of a cut for floors only, not 3D shapes. Positive values leave extra material, negative values cut deeper. All faces are cut using the surface stock value.To cut flats to size, you must enter the negative of the surface stock value. For example, if the process leaves 1mm of surface stock and you want the floor cut to size, you must enter `-1` in the Z Stock.

Z Step:

Depth of each successive pass.

Stepover:

The stepover represents how much the tool moves over for the next pass across the model. This is a distance value between the passes, not a planar stepover. The Stepover value is set as a specific value for the following processes: “Constant Stepover Cut” on page 140, “Intersections - Rest” on page 145, and “Steep Shallow Cut” on page 151.

Stepover/Scallop Height:

The following cut types can define either a stepover or a scallop height when using a ball endmill: “Lace Cut” on page 133, “N Curve Flow” on page 136, “Curve Projection” on page 138, “Constant Stepover Cut” on page 140, “Intersections” on page 143, “Intersections - Rest” on page 145, and

"Steep Shallow Cut" on page 151. The scallop height is a theoretical value based off of a flat surface.

Minimum Stepover:

Each Z step consists of an outer profile and concentric inner shapes. The distance between each inner shape is the stepover. The stepover tends to be a uniform size but some cut types vary the stepover to make the toolpath more efficient. This value sets the smallest stepover. A recommended value is automatically calculated based on the tool size. By default, the smallest stepover is the tool radius on a flat endmill. The value automatically updates with any changes to the tool. This item is used by the following processes: "Pocketing" on page 129 and "Flats Cut" on page 142.

Maximum Stepover:

Each Z step is made of an outer profile and concentric inner shapes. The distance between each inner shape is the stepover. The stepover tends to be a uniform size but some cut types will vary the stepover to make the toolpath more efficient.This value sets the largest stepover. A recommended value is automatically calculated based on the tool size. By default, the maximum stepover value is 90% of the diameter of a flat endmill. The value automatically updates with any changes to the tool. This item is used by the following processes: "Pocketing" on page 129 and "Flats Cut" on page 142.

XY Stepover:

This value represents a parallel 2D stepover in the toolpath from one pass to the next. The default value is 1% of the tool diameter, or 2% if the diameter is 1" (25mm) or larger. This field is displayed for cut method `Pocketing` when `Zig-Zag` is in effect.

Rough Stepover:

The value you specify for `Rough Stepover` ...

Cleanup Stepover:

The value you specify for `Cleanup Stepover` ...

Side Clearance:

This parameter applies only to `Pocketing` with style set to `Zig-Zag`; it is unavailable when the tool is a ball mill. This value specifies how far to offset each end of a zig-zag path from the profile that is being machined. It is useful for cutting flat areas. For example, if you specified a `Side Clearance` value of `0.18`, the endmill might avoid reaching a wall or a boss by zigging 0.18 before reaching either one.

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 leaving stock is being performed. Smaller values should be used for finishing.

Normal-Vector Range:

These items allow you to control the toolpath calculation in order to isolate and machine flatter areas, preventing the tool from dropping down steep walls. By default, the `Minimum Angle` is `0` degrees and the `Maximum Angle` is `90` degrees. Zero degrees is perpendicular to the depth-axis (planar to the CS), and 90 degrees is parallel to the depth-axis (perpendicular to the CS). These is the minimum and maximum allowable contact angles. The toolpath is trimmed from any face that is outside this included angular range. For example, if the range is set to between `0` and `60`, then the faces that are cut all lie between exactly planar (0 degrees) through ±60 degrees from planar.

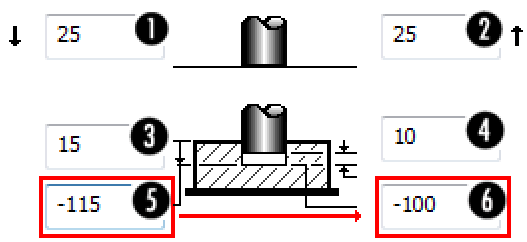
Clearances, Cut Depth, and Plunging

Clearances

The clearances in this dialog are identical to other milling processes. You should set the Entry and Exit Clearance Planes above the stock surface. The Surface Z is set so the system knows where to feed to before cutting and the Final Depth Z helps you control how far to cut.

Two other parameters become available when the Depths from Feature option button is selected:

- Incremental Feature Depth specifies the Z-extent, from top to bottom, of the entire machining feature.
- Feature Depth Z specifies the Z value of the lowest depth of the machining feature.



1. Entry Clearance Plane
2. Exit Clearance Plane
3. Surface Z
4. Final Depth Z
5. (*) Incremental Feature Depth
6. (*) Feature Depth Z

* - Available only when Depths from Feature is in effect

Depth of Cut

Z Step

This is the distance between the passes of the tool.

Cut Angle

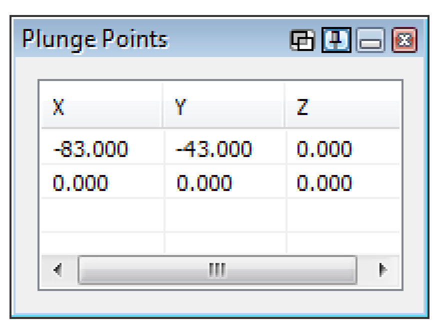
This is the primary angle at which the tool will cut, relative to CS1. Valid values are between `0` and `360` degrees, both positive and negative.

Plunging

Auto Plunge or Plunge

This menu lets you choose to let the system define where the tool will enter the material (Auto Plunge) or to define points at which you would prefer for the tool to enter the material (Plunge...).

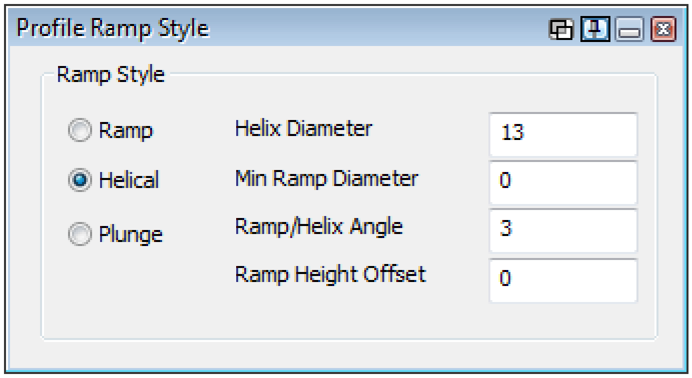
Selecting Plunge... will enable a chooser button . When this button is clicked a dialog opens which lets you select one or more points in the workspace at which the tool will enter the material. The system may filter out the points you have specified if they will cause a gouge or are not needed.



Selecting Auto Plunge means that the system will choose where to enter the material. Auto Plunge also lets you choose how to enter the material by clicking the Advanced Settings button.

Advanced Settings

This option controls how the tool will enter the material by either plunging, ramping or making a helical move into the part.



Helix Diameter

This is the size of the helix. If this diameter is too large for an area where the tool is entering, the system will use a ramp instead of a helix.

Min. Ramp Diameter

This is a minimum profile diameter for profile ramping. You do not want to ramp down along small profiles, as a very tight cutter motion would counterbalance any advantages gained by ramping for the smoothness of transition, so by setting a minimum profile diameter ("span") you will ensure that small profiles will not be ramped down.

Ramp/Helix Angle

This is the maximum angle at which the tool will ramp into the material.

Ramp Height Offset

This value is used to ensure that the tool has fully slowed down from its rapid speed before ramping into the part.

Plunge Area

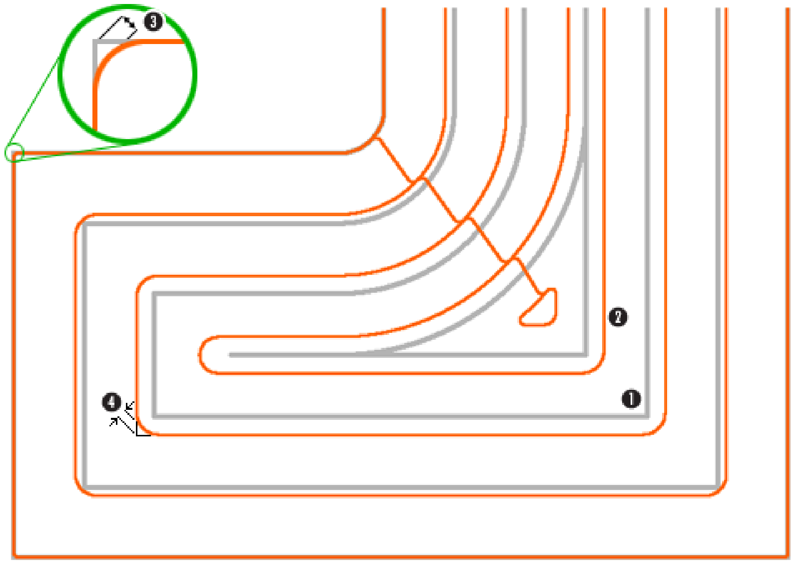
Start Hint lets you specify an X,Y location to suggest a preferred region for the operation to start, in a case where the toolpath might start in any of several different shapes (pockets or contours). The system will regard this as a suggestion: Although it will try to start the operation at or near this location, it will place higher value on avoiding gouges and crashes, and thus might start at a different location. This is illustrated in sample part `Start_Hint__Adv3D_Steep-Shallow.vnc`.

Common Process Controls

There are several controls that are used by multiple toolpath types but are not common to all processes. These items include Profile Smoothing, Cutting Strategy, Cutting Mode, and Down/Up Mill.

Profile Smoothing

The items in this section optimize the toolpath for High Speed Machining (HSM). HSM necessitates the removal of any sharp corners on the toolpath. This allows the tool to run at a higher feedrate, and it reduces cutter wear.



1. Toolpath without Profile Smoothing (the gray toolpath)
2. Toolpath with Profile Smoothing (the orange toolpath)
3. Profile Tolerance
4. Offset Tolerance

An example of using Profile Smoothing.

Profile Smoothing

Selecting this checkbox adds radii to all sharp corners within the toolpath.

Maximum Radius:

This value is the maximum radius size that can be added to the corners of the toolpath. This is applied to both internal and external corners of the toolpath.

Profile Tolerance:

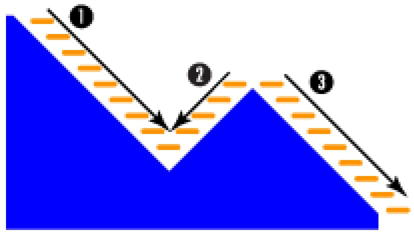
This item controls how much smoothing is added to corners. The value is the maximum distance that the smoothed profile can diverge from the actual contour profile. This value affects the final contour loop.

Offset Tolerance:

This is the maximum distance that the smoothed profile offset will diverge from the inner (offset) profiles. This parameter is identical to the Profile Tolerance:, except that it refers only to the inner (offset) profiles and not to the outer profile.

Cutting Strategy

The Cutting Strategy controls the general type of cut the tool makes. Unidirectional makes all passes with the tool in the same direction. Back and Forth alternates the cut direction where each pass is generally in the opposite direction from the previous pass. Downward forces the tool to always cut from the top to the bottom of the part, dividing up the toolpath as needed to accomplish this task. Upward will force the tool to always cut from the bottom to the top of the part, dividing up the toolpath as needed to accomplish this task. For more information, see Down/Up Mill.



An example of Downward machining seen from the side.

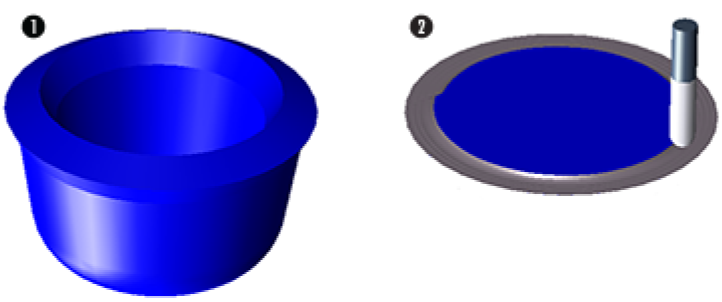
Cutting Mode

This controls whether the tool performs a Climb or Conventional cut. These options are not available when using a Back and Forth Cutting Strategy. The Inside Out option controls which direction the cut will move in under certain circumstances. With this option enabled, surfaces will be cut starting on the inside and heading to the edges, when disabled it starts at the edges and heads inward.

The “Cutting Mode” (Climb versus Conventional) can be confusing when performing cuts on parts with steep shapes.

The method Advanced 3D uses for determining climb cutting verses conventional cutting is determined by taking your shape and flattening it. With the spindle rotating in a CW (clockwise) direction and the spiral cut working from the outside to the inside; the rotation of the spiral cut toolpath should be CW (looking down the spindle towards the tip of the tool).

The image below displays the part to be cut, and the image on the right has been flattened for cutting direction clarity. This image displays a climb “Cutting Mode” with the climb cut starting on the outside and machining to the center. The tool is spinning in a CW direction and the material being removed is on the tools right hand side. Since this is a climb cut, the direction of the cutting tool will be in a CW direction (looking down the tool). The reason for flattening the shape is because the “Cutting Mode” is not determine by the 3D characteristics, but rather what side the material being removed is on in relation to the tool.



1. Original Part shape
2. Flattened shape to determine cut direction

An Example of Climb cut on a steep wall with Advanced 3D - Constant Stepover Cut.

Here is another way of looking at how the Advanced 3D determines the direction of cut. Imagine the cutting tool vector always being normal to the surface. Then think about what side the material being removed is on in relation to the cutting tool. The material is then removed in the specified

"Cutting Mode" (Climb or Conventional) depending on the rotation of your tool (CW or CCW) (clockwise or counterclockwise).

Inside Out:

When this item is enabled, the selected faces will be machined from the inside towards the face edges. When this item is not enabled the machining works from the outside towards the interior of the faces. Inside Out is available when creating a Constant Stepover Cut, a Radial or Spiral Lace Style: with a Unidirectional Cutting Strategy or a Spiral Lace Style using a Downward or Upward cut.

Down/Up Mill

These items are available when the Cutting Strategy is set to Downward or Upward. With these parameters, you can control the behavior of Downward and Upward milling.

Pass Overlap:

This item helps ensure a smoother finish. When connections between passes are broken, each part is extended by this value so that they overlap. Since both passes are extended by this value the actual length of overlap is twice the stated value.

Shallow Angle:

Angles less than the value set here are considered "shallow". Shallow passes can be machined in either direction, as up or down milling is irrelevant, and in these areas the toolpath will be less broken up. They are usually machined in the opposite direction to that produced by up or down mill (as chosen) in steeper areas.

Merge %:

Some sections of the toolpath can be machined upwards where downward movement is preferred, and vice versa, to avoid too much fragmentation. The maximum length of this opposing section is a relative percentage of the whole pass.

Maintain Milling Direction:

If selected, this checkbox ensures that all pieces will be either climb milled or conventionally milled. When this checkbox is deselected, passes will change between climb or conventional passes, depending on the relative position of the cutter at the time.

Common Options

Coolant

This menu lets you select the type of coolant to be used for the operation. Flood is available by default. A custom MDD and post processor is required for other coolant choices.

Machining CS

This menu specifies which coordinate system the part will be machined from.

Pattern

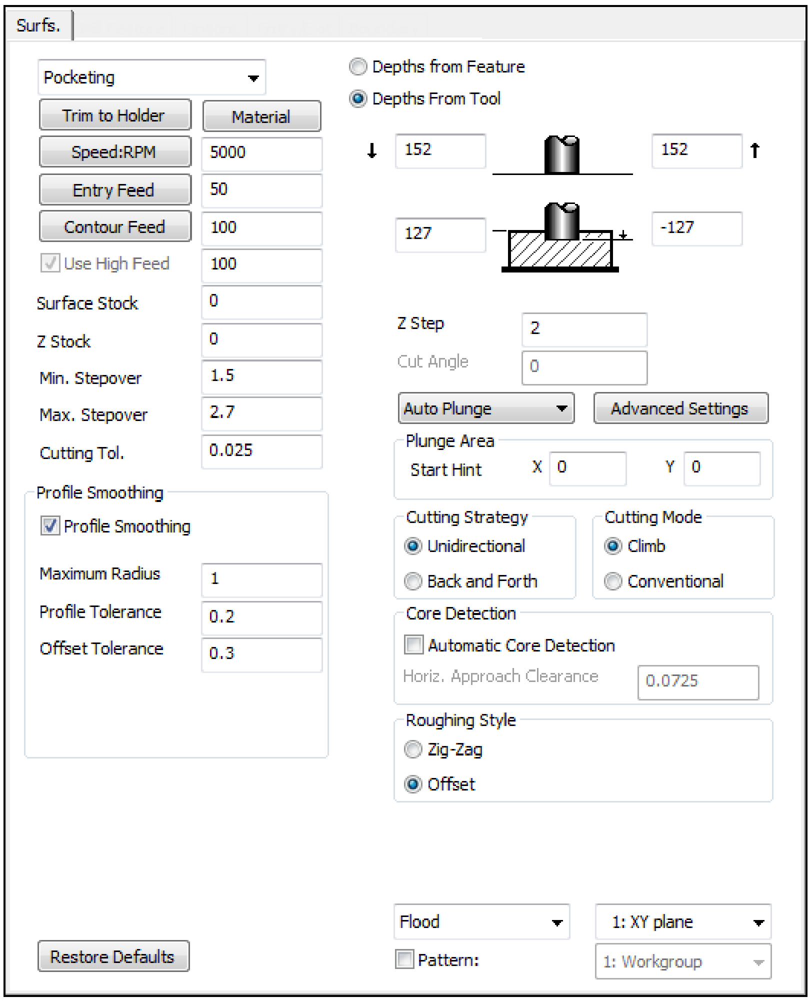
Select this option to perform a Pattern cut where the part will be machined multiple times, with the origin of the part at each point in the selected pattern workgroup.

Restore Defaults

Clicking this button will restore the Advanced 3D process parameters to the system defaults. The default values for most settings will give you the best results, and changing a value may produce unexpected results. It can be very useful to go back to the system's baseline when experimenting with values in the parameters.

Pocketing

You can use this cut type to clear large quantities of material very quickly, leaving only a minimal amount of remaining material. Some amount of remaining material is usually left behind for finish machining.



Cutting Strategy:

For a description of this item, see "Cutting Strategy" on page 126. When Roughing Style is set to Zig-Zag, Cutting Style defaults to Back and Forth.

Cutting Mode:

For a description of this item, see "Cutting Mode" on page 127. Cutting Mode is not available when Roughing Style is set to Zig-Zag.

Core Detection:

This option is useful for machining mold cores. The Pocketing cut type generally cuts from the inside to the outside of a part. This function can override that behavior for cores. This is available when Roughing Style is set to Offset.

Automatic Core Detection:

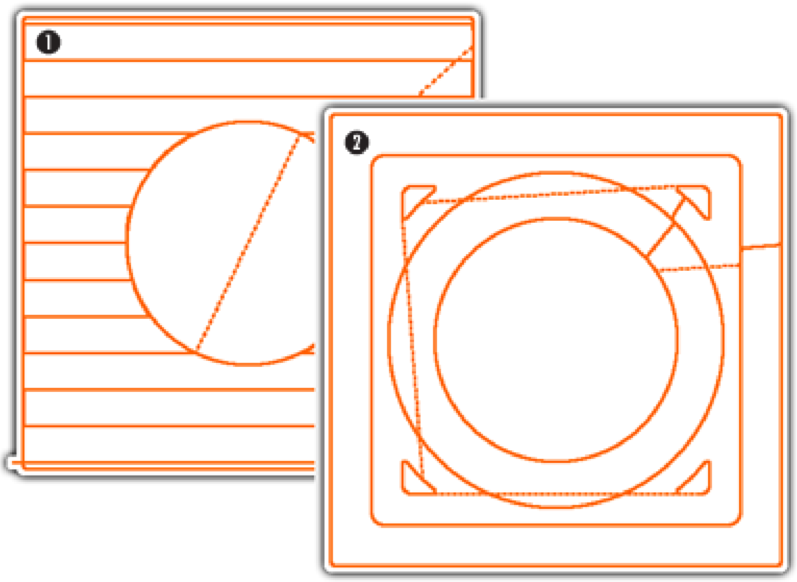
When enabled, the system analyzes the part model. If the model is determined to be a “core,” then the machining is performed from the outer areas to the inner areas of the model. If the model is determined to be a “cavity,” then the standard behavior of starting on the inside is used.

Horizontal Approach Clearance:

This is the distance off of the part at which the tool plunges before entering the material. By default, the distance is the tool radius plus the default “Step Over Clearance: ” on page 158 value. This puts the tool just off of the part. The minimum allowable value is the tool radius. Changing the default Step Over Clearance does not affect this value.

Roughing Style:

You can choose what kind of pocketing motion is used, either Zig-Zag or Offset. Zig-Zag creates toolpath that is composed of linear passes at each Z level, with one pass around the current profile. Offset creates toolpath that is composed of successive offsets of the current Z level profile. Each style has different options that are enabled or disabled in the process dialog box.



1. Zig-Zag Toolpath
2. Offset Toolpath

A comparison of Zig-Zag and Offset toolpath.

Pocketing With Core Detection

Pocketing with Core Detection is a cutting strategy for Advanced 3D milling.

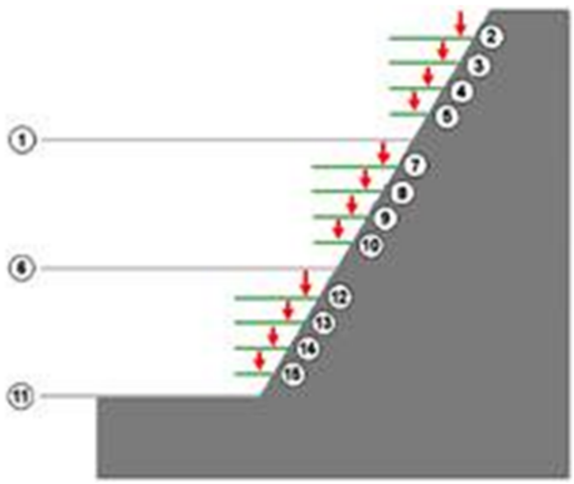
This cut type is a variant of the standard Pocketing cut type, with a few controls that are different:

- Instead of values for minimum and maximum stepover (which govern each pass in a regular Pocketing cut type), you specify values for Rough Stepover and Cleanup Stepover, which govern separate passes.
- No Zig-Zag roughing style is provided. Instead, you have finer control over offset roughing, such as the ability to restrict the number of offsets or to set a value for boundary offset.

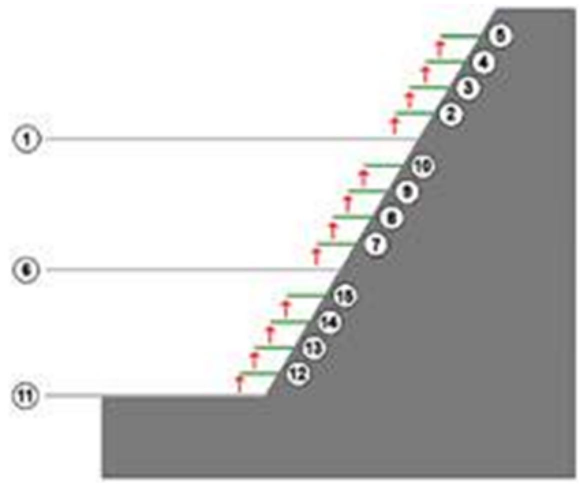
The process dialog also offers several controls under Wall Cleanup Control, as follows:

- Number of Levels: Each level is a pass at constant Z (sometimes called a *step reduction*). If you set this to `0`, the process becomes equivalent to core roughing.
- Number of Offsets: Each offset is a layer closer to the wall.
- Order in which the cleanup passes occur:
  - either During the roughing passes or After all roughing passes; and
  - either Downwards or Upwards – see diagram

Wall Cleanup Control with cleanup During roughing passes 1, 6, and 11



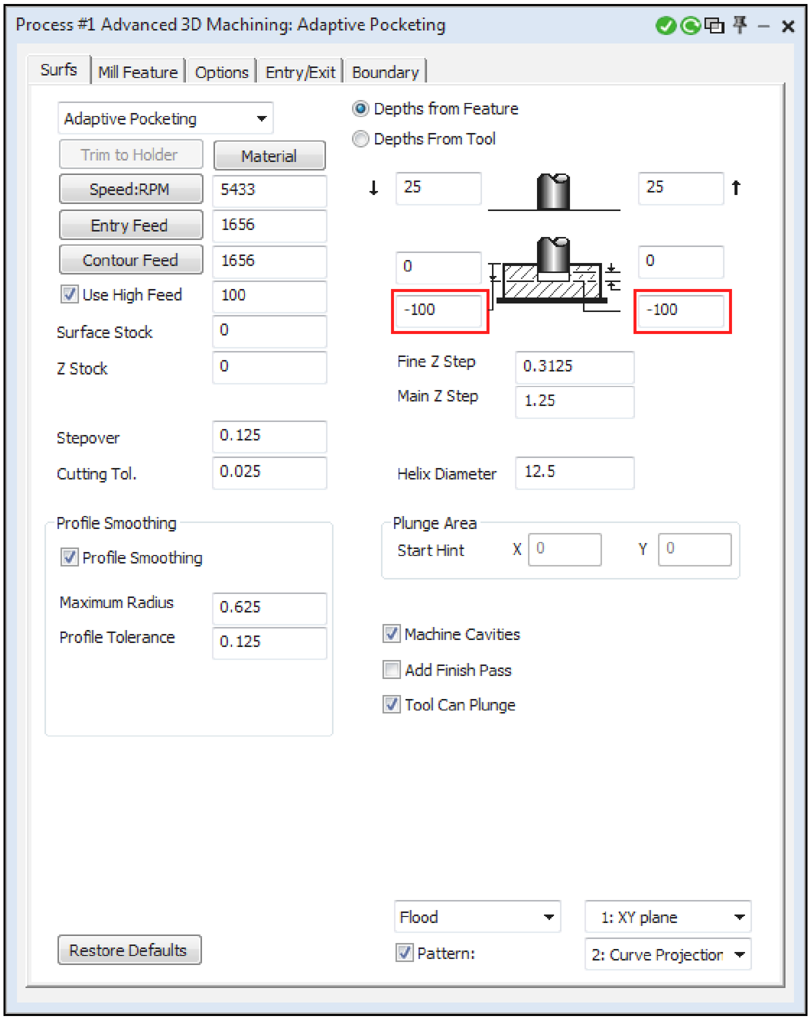
Cleanup with Order = Downwards: roughing pass 1; then cleanup 2,3,4,5 **downwards**; then 6; ...



Cleanup with Order = Upwards: roughing pass 1, then cleanup 2,3,4,5 **upwards**; then 6; ...

Adaptive Pocketing

You can use this cut type to clear large quantities of material very quickly, leaving only a minimal amount of remaining material. Some amount of remaining material is usually left behind for finish machining. The toolpath produced by Adaptive Pocketing is different from regular pocketing in being longer but smoother, with fewer abrupt changes in tool direction or tool load.



Controls for Pocket Center Clearing

The controls in this section allow you to specify the smallest pockets that will be machined.

Clear Center First

If this checkbox is selected, the system will start the adaptive pocketing toolpath from an incision down the middle of the pocket, to open up a center section first. When the checkbox is selected, this section also offers the following types of settings.

Minimum Radius

The distance from the central incision to the outer edge of the pocket must be no smaller than this value.

Minimum Angle

The angle between the central incision and the pocket boundary must be no smaller than this value.

Minimum Offset

The width of the central incision must be no smaller than this value.

Maximum Offset

The width of the central incision must be no larger than this value..

Z Step options

Main Z Step

Distance to step down (along the tool axis) for main passes.

Fine Z Step

Distance to step down (along the tool axis) for fine passes.

Machine Cavities

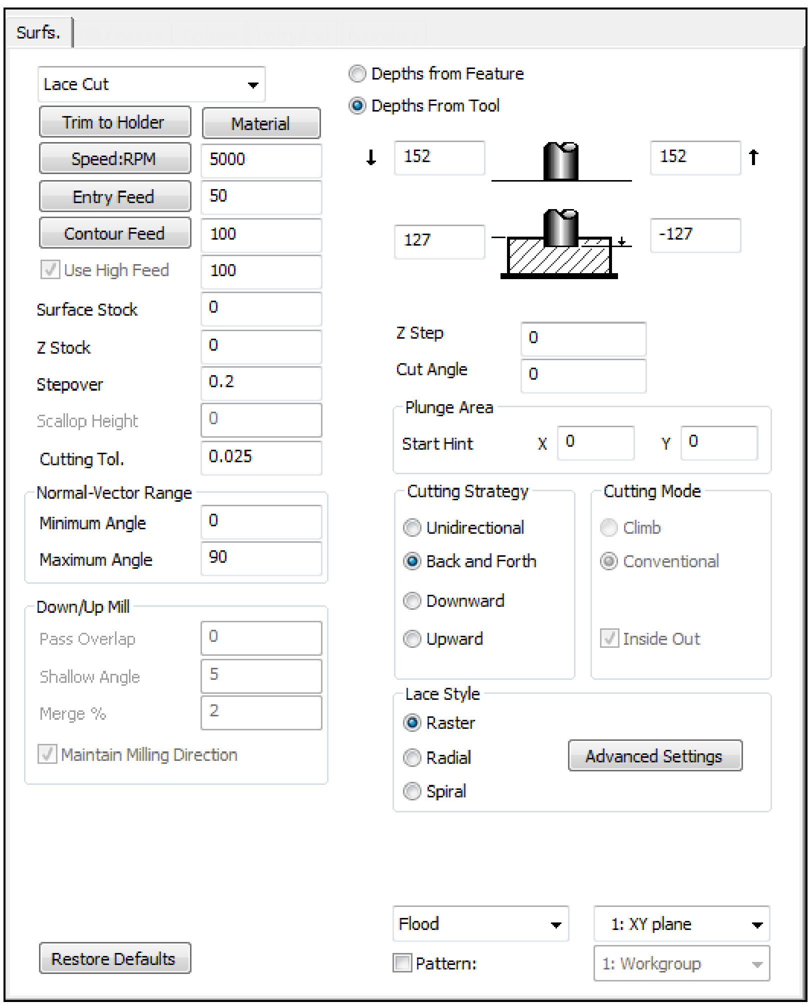
When this checkbox is selected, machining will proceed from inside to outside. When the checkbox is unselected, machining will proceed from outside to inside.

Smoothing Radius

The radius of the sharpest pass. This value will be used for clearing sharp corners and small-diameter slots.

Lace Cut

Usually, you use the Lace Cut cut type to create a finishing operation. There are several different styles of lace cuts available.



Cutting Strategy:

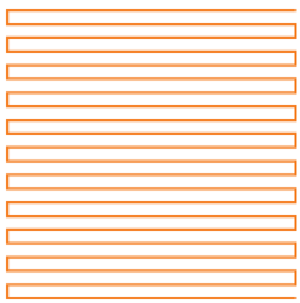
See “Cutting Strategy” on page 126 for a description of this item.

Cutting Mode:

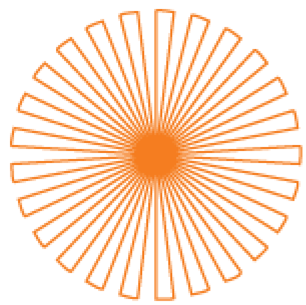
See “Cutting Mode” on page 127 for a description of this item.

Lace Style:

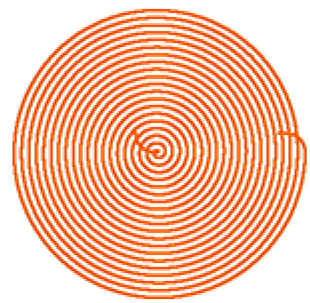
The Lace Style controls the basic form of the toolpath. Raster creates parallel passes at a specific angle with a constant stepover and may have multiple Z Steps. Radial creates toolpath that converges on a central point. The stepover of the radial passes is calculated from the circumference of the circle described in the Advanced Settings: dialog box. Spiral creates spiral toolpath that starts from a central point.



Raster Lace Cut



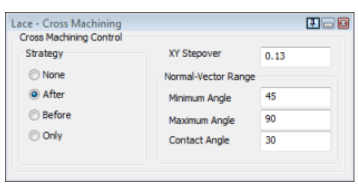
Radial Lace Cut



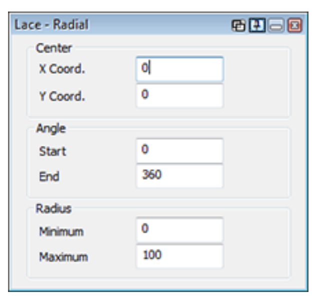
Spiral Lace Cut

Advanced Settings:

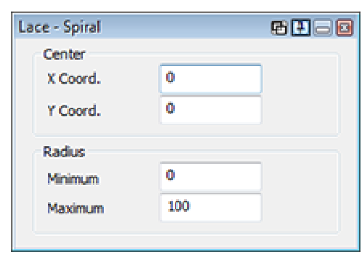
Each of the Lace Style settings has options to control the toolpath.



Raster options



Radial options

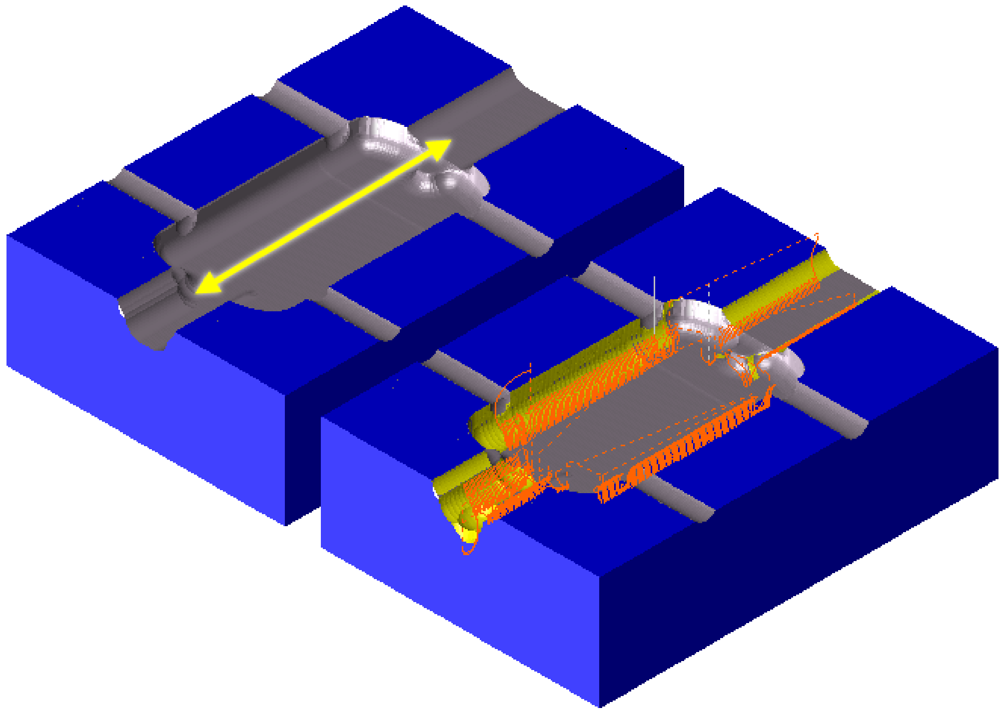


Spiral options

Cross Machining Strategy:

Cross Machining is an option to create perpendicular toolpath in order to clean up material left behind on unmachined or partially machined areas. The toolpath is optimized to access only those areas with excess material, such as walls that are parallel to the direction of cut. Cross Machining Strategy is only available for Raster toolpath. None is the default setting, no cross machining is generated. After places the cross machining at the end of the generated toolpath. Before creates the cross machining before the rest of the toolpath. Only generates just the cross machining toolpath. The system looks at what the lace cut would normally be and then generates the cross machining based on what the lace cut would be. XY Stepover describes the spacing between cross cut passes. The Normal-Vector range items let you specify the tolerance range for the contact angle of the cutter. For more information, see “Normal-Vector Range: ” on page 123.

Below is an image of a part showing the results of a lace cut (the yellow arrow shows the direction of the cut) and the resulting cross machining toolpath to clean up the areas with extra material that was left behind.



Example of Cross Machining after a Lace Cut.

Center:

This describes the convergence or central point for Radial and Spiral cuts.

Angle:

The angle defines the extent of Radial cuts. Generally these cuts are a full circle from zero to 360 degrees. Radial cuts do not need to be a full circle. You may define a custom range of radial motion by changing the Start and End values. The Start value must be less than the End value and only positive numbers are valid. That is to say, you could use the values of 180 and 270 to create a 90 degree arc but you cannot enter -90 and -180 or 270 and 180 to create the same arc.

Radius:

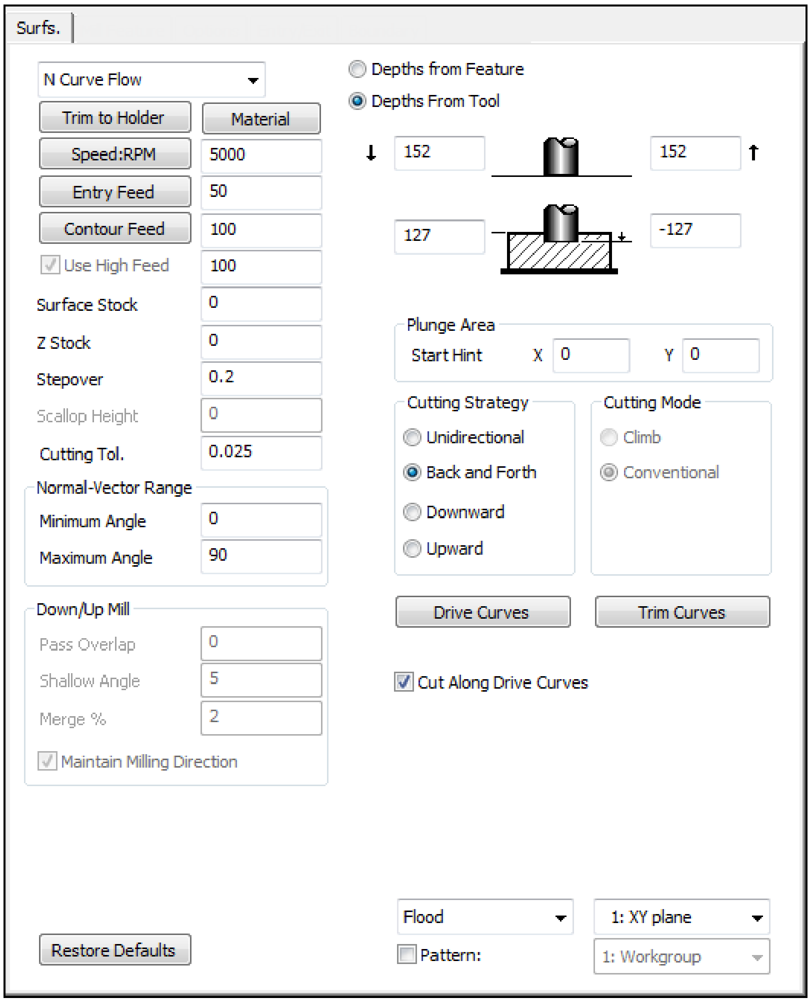
The Radius describes the size of Radial and Spiral cuts. By default the cuts go all the way to the center or convergence point of a circle but you may set an outer and inner radius to limit the area that is cut, creating a disk shape.

Down/Up Mill:

See “Down/Up Mill” on page 128.

N Curve Flow

This process creates toolpath where each pass is parallel or perpendicular to the selected curves. The toolpath morphs between the curves. This process accepts either two Drive Curves or any number of Drive Curves with two Trim Curves. The Trim Curves must be coincident with the endpoints of the drive curves.



Cutting Strategy

For a description of this item, see “Cutting Strategy” on page 126.

Cutting Mode

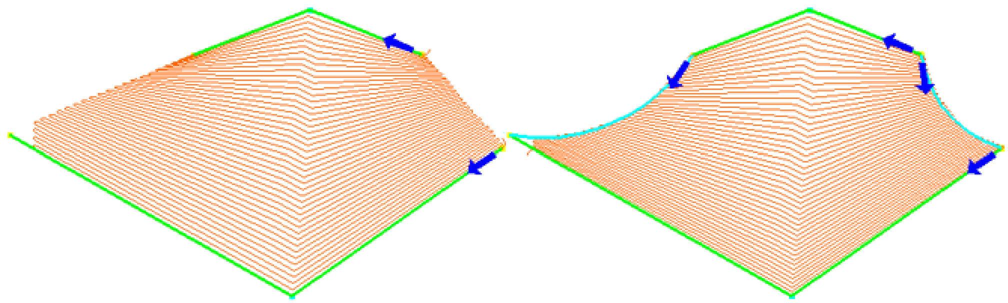
For a description of this item, see “Cutting Mode” on page 127.

Drive Curves

Clicking this button opens a dialog that allows you to choose the geometry that is the basis for the shape of the toolpath. The ends of the toolpath are automatically trimmed to the ends of the Drive Curves. This function requires a minimum selection of two Drive Curves. The Drive Curves cannot cross each other.

Trim Curves

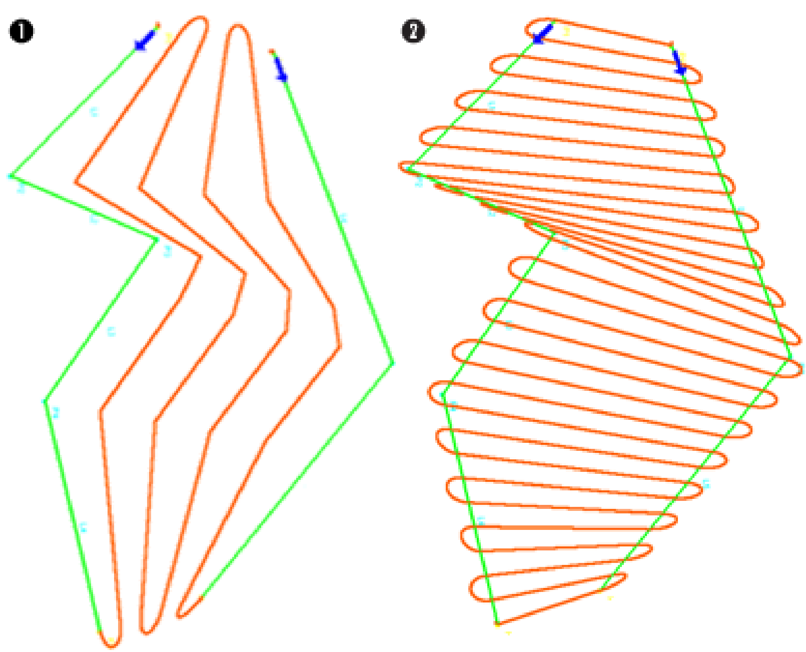
Clicking this button opens a dialog box that allows you to choose geometry that defines the border of the toolpath. Trim Curves must be coincident with the end points of Drive Curves. This function requires a maximum selection of two Trim Curves. The Trim Curves cannot cross each other. Also, the Trim Curves must pass through the end-points of the Drive Curves or terminate at those end-points.



Example of toolpath without and with Trim Curves (the curved shapes).

Cut Along Drive Curves

This item lets you choose whether the toolpath is parallel or perpendicular to the Drive Curves. When this item is selected, the system attempts to make the toolpath parallel to the drive curves. When this item is not selected, it attempts to make the toolpath perpendicular to the drive curves. The terms "parallel" and "perpendicular" are generalizations as the toolpath is morphed between the curves so the toolpath is usually not exactly perpendicular or parallel, as seen in the following images. Cut Along Drive Curves is most beneficial when only two Drive Curves and no Trim Curves are selected.



1. Cutting along the Drive Curves
2. Cutting across the Drive Curves

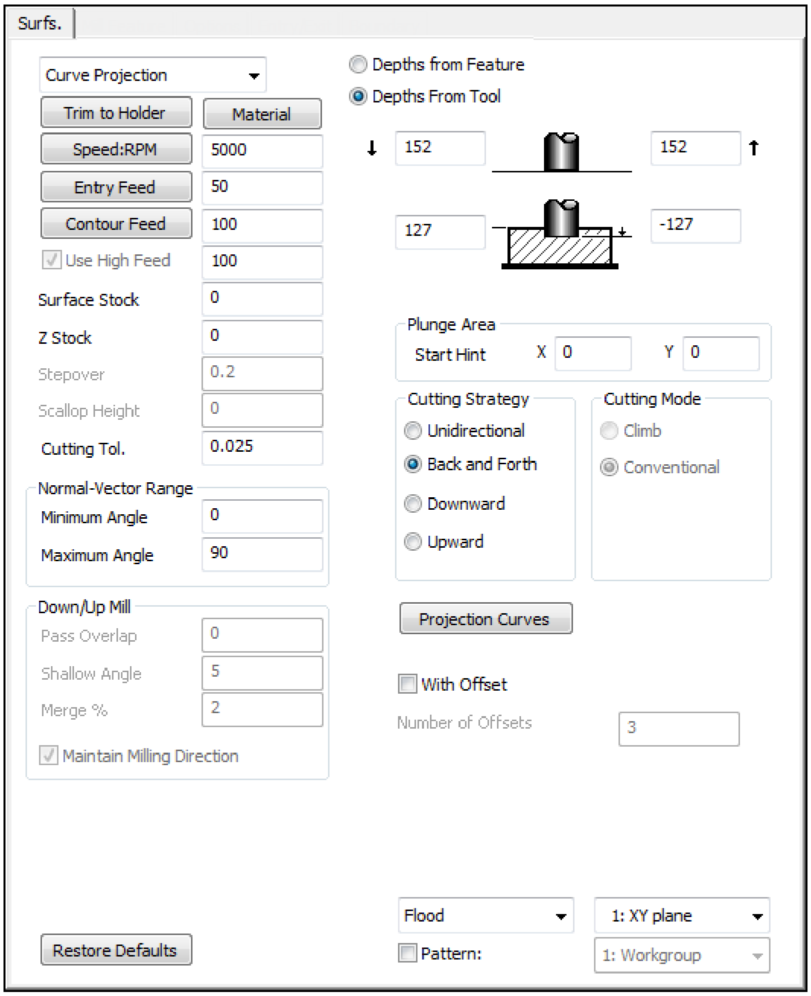
An example of what Cut Along Drive Curves can result in.

Down/Up Mill

See "Down/Up Mill" on page 128.

Curve Projection

This process creates toolpath that projects a curve (a spline) onto the faces of a solid. The toolpath is just a single Z-step down but you may perform multiple offsets to widen the area machined.



Cutting Strategy

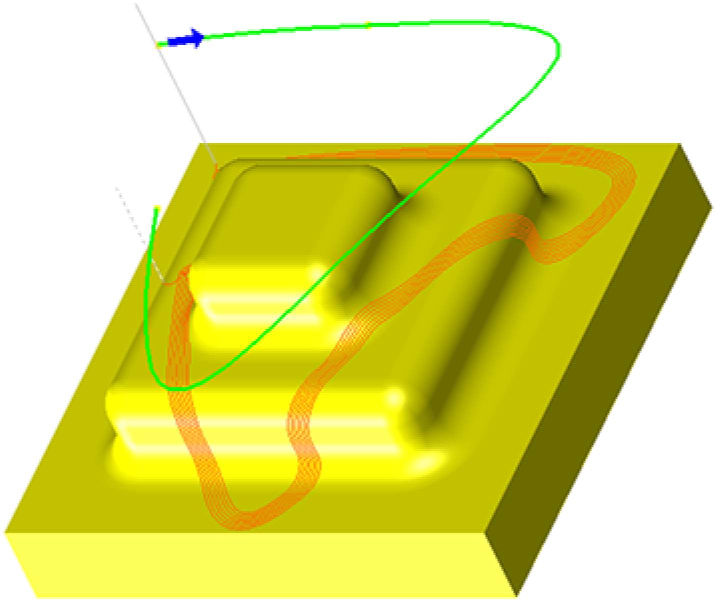
See “Cutting Strategy” on page 126 for a description of this item.

Cutting Mode

See “Cutting Mode” on page 127 for a description of this item.

Projection Curves

This button opens a dialog that allows you to select the curves that are to be projected onto the part. You may select either a single spline, a group of connected splines or multiple individual splines and/or lines and circles. The system will determine the machining order when multiple individual splines have been selected.



An example of a projected spline with offsets.

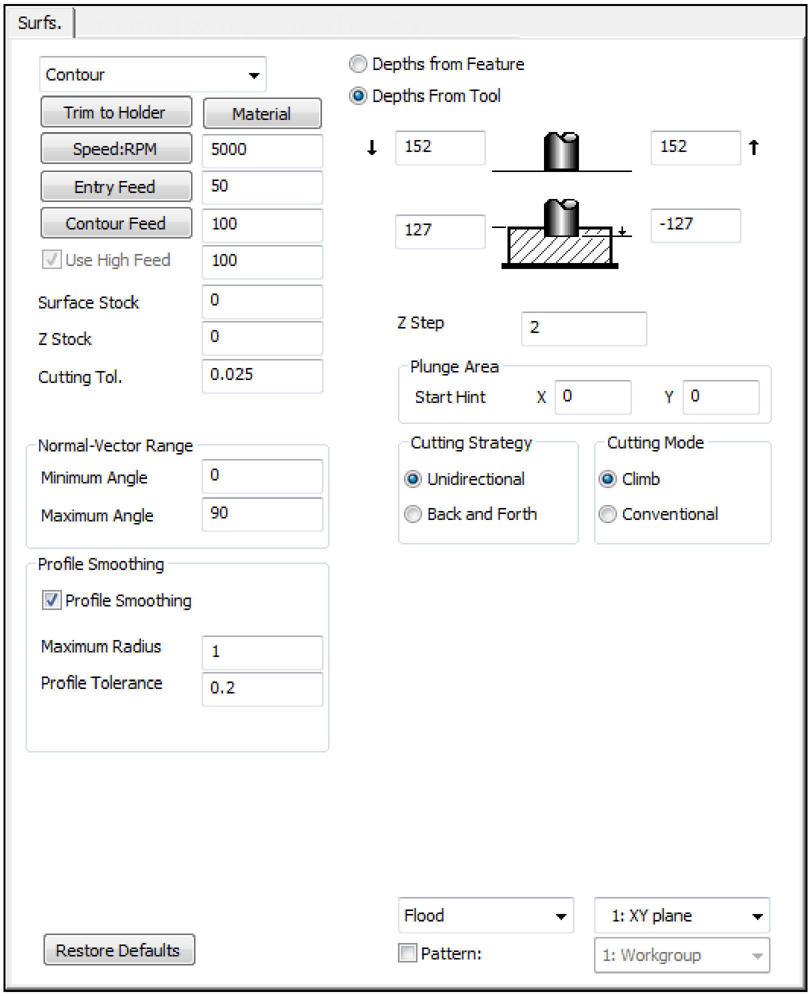
With Offset
Selecting this checkbox lets you create multiple offsets beyond the spline, creating a wider cut. The distance of each offset is equal to the XY Stepover.

Number of Offsets
This is the number of offsets on each side of the spline.

Down/Up Mill
See Down/Up Mill.

Contour

This process is typically used for semi-finishing or finishing passes and is sometimes referred to as "Waterline Machining" or "Constant Z Machining". This process is best used on parts with vertical or near vertical walls. This is because the step down is calculated in the Z-axis and the results are better on vertical or near vertical areas. The material cannot be removed as effectively on horizontal areas which would normally be machined using a different strategy.



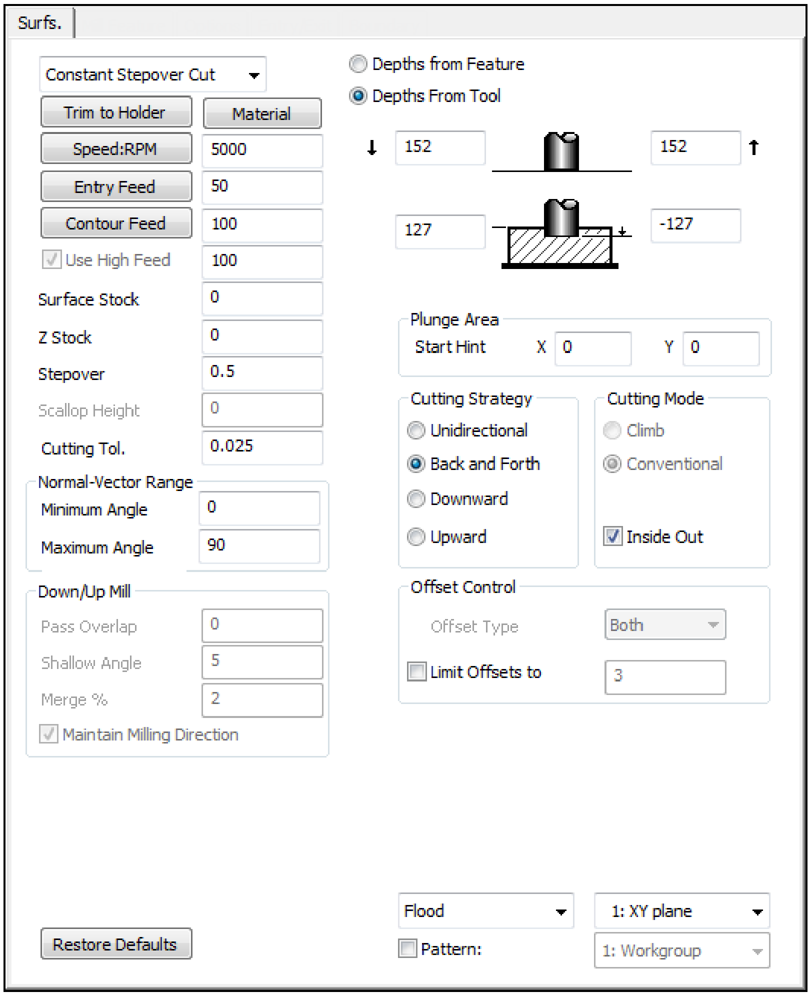
Cutting Strategy
See “Cutting Strategy” on page 126 for a description of this item.

Cutting Mode
See “Cutting Mode” on page 127 for a description of this item.

Profile Smoothing
See “Profile Smoothing” on page 125.

Constant Stepover Cut

This process type creates toolpath with a 3D stepover in all directions. This produces a very good surface finish, ensuring a constant height between every pass. This toolpath can be used as a semi-finish or a finishing operation and will often produce a much more efficient toolpath than the more traditional parallel plane and constant Z toolpath. This toolpath is very similar to a Pocketing operation using the Project 2D Toolpaths (found on the Solids tab) with one Z-step, except the Pocket op won't produce toolpath on any vertical walls and Constant Stepover Cut does. This operation type can be thought of as a constant stepover lacecut.



Cutting Strategy

See "Cutting Strategy" on page 126 for a description of this item.

Cutting Mode

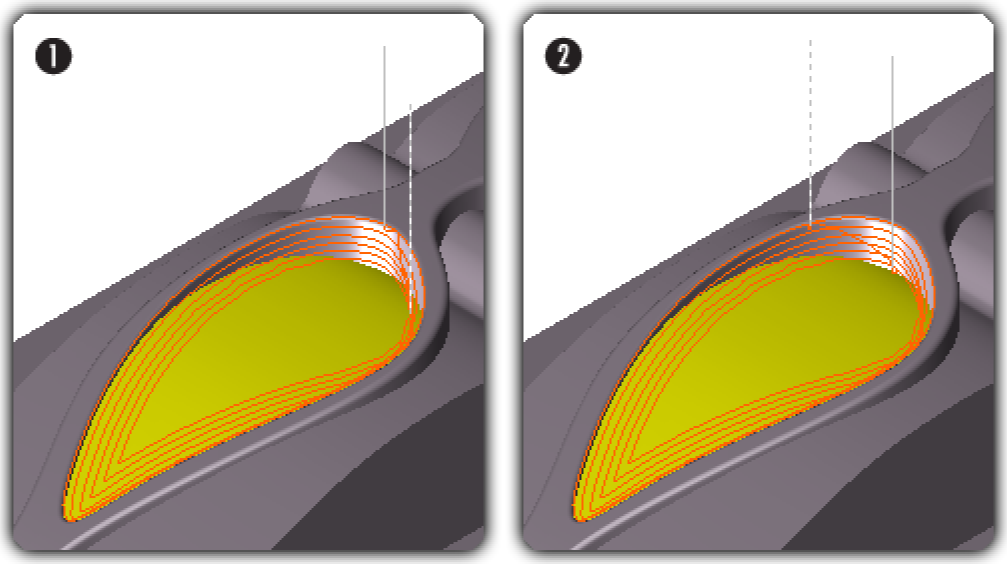
See "Cutting Mode" on page 127 for a description of this item. For the Inside Out checkbox, see below.

Down/Up Mill

See "Down/Up Mill" on page 128.

Inside Out

Performing a cut with Inside Out: enabled will start the toolpath nearest to the center of the selected face and machine toward the outer edges of the face. Alternately, performing a cut with Inside Out disabled will start the toolpath nearest to the edges of the selected face and machine toward the center of the face. See images below:



Example of the Offset Control functioning with Inside Out disabled (1) and enabled (2) on the same selection.

Offset Control

Offset Type

For Offset Type, specify whether the drive curve is an Outer curve (with offsets proceeding inward from it) or an Inner curve (with offsets proceeding outward from it).

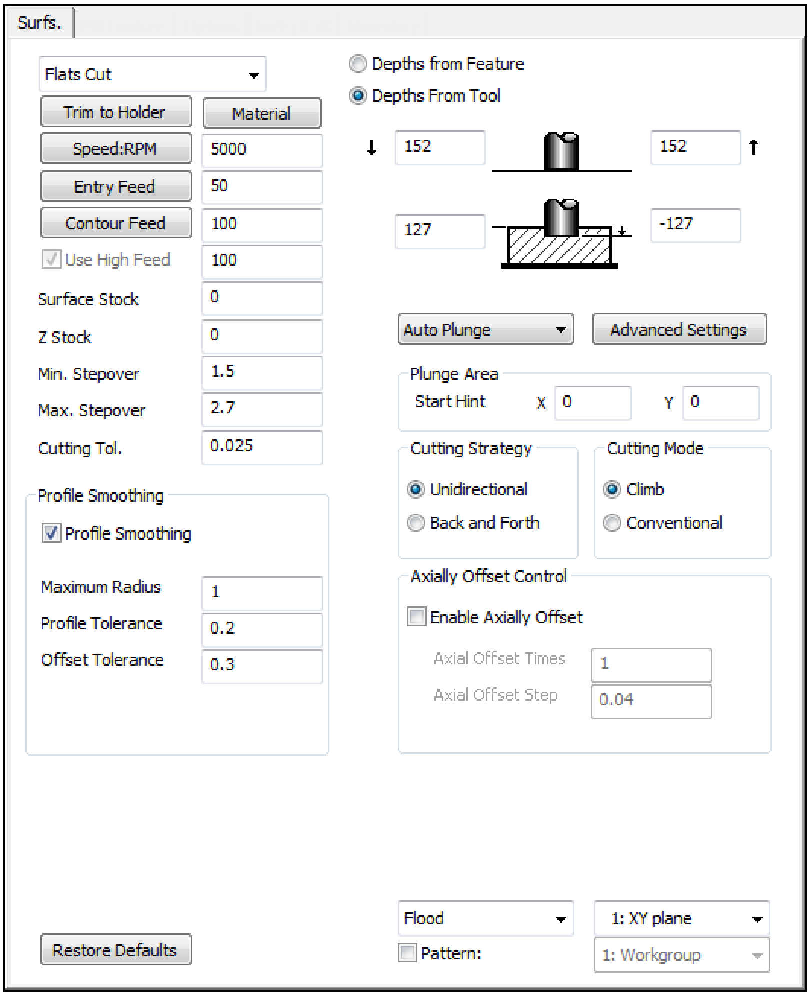
The default choice, Both, specifies a drive curve that is both an outer curve and an inner curve (with offsets proceeding in both directions).

Limit Offsets to

When this option is active, the toolpath will be limited to a specified number of stepovers from the edge of the selected face. All of the toolpath is calculated from the edge of the selected face. Performing an Inside Out cut with limited offsets could start the toolpath in the middle of selected faces and only machine to the edges of the faces.

Flats Cut

This process type machines all of the flat areas of the selected faces. This process is often used for finishing a part where excess material has already been cleared. Flats Cut acts upon completely flat areas only. Any selected face or faces that include both a flat and non-flat area will machine the flat area up to the non-flat area, where the tool's contact-point tangency touches the non-flat area.



Cutting Strategy

See "Cutting Strategy" on page 126 for a description of this item.

Cutting Mode

See "Cutting Mode" on page 127 for a description of this item.

Profile Smoothing

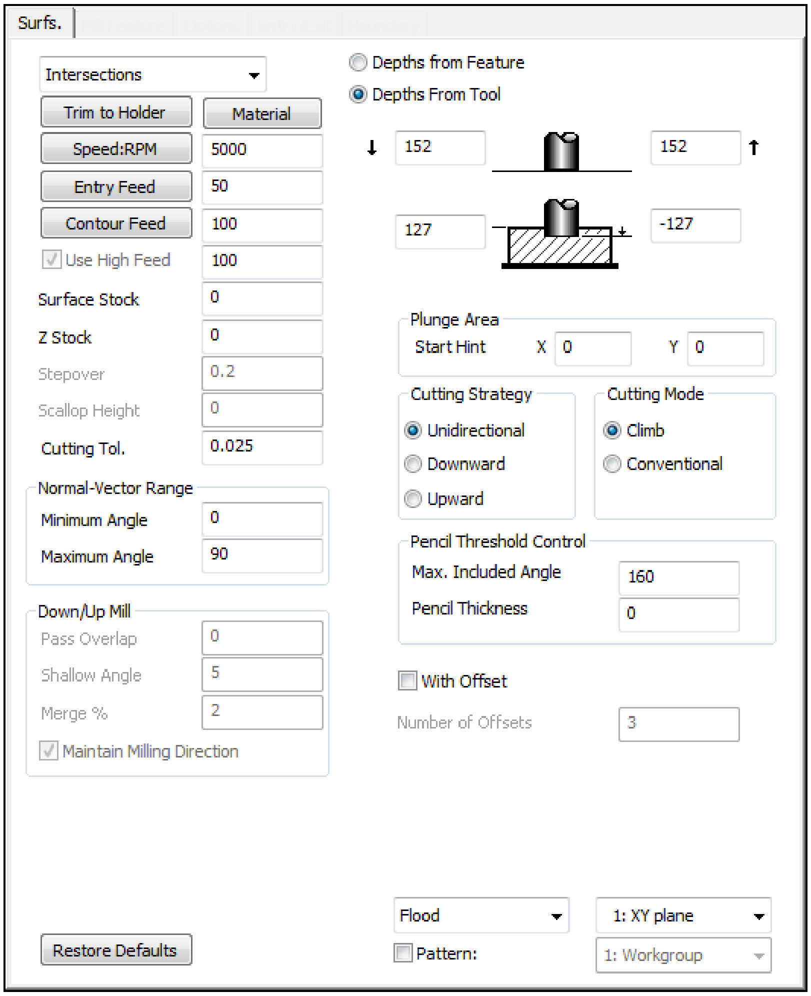
See "Profile Smoothing" on page 125.

Axially Offset Control

Enabling this item allows you to create multiple cuts along the tool axis. This effectively makes multiple Z-steps. Specify the number of times to offset the toolpath and by how much.

Intersections

This process type creates a single pass along all corners of the part. This process is intended to finish the part using pencil or rest milling after general machining of a component with a larger tool. The area to be machined is determined by the diameter of the tool selected. If there is a radius on the model and the radius value is larger than that of the tool diameter this area will not be machined. When there is extra stock to be removed or if the tool is very small, multiple consecutive vertical passes can be calculated.



Cutting Strategy:

See “Cutting Strategy” on page 126 for a description of this item.

Cutting Mode:

See “Cutting Mode” on page 127 for a description of this item.

Pencil Threshold Control:

These items define the parameters for what to machine.

Max Included Angle:

This is a special angular tolerance that you can use to control the system precision when searching for areas to machine. All machining operations have a tolerance and the smaller the value, the more accurate the calculation. The default value should be adequate for most parts.

Pencil Thickness:

This is an extra thickness that can be temporarily applied to the cutter in addition to its normal thickness. You can use this setting to make passes along fillets where the radius is greater than that of the cutter.

For example, if you have a surface with 8mm corner radius fillets and you want to create a Pencil Milling along it with a 10mm ball-nosed cutter, you can apply an Overthickness of 3 mm to it. The Pencil Milling will be made for a ball-nosed cutter of size 18 x 9 mm (which will detect this fillet), and then projected back onto the surface to make a toolpath for the 10 x 5mm cutter.

With Offsets:

Enabling this option creates more than one pencil trace.

Number of Offsets:

This field lets you control how many times the pencil trace is offset. Each trace is offset by the XY Stepover.

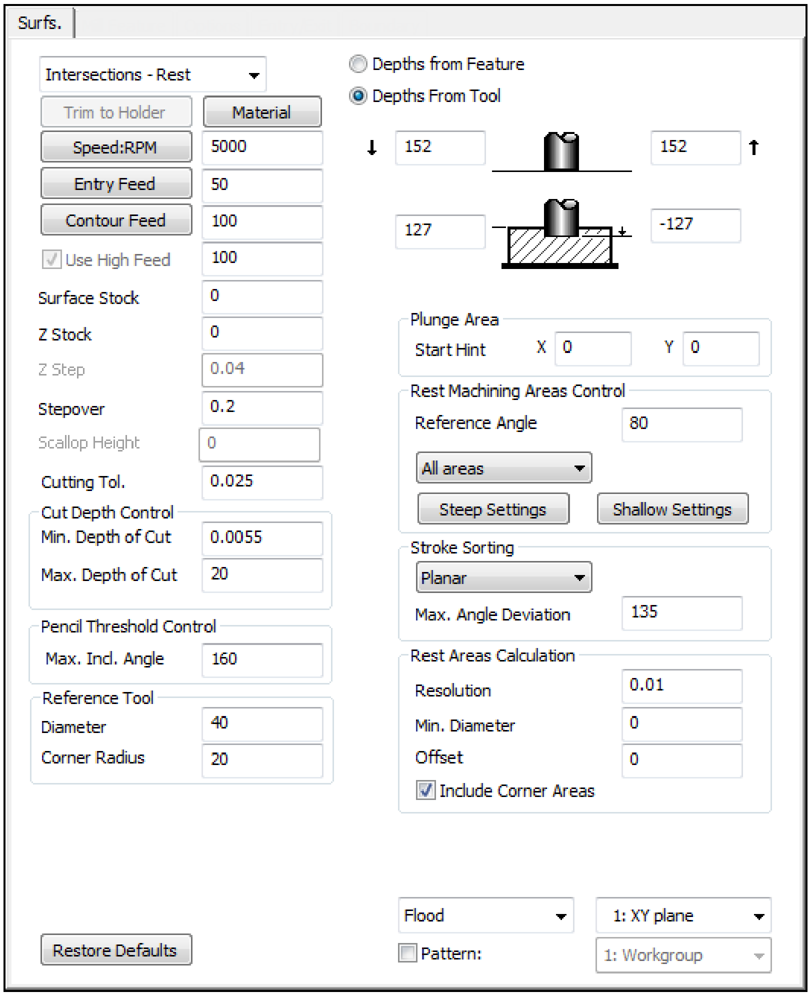
Tips for working with Intersections toolpath:

There may be times when an Intersections operation creates toolpath in areas that may not be desirable. Parameters that will particularly help alleviate this are Max Included Angle: and Cutting Tolerance: . The larger the included angle, the more “noise that can be included in the toolpath and the tighter the tolerance is, the more accurate tessellation will be which also helps minimize “noise”.

Intersections - Rest

This process type creates toolpath that removes all the un-machined areas of a hypothetical previous tool. This is basically a pencil milling strategy calculated using a Reference Tool diameter. If there are radii on the model with values equal to or greater than the Reference Tool radius, these areas are not machined. This allows for fast toolpath calculation and good finishing.

You can also use boundaries to influence this calculation. The boundary, if used, is interpreted as an external shape and is used to trim the passes.



Rest Machining Areas Control

The items in this section help you define the areas to be cut, focusing on only steep or shallow areas if that is what you need. The default setting is All areas. The pull-down lets you choose which area you will machine.

Reference Angle

This value lets you define an angle that splits the Steep areas from the Shallow areas. A value of 0 means that all passes are seen as steep and will be machined along the corner (Image 1). A value of 90 means all passes are seen as shallow and will be cut across the corner (Image 2). A setting of 45 will cause areas between 0-45 degrees to be machined along the corner while areas between 45-90 degrees will be machined across the corner. (Image 3)

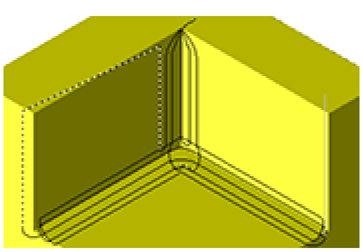


Image 1

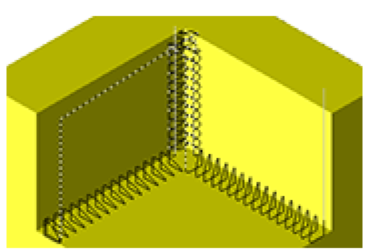


Image 2

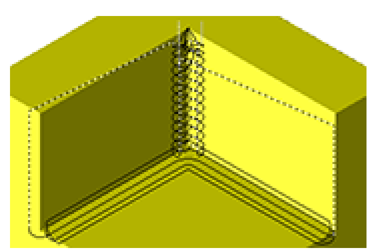
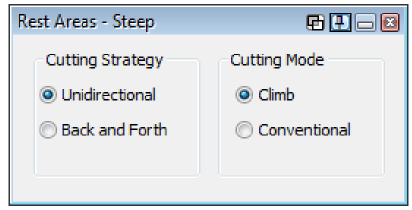


Image 3

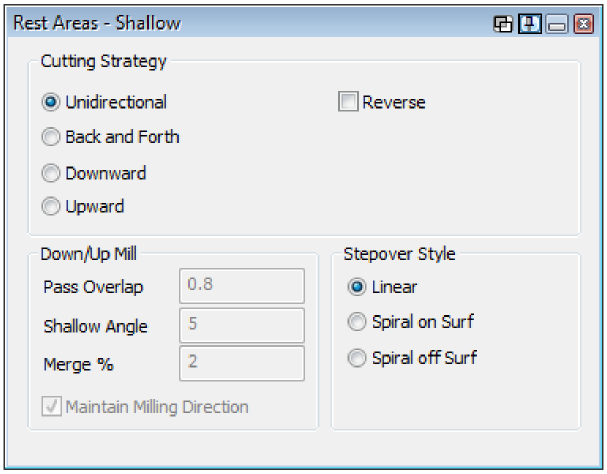
Steep Settings:

Clicking on this button lets you define how “steep” areas will be machined. You can set whether the cuts will be Unidirectional or Back and Forth and you can set whether the cuts will be Climb or Conventional.



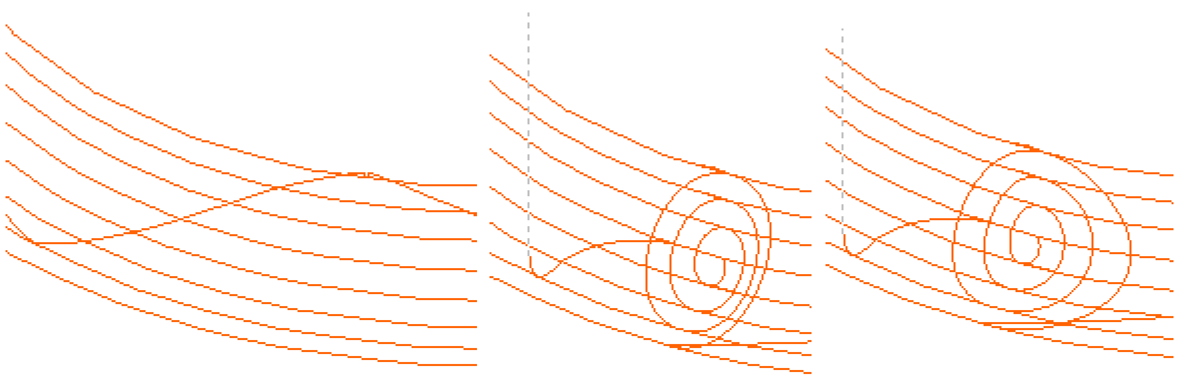
Shallow Settings:

Clicking on this button lets you define how “shallow” areas will be machined. You can set whether the cuts will be Unidirectional, Back and Forth only Downward or only Upward. Checking the Reverse box will change the machining direction.



Stepover Style:

This option controls the stepover (the connect or link) move from one pass to the next pass, as with connect moves from one Z-step to the next.



Example of a Linear Stepover | Example of a Spiral On Stepover | Example of a Spiral Off Stepover

Linear:

The stepover is made in a linear (line-type) motion. The linear move leaves the surface at the end of one pass and moves through the air toward the start of the next pass, where it then touches-down at the start of that next pass.

Spiral on Surf:

The stepover is made in a helical motion. The helical motion always maintains contact with the part surface, that is, it never moves up into the air.

Spiral off Surf:

The stepover is made in a helical motion. The helical motion leaves the surface at the end of one pass and moves through the air toward the start of the next pass, where it then touches-down at the start of that next pass.

Down/Up Mill:

See Down/Up Mill for a description of this item.

Stroke Sorting:

The items in this section allow you to organize the pencil pass strokes to get more connected and uniform machining passes.

Planar:

With this technique the system looks at the pencil passes from the tool axis, i.e. from +Z, and connects passes that have an angle deviation less than the value set by the Maximum Angle Deviation: .

Angular:

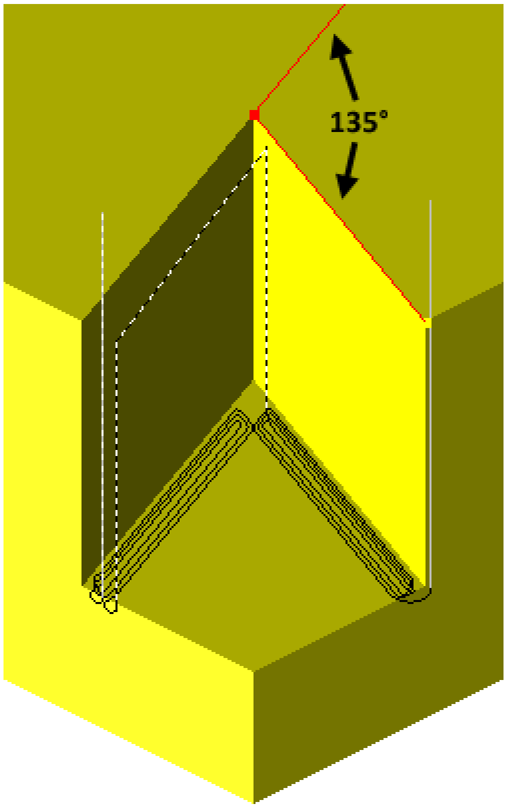
With this technique the system looks at the pencil passes from any 3D point of view and connects the passes with an angle deviation less than the value set by the Maximum Angle Deviation: .

None:

This technique will not optimize the toolpath but will group strokes and converge their ends to a point. Looking at a corner, all of the passes in the vertical edge will be grouped as will all of the passes along the left horizontal and right horizontal edges. This technique may leave uncut material on the corners or create excess toolpath at convergence points.

Maximum Angle Deviation:

The Maximum Angle Deviation is used to control the splitting of strokes. If an angle of a corner is greater than this value (Image 1, below), the strokes will be split into multiple sections. If the angle is less than or equal (Image 2, below) then the strokes will be grouped together.



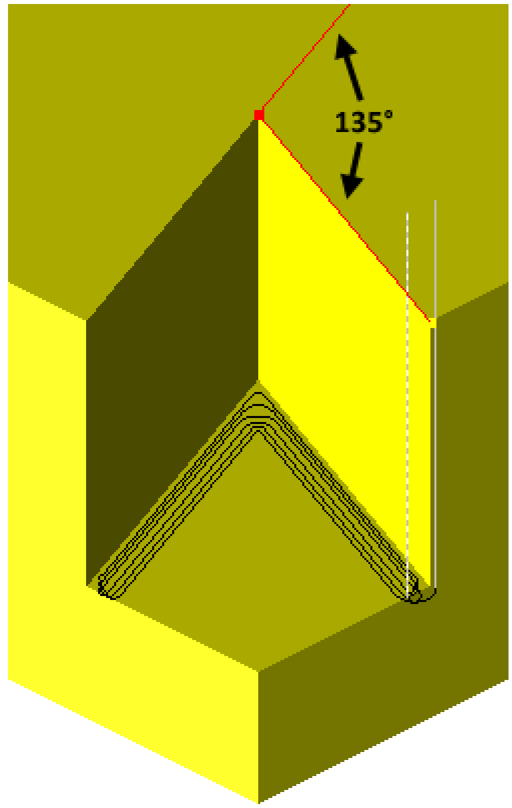


Image 1: Set to 134 degrees

Image 2: Set to 136 degrees

Rest Area Calculation:

The items in this area are only available when using a flat or bullnose endmill.

Resolution:

This value is used in calculating the boundary for the tool and the Rest Intersections toolpath. The smaller the value, the finer the detail of the boundaries. Small values will result in longer calculations.

Minimum Diameter:

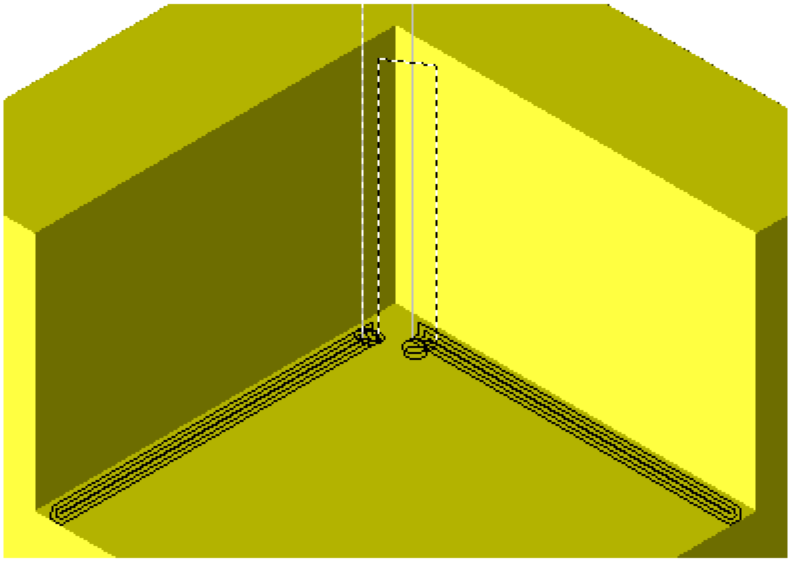
This value is used to remove areas from the calculations. Any area that is smaller than this value, i.e. a diameter or point-to-point, will not be used when calculating the boundary for what is to be rest milled.

Offset:

Calculated boundaries can be offset by this amount. Without adding an offset the exact rest area is calculated, which can leave marks or cusps. Adding a small offset smooths the rest area and can help minimize jagged edges.

Include Corner Areas:

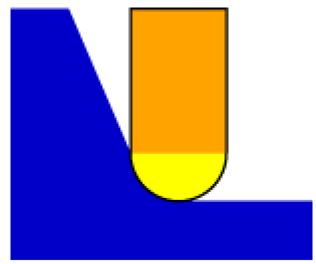
Corner areas can be left in boundary area calculations or left out by deselecting this option.



Cut Depth Control:

The items in this section control how much material will be removed with each pass. These controls can be very important for maintaining tool life.

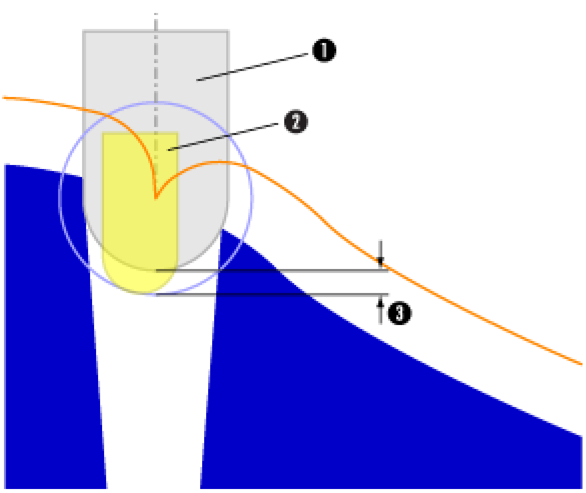
Minimum Depth of Cut:



This item controls the minimum amount of material to be removed from the areas to be machined. It is very useful for resolving situations where the blend radius of the model is roughly equal to the reference radius because in theory there is no material to remove, as seen to the right. Normally in this situation the system creates pencil passes but the rest material passes will not be made to avoid creating unnecessary rest machining passes. The recommended value for this parameter is `[Cutting Tolerance] * 1.1`.

Maximum Depth of Cut:

This parameter allows you to set the maximum Z height of material to be removed. This parameter is very important for preventing damage to the tool when machining hard material by ensuring the tool does not make too heavy of a cut. The default value should be 1/3 of the tool radius and we recommend that you do not set this value greater than the tool radius. The image below shows how the system controls the cutting depth using this parameter.



1. Reference tool
2. Current tool
3. Maximum depth of cut

Please note that there may be rest material that is not completely removed in one operation but attempting to machine all of the rest material in one step risks breaking the tool. In a situation were one rest machining operation does not remove all of the material, create one or more rest machining operations, each with a progressively smaller Reference Tool: .

Pencil Threshold Control:

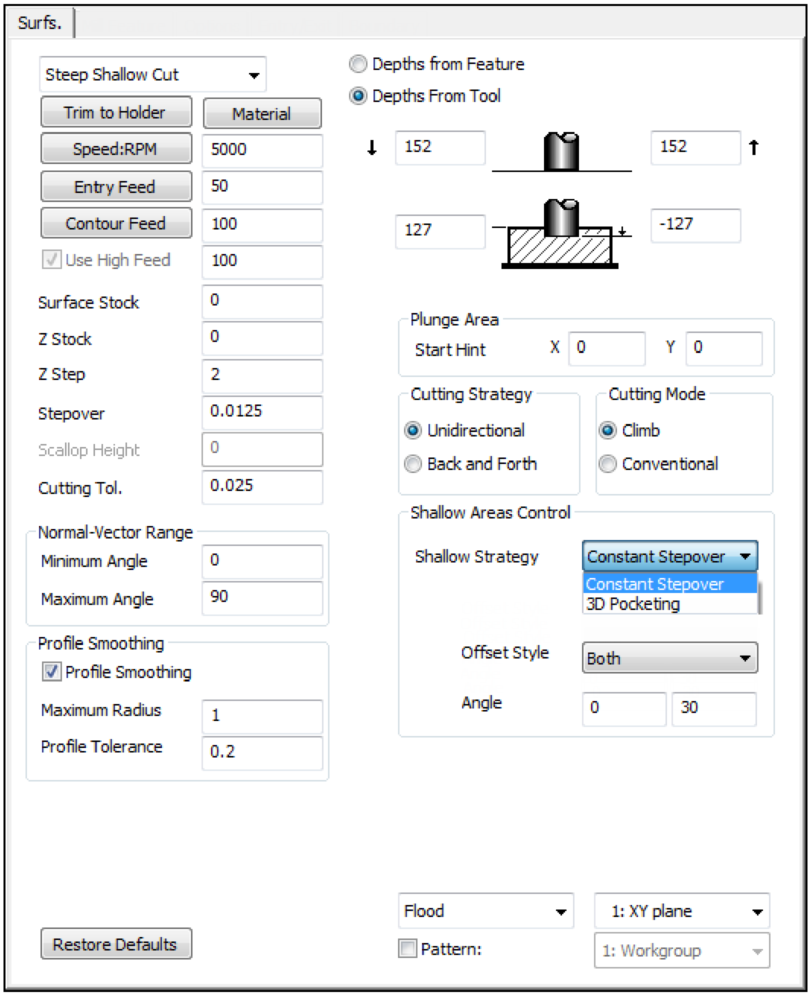
Any faces with an included angle beyond the Maximum Included Angle are not used for an Intersections operation. With the default of 160, the faces must be almost flat to be ignored.

Reference Tool:

The Reference Tool is a hypothetical tool that has run over the surface of the model to be machined. From the reference tool size and the model topology, the system can derive a theoretical remaining material condition. It is these areas that are machined by a Rest milling operation. Simply enter the Diameter of the hypothetical tool. Any corner smaller than the displayed Corner Radius is potentially machined by this process. If this operation is being created with a flat or bullnose endmill, then you must also enter the Corner Radius of the hypothetical tool.

Steep Shallow Cut

This process type creates toolpath that machines Steep areas and Shallow areas using different strategies. This is a “one-step” finishing routine. Very complex parts can be finished using just this process because you can define an angle that splits the Steep areas from the Shallow areas and thus define different machining strategies for each zone.



Shallow Areas Control

The results of Shallow Area Control are dramatically affected by the setting of "Cutting Strategy" on page 126. The results are typically seen when using Unidirectional. Shallow Area Control affects the resulting behavior of the Options tab item, "Connect Move Control " on page 158. When using Back and Forth, the toolpath stepover moves are typically connected with a non-retracted, linear stepover. But when using Unidirectional, the toolpath typically requires retracted moves from the exit move of one toolpath segment to the following start move of the next toolpath segment.

Additionally, the Offset Style is directly affected by the angular range values input into the Angle fields, of Shallow Area Control. For instance, if the minimum angle is set to 5 degrees and the selected part faces have angles less than this 5 degree angle (for example, 0 degrees planar), then toolpath is not created on the part faces with less than the 5 degree value. Therefore, connect moves are not needed in those areas.

Shallow Strategy

This controls the variation in stepover in shallow areas.

When set to Constant Stepover, the stepover will be constant. You can fine-tune the toolpath using other parameters, such as Normal-Vector Range, Offset Style, and Angle.

When **Shallow Strategy** is set to 3D Pocketing, the software will vary the stepover according to the degree of shallowness, given the conditions specified elsewhere in the process. Because the 3D Pocketing algorithm is simpler – it offsets in constant Z and then projects the pattern onto the part – you can use it to produce toolpath much more rapidly.

Offset Style

This controls how stepovers are created when the toolpath moves from one Z-level to the next. This option controls the specified "Stepover: " on page 122 distance from the outermost pass of one Z-level to the outermost pass (start) of the next Z-level, regardless of whether the Z-levels are steep or shallow. This is most noticeably visible on a part like a stepped block, as described below.

Both

The toolpath shape on the shallow areas is centered in XY between each Z-level and then the toolpath stepover shape is offset from the outer edges of the part's face at that Z-level. It is not centered on the face of each Z-level. Additionally, where the outermost loop of toolpath on an upper face overlaps a face below, that upper loop is offset from the outermost loop (the loop below it) on the lower face by the stepover value. For example, if the stepover value is 1MM, the upper loop is positioned 1MM away, in XY, from the lower loop.

Upper

This adjusts the toolpath on the upper shallow faces to be offset from the toolpath on the lowest face. For example, the toolpath is first calculated on the lowest face so the stepover shape is offset from the outer edges of the lowest face. The next shallow face above (upper) is then calculated and the resulting offset shape is a mixture of the lower face toolpath and the current face edges. If there are additional upper faces, the offset shape continues to change at each face. Additionally, where the outermost loop of toolpath on an upper face overlaps a face below, that upper loop is offset from the outermost loop (the loop below it) on the lower face by the stepover value. For example, if the stepover value is 1MM, the upper loop is positioned 1MM away, in XY, from the lower loop. Note that the toolpath actually machines from Z-positive to the Z-negative.

Lower

This works in reverse to "Upper", with the exception of the following: where the outermost loop of toolpath on an upper face overlaps a face below, that upper loop is offset from the outermost loop (the loop below it) on the lower face by the stepover value. For example, if the stepover value is 1MM, the upper loop is positioned 1MM away, in XY, from the lower loop. Note that the toolpath actually machines from Z-positive to the Z-negative.

Angle

The Angle values allow you to define what is shallow. The two text boxes define the minimum and maximum values for faces that are "shallow". Any face that is beyond this angle is considered "steep".

Cutting Strategy

For a description of this item, see "Cutting Strategy" on page 126.

Cutting Mode

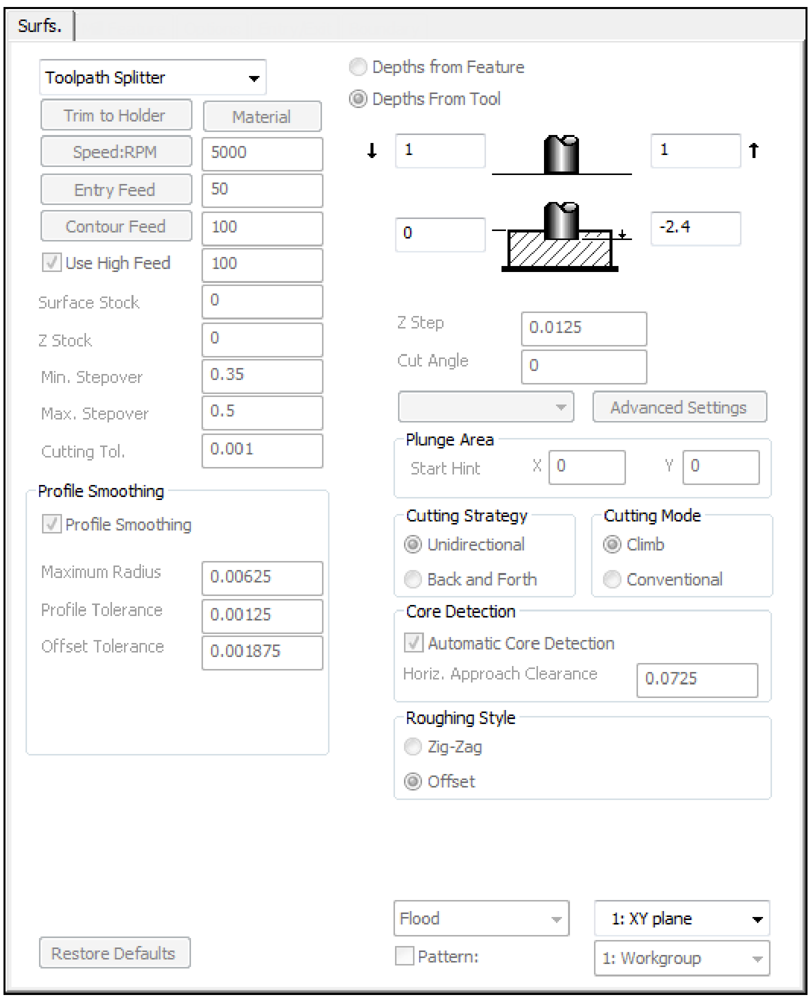
For a description of this item, see "Cutting Mode" on page 127.

Profile Smoothing

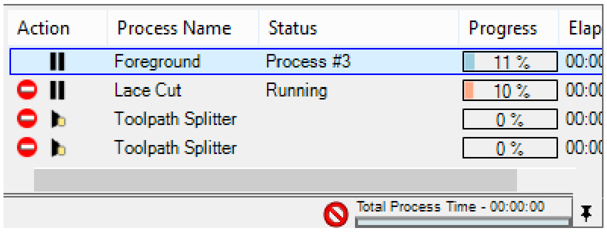
For an explanation of these items, see "Profile Smoothing" on page 125.

Toolpath Splitter

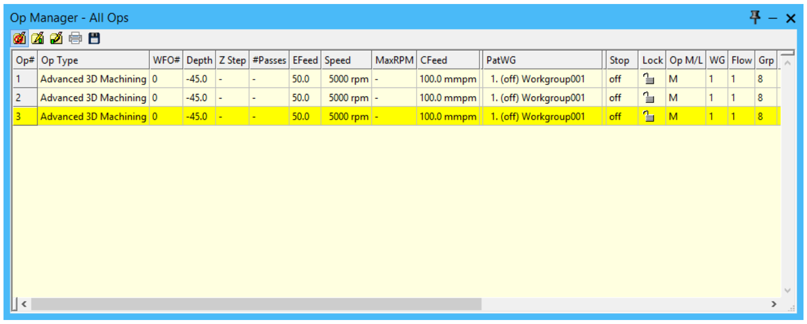
This process type is for splitting up very long toolpath. This is intended as a method to force a tool change, saving the cutter. To create this process type you must start with an Advanced 3D process. You must also define one or more additional tools that are geometrically identical to the one used in the process. Place the additional tool tile in the process list and drop an Advanced 3D process onto the tool and select Toolpath Splitter for the cut type. The process dialog will be entirely grayed out as all of the information is identical to process #1. The system will generate the toolpath and then split the toolpath into two separate operations. The operations will be approximately the same length, with a split being at the end of a feature so as to prevent witness marks or scrapes from the tool retracting and a new tool coming in. To split the toolpath even more, create more tools and processes, e.g. 5 tools and process will create the same toolpath with each operation being about 20% of the total toolpath.



The split processes will be blocked from being generated while the bulk of the toolpath is being generated.

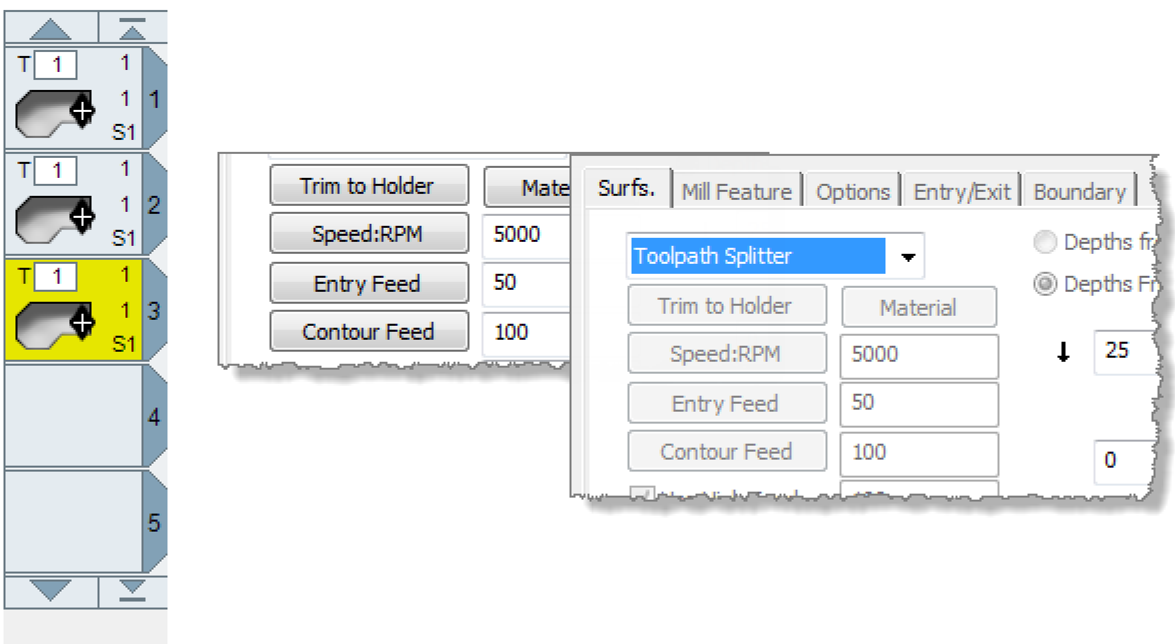


Once it is done, you can see how the toolpath is split up. The operations may not be of exactly the same length as the model topology may not allow a precise division and keep optimal toolpath.



Toolpath Splitter process example

There are three processes in the process panel. The first one is a Lace Cut. The second and third ones are Toolpath Splitters which can only be attached to the first one which is a base process. The Toolpath Splitter use the same GUI as the base process except that the controls are all grayed out. The Toolpath Splitter is only enabled when the process is using the same tool as the base process and the process is not the first process (base process) in the process group. When executed, 3 operations will be created and the toolpath created from the base process will be split into three parts and saved into the three operations respectively.



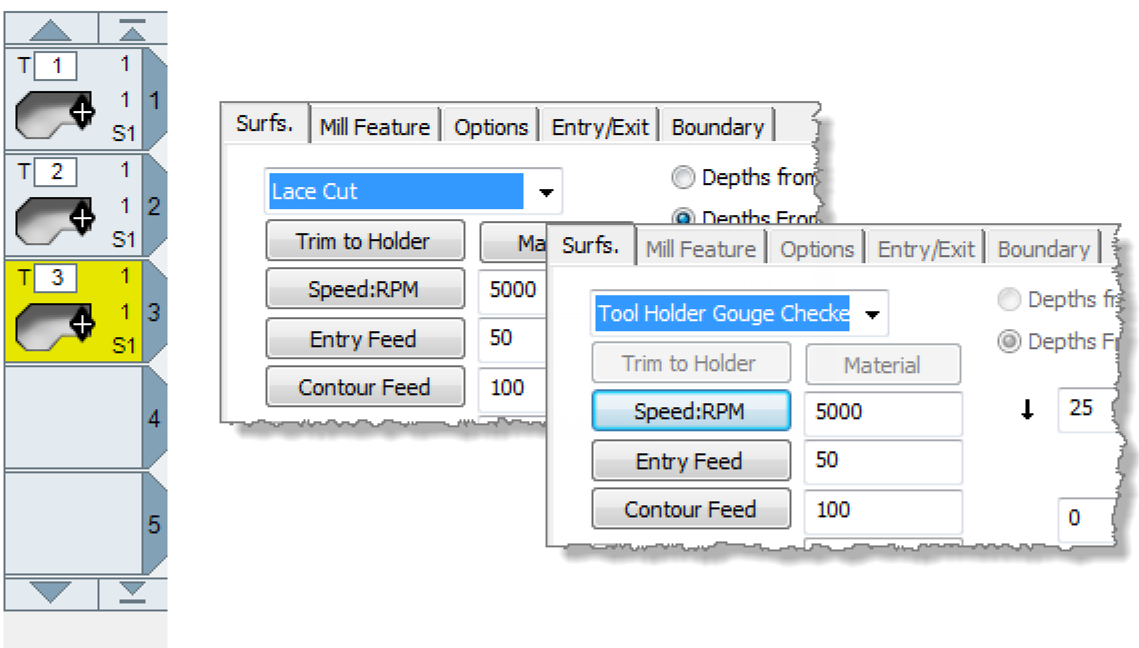
Tool Holder Gouge Checker

This process type splits up the toolpath of another process type. The toolpath is split based upon where the tool and its holder can fit. To create this process type, you must start with an Advanced 3D process. You must also define one or more additional tools that are identical to the one used in the process except for the tool length or Length Out Of Holder. ***Warning:*** If the value for Length Out Of Holder is too short, the final process of type Tool Holder Gouge Checker will gouge the part with the holder. This is because the gouge checking is applied to the process before it, and the remaining "gouging" toolpath is applied to the Tool Holder Gouge Check process with gouge checking turned off.

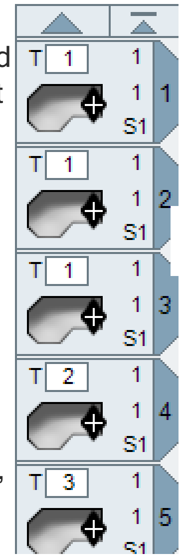
Place the additional tool tile in the process list and drop an Advanced 3D process onto the tool and select Tool Holder Gouge Checker for the cut type. The process dialog will be entirely grayed out, as all of the information is identical to process #1. The toolpath for each operation will be based on the depth to which the tool can go.

Tool Holder Gouge Checker process example

There are three processes in the process panel. The first one is a Lace Cut (using tool 1 which is defined with a tool holder) which is also a base process. The second and third ones are Tool Holder Gouge Checker processes which are dependent on the Lace Cut (the base process). Tool Holder Gouge Checker must use different tools but with same tool parameters except for a different Length out of Holder. The toolpath of the Lace Cut process is gouge checked against the Tool Holder in tool 1. The gouge free part is left in the Lace Cut operation corresponding to the Lace Cut base process. The left over toolpath is gouge checked against Tool Holder in tool 2 and so on.



Example of Toolpath Splitter and Tool Holder Gouge Checker used together

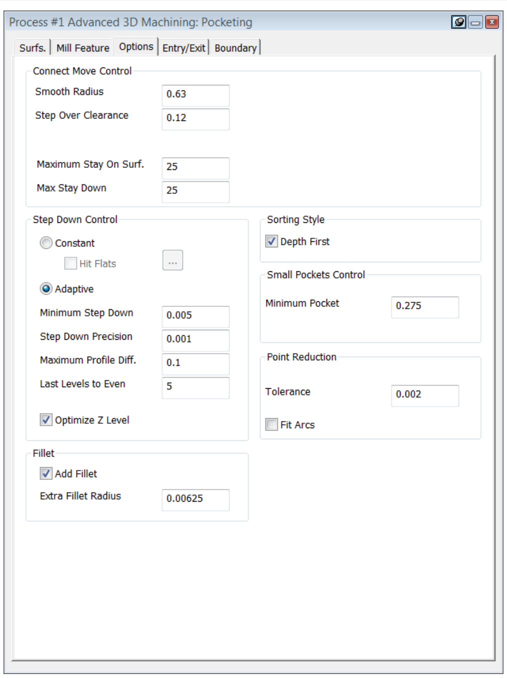
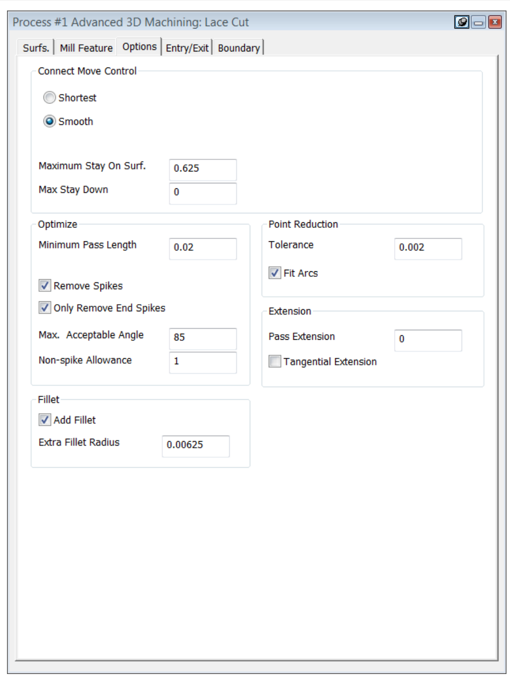


In this process group there are five processes. The first one is the base process. The second and third processes are of type Toolpath Splitter, which split toolpath generated by process one. The fourth is a Tool Holder Gouge Checker which is attached to the first one (base process). The fifth is another Toolpath Splitter. This multi-process group works in the following manner:

1. Toolpath is generated for process 1. Gouge checking is applied against the tool holder in process 1, and gouge-free toolpath is created in operation 1.
2. The toolpath that gouges from process 1 is put into process 4.
3. Process 2 and 3 will split toolpath from process 1 into three parts as a result of the Tool Splitter processes, creating operation 2 and operation 3.
4. Likewise, process 5 will split the gouge-checked toolpath process 4 into two parts, creating operation 4 and operation 5.

Options Tab

This tab provides options for the toolpath type you are generating. For the most part, they are generic, with each toolpath cut type having slight differences in the options. To minimize duplication of information, the options for all toolpath cut types are described here.

Options Tab with Stepdown, Sorting, and Small Pockets Control	Options Tab with Optimize and Extension
	

Connect Move Control

The items in this section define moves between steps and passes.

Smooth Radius

The ends of toolpath have loops on and off the part. This is the radius of the loop. A value of `0` makes a straight line. You can use this item with the following toolpath types: "Pocketing" on page 129 and "Flats Cut" on page 142.

Step Over Clearance:

Whenever the tool steps to the next cut depth, it clears the edge of the boundary by this value. You can use this item with the following toolpath types: "Pocketing" on page 129 and "Flats Cut" on page 142.

Shortest/Smooth

These items control the move between passes. Selecting Shortest makes a direct connection, that is, a straight line, to the next pass. If Smooth is selected a smooth, curving connection is made. There are no sharp corners with the Smooth option so the cutter speed is not changed by much. You can use this item with the following toolpath types: "Lace Cut" on page 133, "N Curve Flow" on page 136, "Curve Projection" on page 138 and "Intersections" on page 143.

Shortest/Angled

These items control the move between passes. Selecting Shortest makes a direct connection, that is, a straight line, to the next pass. If you select Angled, the tool ramps down to the next pass at the specified angle. You can use this item with the following toolpath types: "Contour" on page 139, "Constant Stepover Cut" on page 140, "Intersections - Rest" on page 145, and "Steep Shallow Cut" on page 151.

(Angled) Smooth Ramp

If you select this item, the angled move between passes is a smooth, S-shaped curve.

(Angled) Trim to Ramp Advance

If you select this item, the link moves between slices angled are spiraled. This item is designed for waterline-type cuts.

Maximum Stay on Surface

When moving between passes and steps, the tool either stays near the part or retracts to the clearance plane if it needs to move more than this value. This item is used by all of the "Toolpath Cut Types" on page 121.

Steep / Shallow

Defines separate values for Steep and Shallow sections of the model when you use the following toolpath type: "Steep Shallow Cut" on page 151.

Maximum Stay Down

Maximum Stay Down lets you override normal retract moves. Retracts that would otherwise occur are suppressed when the tool travel distance is less than the value you specify.

Step Down Control

Step Down Control is available with the following toolpath types: "Pocketing" on page 129, "Flats Cut" on page 142, "Contour" on page 139, "Intersections - Rest" on page 145, and "Steep Shallow Cut" on page 151. Each step down in Z can either be Constant, which is the value specified by the Z

Step on the Surfaces tab, or Adaptive.

Hit Flats

For some toolpath types, Constant offers the Hit Flats option. Clicking the ellipsis button (...) opens the **Flats** dialog box. This lets you use `Alt+click` to interrogate flat surfaces in the part, such as a boss top or pocket floor. Additional passes will occur at the Z levels of the interrogated flat surfaces.

Note: In releases before GibbsCAM 2013 v10.5, flat surfaces were added using `click` and `Ctrl+click`. The change to `Alt+click` at v10.5 and later makes the behavior consistent with other actions to interrogate parts.

Adaptive

An Adaptive step is a variable step distance that adds extra steps to the toolpath. Selecting the Adaptive step down activates several items to help define the steps.

Optimize Z Level

The Optimize Z Level option is available for all stepdown controls. It causes small adjustments to slices' Z levels to avoid surfaces that are almost horizontal. (Without adjustment, not-quite-horizontal surfaces can cause severe fragmentation of the slice.)

Minimum Step Down

This is the smallest distance allowable between steps.

Step Down Precision

This value is a tolerance for how accurate the system is when determining the appropriate place to insert a new step.

Maximum Profile Difference

This is the maximum allowable XY distance change between adjacent Z steps.

Last Levels to Even

This item lets the system average the step distance of the last several Z steps. Averaging the steps prevents the last step from being a very shallow cut. This value is the number of final Z steps over which the averaging is distributed.

Optimize

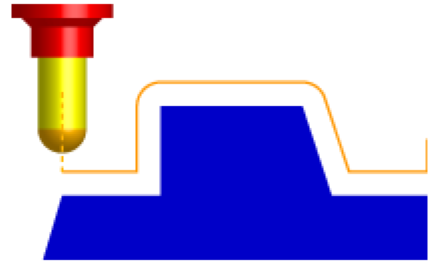
The items here help you to refine the toolpath. You can use this item with the following toolpath types: “Lace Cut” on page 133, “N Curve Flow” on page 136, “Curve Projection” on page 138 and “Intersections” on page 143.

Minimum Pass Length

The Minimum Pass Length value prevents any passes that are shorter than this length from being generated, preventing a lot of small, short passes.

Remove Spikes

Sometimes at the end of a pass, where one surface is adjacent to another at a very steep angle, there is a sharp jump. This can happen where the cutter touches a steep wall and is lifted to the top, or where it "falls off" a high ledge and drops to the bottom. Activating the Remove Spikes item can remove these sharp jumps.



Only Remove End Spikes

Only spikes found at the ends of the passes are removed.

Maximum Acceptable Angle

Spikes or jumps with an angle greater than this are removed from the toolpath. The angle is measured from the horizontal plane.

Non-Spike Allowance

You can trim off any small horizontal areas left at the top or bottom of the spike. The value here is the maximum length of area that is edited from your toolpath.

Sorting Style

The items in this section help you to control the behavior of the toolpath passes.

Depth First

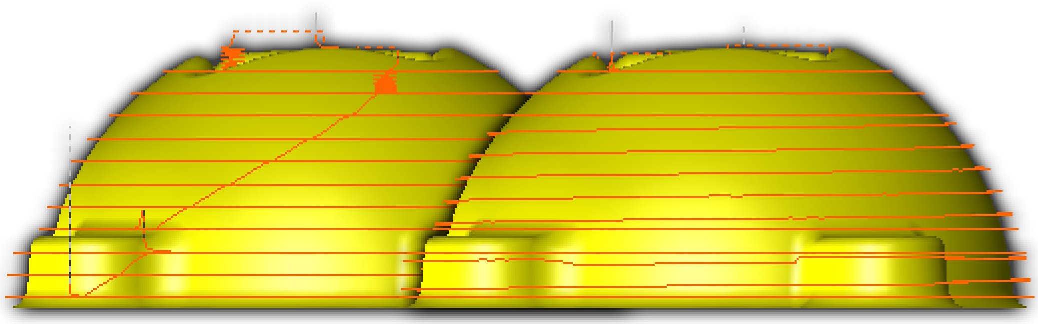
Machines each area (such as a pocket) completely before moving to other areas of the model.

Helical

Sets toolpath passes as helical rather than waterline for the following toolpath types: "Contour" on page 139 and "Steep Shallow Cut" on page 151.

Because this uses projection type of strategy, if Helical is used on vertical or near-vertical surface shapes, jagged toolpath might be generated.

The following image shows the same process being applied to a model. The left side shows a standard contour operation; the right side shows a helical contour.

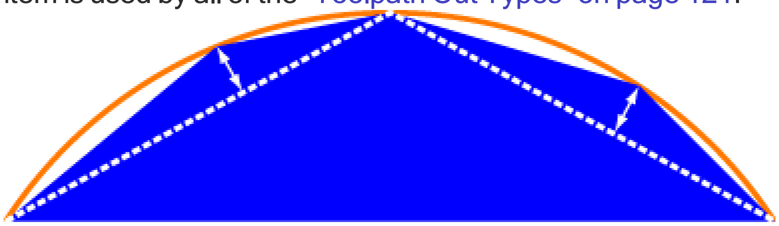


Small Pockets Control

Any cavity that is smaller than the Minimum Pocket values (a diameter) will not be included in the toolpath. This is avoids difficulties with ramping a tool into a small area. The default value of `0` means that this function is disabled and the toolpath attempts to machine the cavity. For toolpath types that use this item, see "Pocketing" on page 129, "Lace Cut" on page 133, "Contour" on page 139, and "Steep Shallow Cut" on page 151.

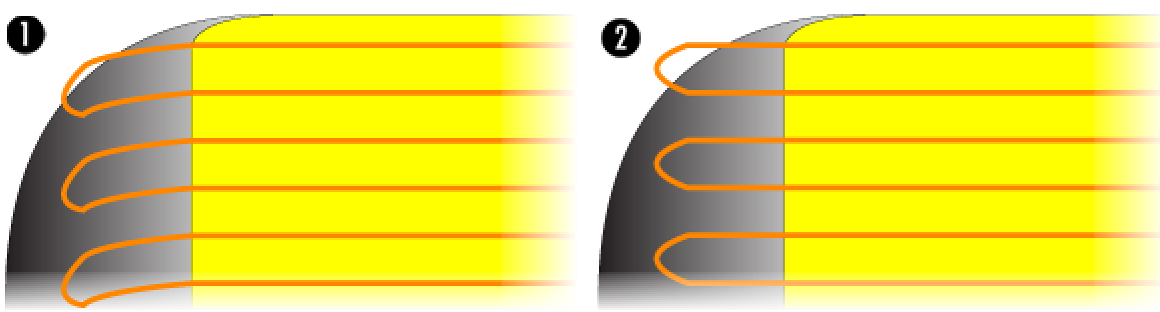
Point Reduction

This item attempts to simplify the topology of the part. The Tolerance setting is used to measure chord heights between points on the model and if the chord height is less then the Tolerance then that point is simplified. If the Fit Arcs item is enabled, then the system attempts to create an arc across the points to create arc-optimized toolpath. This is illustrated in the following image. This item is used by all of the "Toolpath Cut Types" on page 121.



Extension

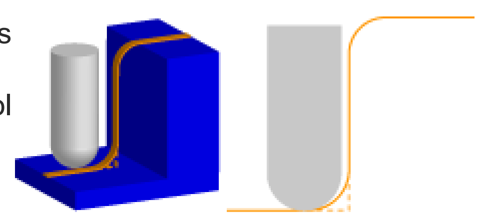
With this item you can extend the toolpath beyond the edge of the boundary. This allows the tool to move into the part at the machining feedrate rather than the rapid feedrate. Enter the distance by which you wish to extend the pass in the Pass Extension text box. By selecting the Tangential Extension item, the passes are extended tangentially from the generated toolpath. When disabled the passes are extended along the faces.



Example of toolpath where Tangential Extension is disabled (1) and enabled (2).

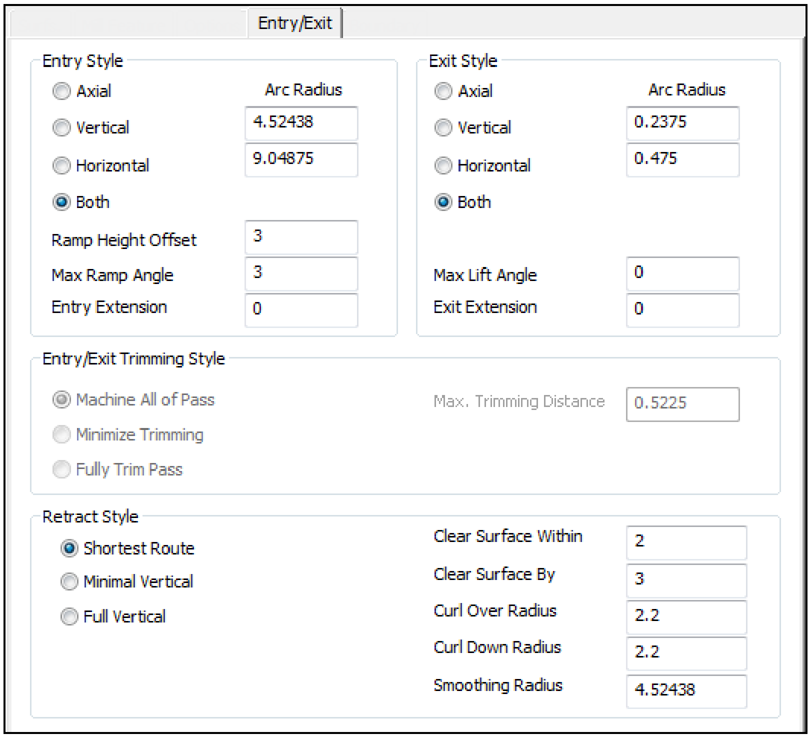
Fillet

Selecting the Add Fillet checkbox allows you to specify a radius value for sharp (concave) corners to maintain high machining speeds. The Extra Fillet Radius value defaults to 2.5% of the tool diameter. This value and the tool radius define the fillet added to the toolpath.



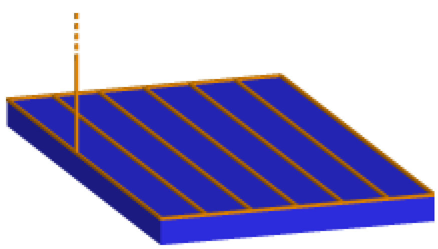
Entry/Exit Tab

The items found on this tab control the toolpath as it enters or leaves the surface to be cut.

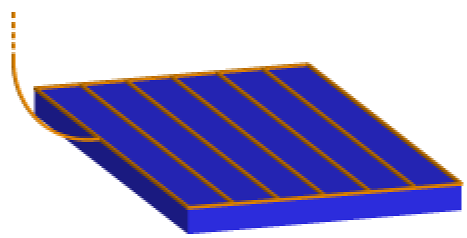


Entry Style

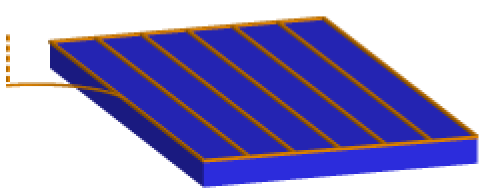
The items in this section help you determine how the tool will enter the part.



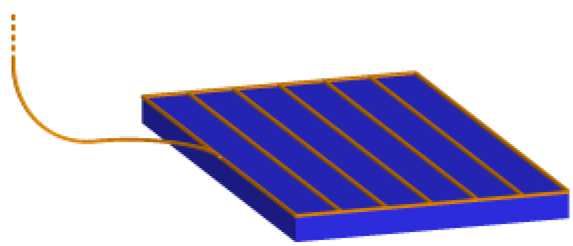
Axial Entry



Vertical Arc Entry



Horizontal Arc Entry



Both Vertical and Horizontal Arcs Entry

Axial

The tool will approach the part in a straight line from along the tool axis

Vertical
The tool will approach the part using a vertical radius. By default the Arc Radius is 25% of the tool radius.

Horizontal
The tool will approach the part using a horizontal radius. By default the Arc Radius is 50% of the tool radius.

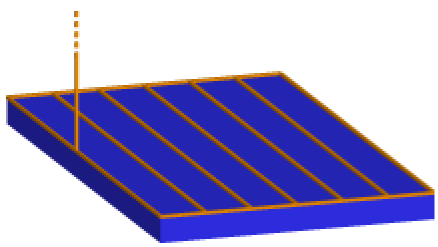
Both
Using this choice the tool will approach the part with a vertical radius followed by a horizontal radius.

Max Ramp Angle
Processes that have ramping moves will cut into the material at no steeper an angle than this value, measured from horizontal.

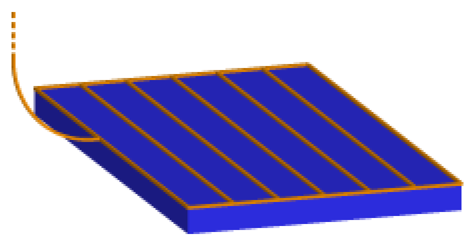
Entry Extension
This item is used with ramps. It is an extra value added to the top profile. This ensures that the tool has fully slowed from its rapid speed. This lets the tool enter the material smoothly at the ramping angle.

Exit Style

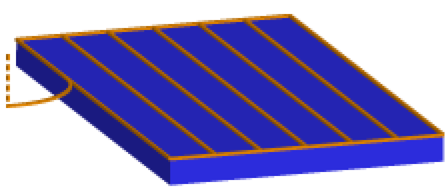
The items in this section help you determine how the tool will leave the part.



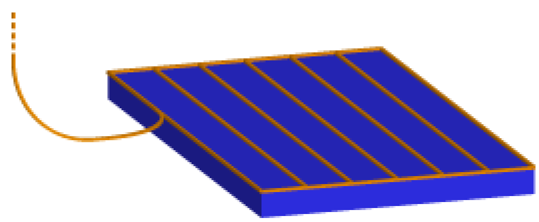
Axial Exit



Vertical Arc Exit



Horizontal Arc Exit



Both Vertical and Horizontal Arcs Exit

Axial
The tool will leave the part in a straight line from along the tool axis

Vertical
The tool will leave the part using a vertical radius. By default the Arc Radius is 25% of the tool radius.

Horizontal
The tool will leave the part using a horizontal radius. By default the Arc Radius is 50% of the tool radius.

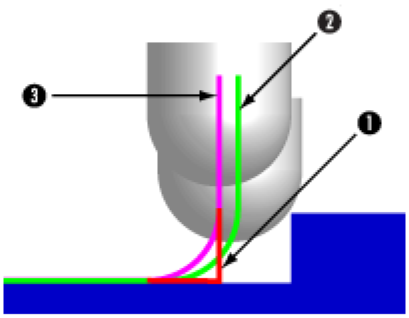
Both
Using this choice the tool will leave the part with a vertical radius followed by a horizontal radius.

Max Lift Angle
Processes that have ramping moves will leave the material at no steeper an angle than this value, measured from horizontal.

Exit Extension
This item is used with ramps. It is an extra value added to the top profile. This lets the tool gradually reach its rapid speed.

Entry/Exit Trimming Style

This option allows you to choose how the approach/retract arc should be applied to the toolpath.



1. Machine All of Pass
2. Minimize Trimming
3. Fully Trim Pass

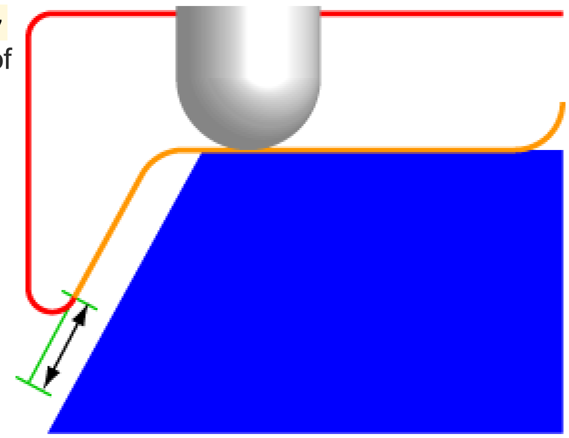
Machine All of Pass
The toolpath will follow the model, including vertical surfaces and corners. An arc will be inserted at the end of the operation, only if enough space is available to avoid any collisions.

Minimize Trimming
The retract movement will remain close to the model, keeping a minimum distance, in order to apply the arc to the radius setting.

Fully Trim Pass
In critical cases, this is the safest option for avoiding collisions. The toolpath is trimmed so the entire arc will fit and the tool will not be any closer then the Maximum Trimming Distance.

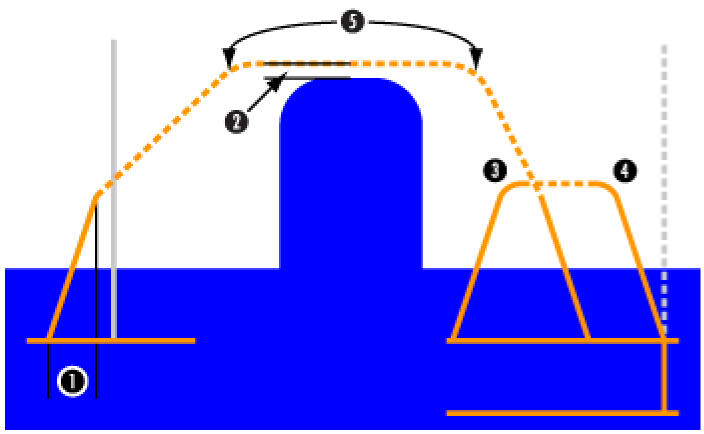
Maximum Trimming Distance

This option affects both the Minimize Trimming and Fully Trim Pass functions. It represents the maximum value of trimming. If the amount trimmed exceeds this value then an arc is not used. Instead the whole pass is machined and a vertical motion is added at the end.



Retract Style

These items control how the tool moves between passes and sections to be machined.



1. Clear Surface Within
2. Clear Surface By
3. Curl Over Radius
4. Curl Down Radius
5. Smoothing Radius

Shortest Route:

With this retract style the cutter clears the surface and takes a direct route from one pass to the next using curves to speed the process. This option creates moves that require machines that can perform interpolated rapid moves.

Clear Surface Within:

This is the total horizontal distance the cutter moves away from the surface before rapiding. This creates an angular retract or entry feed.

Clear Surface By:

This is the distance by which the cutter clears the surface by when performing rapid moves.

Curl Over Radius:

This controls the arcing of the toolpath after the exit arc and before the rapid to the next position.

Curl Down Radius:

This controls the arcing of the tool path after the rapid and before the next entry arc.

Smoothing Radius:

When moving between areas to be machined, the system may make polyline rapid moves to clear a surface. The Smoothing Radius is an arc that is added between the straight rapids of the polyline, making a smooth transition. This creates more efficient toolpath.

Minimal Vertical:

Cutter moves vertically to the minimum Z height needed to be clear of the surface. The cutter then moves along this plane to the start of the next pass and then lowers along a vertical line to the start position.

Clear Surface By:

This is the distance by which the cutter clears the surface by when performing rapid moves.

Full Vertical:

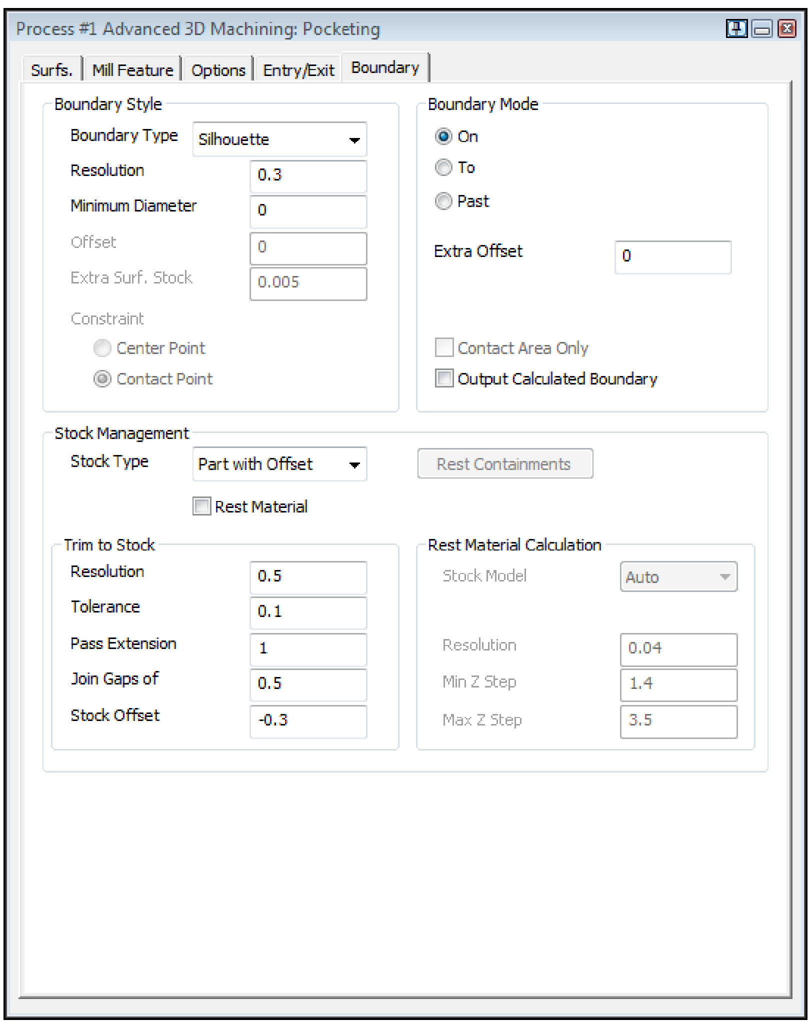
Cutter moves vertically to the clearance plane. The cutter then moves along this plane to the start of the next pass and then lowers along a vertical line to the start position.

Clear Surface By:

This is the distance by which the cutter clears the surface by when performing rapid moves.

Boundary Tab

Boundaries allows you to restrict the toolpath computation to the geometrical size of the model or to use pre-defined geometry as a boundary limitation. Advanced 3D Machining processes are boundary based, meaning that each and every operation needs a boundary to be defined, even if it is just the default stock definition. It is important to note that the boundary shape can also effect the final result of the toolpath. The items on this tab let you define what the boundary is and how the toolpath should interact with it.



Boundary Style

Here you define the type of boundary to be used. In some cases, what you have selected in the workspace will control your available options.

Boundary Type

There are 5 boundary types: Bounding Box (default), Silhouette, Shallow Areas, Cutter Contact Areas, and Selected Curves. If faces are selected, boundary type is limited to Cutter Contact Area or Silhouette. If a closed shape is selected, boundary type is limited to Selected Curves.

- Bounding Box uses a 2D rectangular shape to define the stock size. This shape will encompass the faces selected for machining.
- Silhouette creates a 2D shape using the edges of the model or selected faces as seen from the tool axis (i.e., normal to the Machining CS).
- Shallow Areas will only look at the relatively flat areas of the model, as defined by the values specified on the **Surfs.** tab (under Normal-Vector Range) for Minimum Angle and Maximum Angle. For example, if these values are `0` and `45`, then the system will generate toolpath for shallow areas between 0 and 45 degrees.
- Cutter Contact Areas will only look at areas where the cutter is in contact with the selected model. This item is intended for faces at an angle less than 80 degrees.

Constraint: Center Point / Contact Point

Most machining operations and the creation of 3D boundaries constrains or limits the tool center to the edge of the boundary or surface; the tool can move no further than this. When creating a boundary with Cutter Contact Areas, you can constrain or limit the tool contact point to the boundary instead. In effect, the machinable area is offset by the radius of the cutter. When using Center Point, the tool center point is always within the boundary. When using Contact Point, an edge of the cutter is always within the boundary.

- Selected Curves will machine within one or more closed geometric shapes. For this item to be available the geometry must be selected.

Resolution

This is the tolerance of the boundary calculation. A smaller value provides a more accurate and detailed boundary, but takes longer to calculate.

Minimum Diameter

This is a setting that can create a simpler boundary shape. Boundaries with a distance between two points that is smaller than this value will not be included in the boundary calculation.

Offset

After the boundary is calculated they can be offset by this amount. Adding a small offset value can be helpful in eliminating boundaries with jagged edges. This value can be as large as the tool radius.

Extra Surface Stock

This values is added to cutter calculation in order to keep the tool away from constraint faces.

Boundary Mode

These settings control the position of the tool when it is in contact with the boundary.

- On specifies that the center of the tool will cut to the boundary.
- To specifies that the edge of the tool will cut to the boundary. Specifically, it will remain inside and tangent to the boundary.
- Past lets the tool travel past the boundary but remain tangent.

Extra Offset
You may add an amount the tool can travel by entering a positive or negative value in this field.

Contact Area Only
Tool path will be generated only where the tool touches the part.

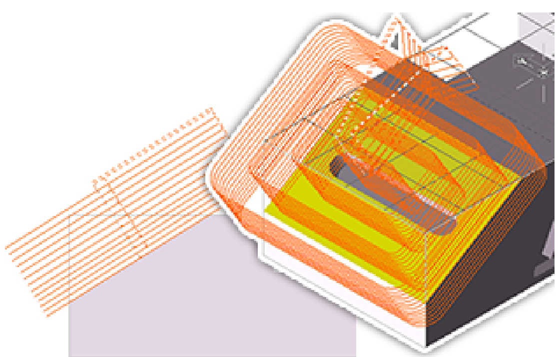
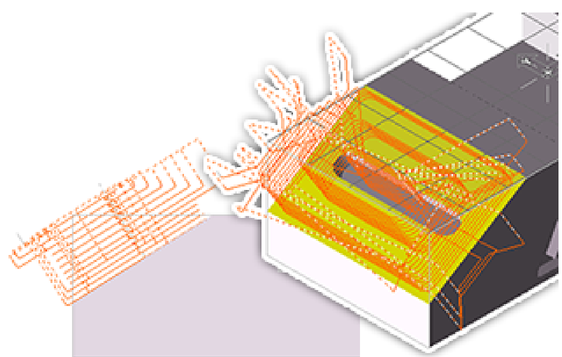
Output Calculated Boundary
This will output the geometry of the Boundary in the current workgroup.

Stock Management

These settings control the interaction of toolpath and stock.

Stock Type:
This is how the toolpath is trimmed after being generated. There are four types of stock to choose from:

- None does not trim the toolpath against any further definition.
- Workpiece Default is the default stock boundary. This is typically the stock definition in the Document Control dialog. In some cases, such as using the Cutter Contact AreaBoundary Type, the bounding box is a “shrink-wrapped” model: that is, a box that can wholly contain the solid.

None	Workpiece Default

- Part with offset is the selected model plus some offset amount.
- Solid uses the selected model.

When the Rest Material checkbox is selected, the system will analyze the remaining material condition of the stock to determine where cuts should be made, using values you specify under Rest Containments, Trim to Stock, and Rest Material Calculation.

Trim To Stock:

Available when the Rest Material checkbox is selected, using the Part with Offset stock condition. These are extra controls for how the toolpath interacts with the stock.

Resolution:

This is the distance between points to calculate toolpath. At each point, the system checks to see where the tool is in relation to the stock. The larger the value, the more quickly stock is calculated, but the stock condition is not as accurate.

Tolerance:

This is the accuracy of the toolpath to the stock condition. The tool may be above or below the stock condition by this amount.

Pass Extension:

The toolpath is extended away from the material into fresh air by this amount. The trimmed passes are lengthened in each direction by this value. The pass extension parameter can be used to allow the tool to move into the part at machining feedrate rather than rapid movements.

Join Gaps of:

When toolpath is linked, it can become fragmented with many small gaps. Any gaps between toolpath sections that are smaller than the specified value are joined together, thereby reducing fragmentation.

Stock Offset:

This value determines the amount of material to be removed or retained on the surface of the stock. This additional stock offset can be a positive or negative value.

Rest Containments

Available when the Rest Material checkbox is selected. Clicking this button lets you choose containment curves to further define the rest material condition.

Rest Material:

Available when the Rest Material checkbox is selected. Enter the tolerance as well as the minimum and maximum distances you want to be calculated between the Z-slices. The greater the distance between the Z-slices, the less accurate the representation is of the model after passes have been made, but the calculations are faster.

Resolution:

This is the granularity of the calculation: the smaller the value, the finer the detail, but the calculation is slower. Using a larger resolution will decrease detection time, but may lead to very small features being missed.

The system searches along the cutter paths, examining appropriate points along the cutter pass and recording whether that position is above or below the surfaces. The current and previous positions are compared and, if they are different (that is, one above and one below), then the tolerance is used to locate the precise position of the change between above and below. This information is then used to trim the cutter pass. The system checks points along a pass where the direction changes, but long and straight passes are supplemented by extra points. The resolution is used to determine the distance between those points.

Min. Z Step

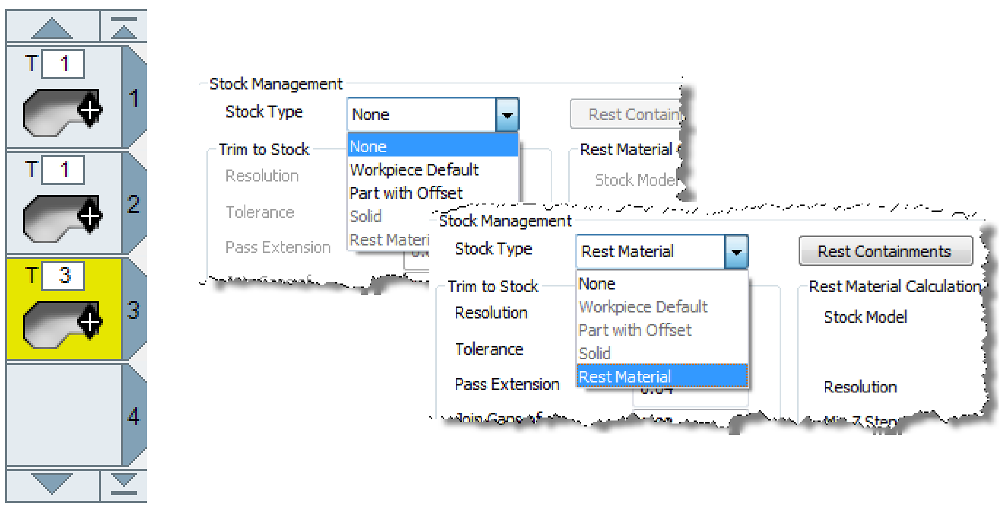
This is the minimum distance allowed between Z slices.

Max Z Step

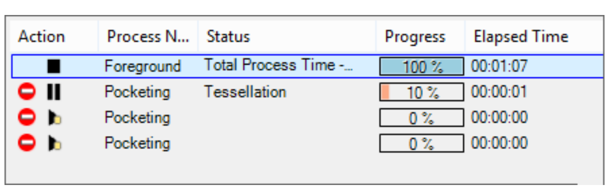
This is the maximum distance allowed between Z slices.

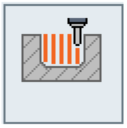
Material Only process example:

There are three processes in the process list. The first one is the base process, in this case, a Pocketing process. The following two are dependent ones which are attached to the base one. The Rest Material option is grayed out in the stock type list box for the base process, since there are no preceding processes in the process list. For the two dependent processes, the stock type is set as Rest Material.



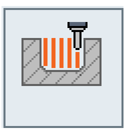
When you click Do It, the process group is added to the Task Manager. The second and third process status is set to “blocked” while the first one is running. The second and third processes are dependent on the first process and cannot be calculated until the first process is completed. When the first process is finished, the second one will start calculating and so on. The second process only removes the material remaining from the first, and the third process only removes the material left over from the second.





Plunge Rough Process

The Plunge Rough function is available with the SolidSurfacer option. The function is created like any other process such as drilling, pocketing or surfacing. The function is defined by combining a tool and a process tile. The Plunge Rough process is applied to solid bodies. Selected bodies and faces may be used in several ways to control the machined area including as constraint surfaces and stock bodies. Geometry may be used as a constraint boundary or as a guiding contour.



What is A Plunge Rough Process?

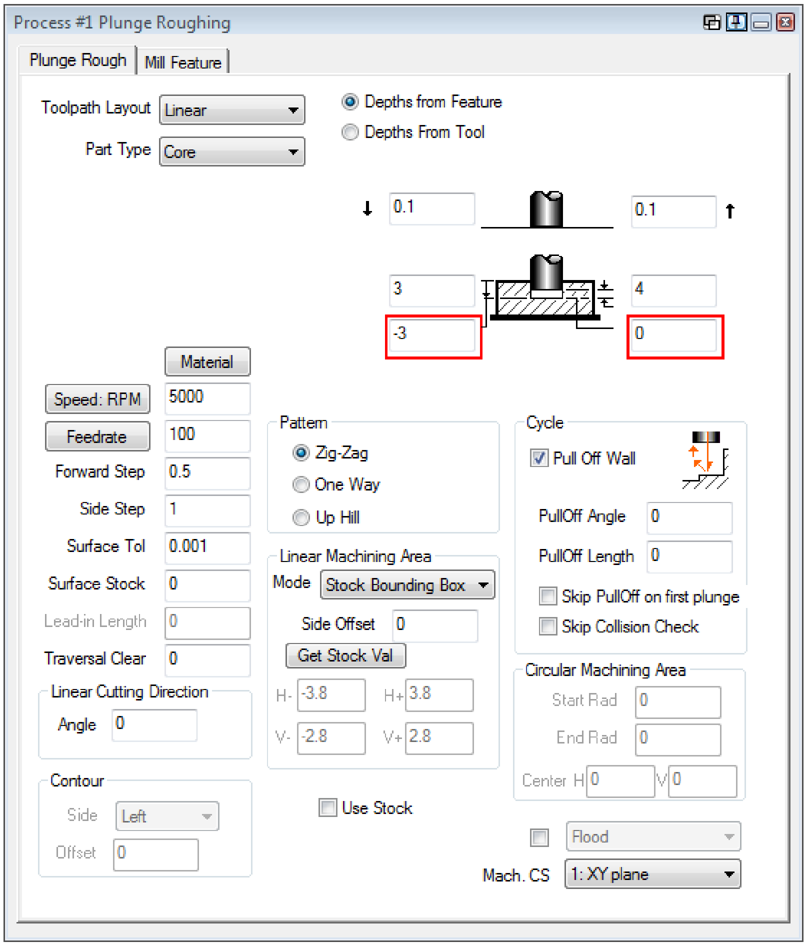
A Plunge Rough process removes large amounts of material along the Z axis with numerous close-together plunge and retract moves, effectively roughing with a drilling operation. This allows you to maintain maximum feeds and speeds. Plunge Roughing is an ideal solution for clearing out material on molds.

The Plunge Rough process can create linear or circular patterns, patterns between guide curves or toolpath that follows a contour.

You may define this process to be used with either a face mill (plunge endmill) while using less than 50% of the tool's radius or you may use a drill, bullnose, ball endmill or flat endmill using as much of the tool as desired.

User Interface

The process dialog for Plunge Rough is similar to other mill process dialogs in its controls for Material, Speed RPM, Feedrate, and clearances (depths diagram), as well as its **Mill Feature** page.



Toolpath Layout:

You can set the toolpath basic motion from this pull-down menu. The choices include Linear, Circular, Guide Curves, and Contour.

Part Type:

When Toolpath Layout is set to Linear, you specify whether the **Part Type** is a Core or a Cavity. The toolpath provides different results depending upon the choice.

	Geometry Selection	Drill Pattern
Core	You can select multiple contours to specify the machining area. If no geometry is selected, then the part is machined to the stock boundary.	If the value for Angle (under Linear Cutting Direction) is `0`, then the drill pattern will follow a grid from left to right and from bottom to top. If Zig-Zag (under Pattern) is selected, then alternating rows will be inverted.
Cavity	You must select exactly one start point and exactly one contour. The location of the point is used as the start location. You must ensure that material at the start location has	The machining begins at the start point and move in the positive direction. When reaching the end, the machining will restart at the start point and move in the negative direction to finish the current cut. The process will then

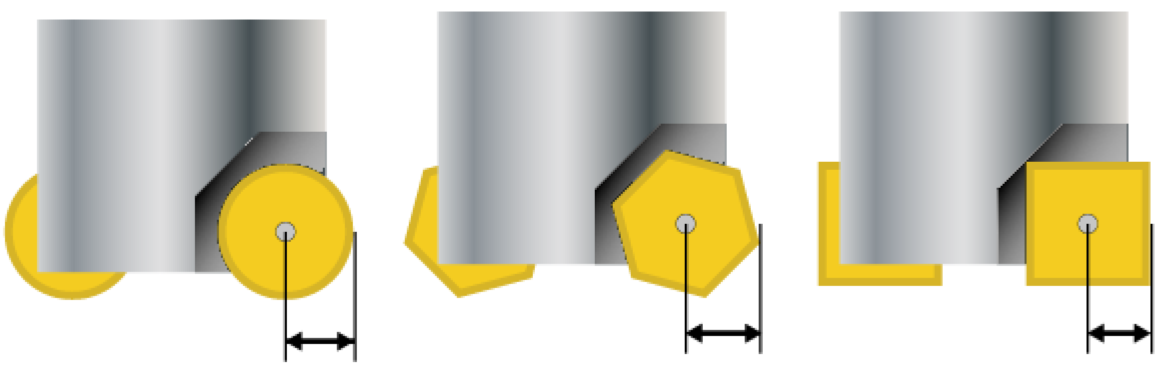
Geometry Selection	Drill Pattern
already been removed. The best way to ensure this is to create a drilling operation using the start point.	finish all the cuts to the right, before finishing all the cuts to the left of the start point.

Clearances:
As with all process dialogs, you set the entry clearance, exit clearance, surface Z value, and final cut depth. If Depths from Feature is selected, you can also provide feature-specific settingss like Feature Top Surface Z and Feature Depth Z. Also, the **Mill Feature** page lets you use values that are based on attributes and/or calculated automatically.

Please note that this is *not* a gouge-protected process. For example, if the value for surface Z is lower than the highest portion of a selected solid, the tool will gouge the part.

Forward Step:
Distance between successive cuts.

Side Step:
Stepover between rows of cuts. When you use a tool that has a non-cutting surface (such as an indexable tool), you must set the proper stepover based on the tool's cutting radius, as illustrated below.



Several examples of the cutting radius of an indexable tool.

Please note that if the Side Step is larger than the Forward Step, the system will add plunges between the side steps at a distance equal to the Forward Step.

Surface Tolerance:
Maximum allowable deviation of the toolpath from the surface of the solid.

Surface Stock:
Amount of stock left by the operation.

Lead-In Length:
Available only when Toolpath Layout is set to Circular. Distance off the part at which the operation will begin. Generally, this is less than the value for Forward Step.

Traversal Clear:
Set this to a value high enough for the tool to clear all fixtures and material as it moves from one plunge to another.

Linear Cutting Direction:
The Angle parameter is available only when Toolpath Layout is set to Linear or Circular. This parameter provides the ability to define cutting direction and angle, as follows.

Linear Toolpath Layout	Circular Toolpath Layout
The linear cutting direction is the direction that the tool moves when taking forward steps. The value for Angle corresponds to the angle between the cutting direction and the H axis of the active CS.	When Toolpath Layout is set to Circular, however, Linear Cutting Direction must be set to either `0` and `180`: • When set to `0`, the tool moves in a counter-clockwise direction at first. • When set to `180`, the tool moves in a clockwise direction at first. • If Pattern is set to One Way or to Up Hill, then all cuts are the same direction. • If Pattern is set to Zig-Zag, however, this setting applies only to the *first* cut. Subsequent cuts alternate between clockwise and counter-clockwise.

Pattern:
This choice specifies the behavior of the toolpath.

- Zig-Zag toolpath cuts back and forth.
- One Way cuts in a single direction according to the selection made for Cutting Direction.
- Up Hill cuts back and forth while always climbing.

Linear Machining Area:
Offers settings that are available only when Toolpath Layout is set to Linear.

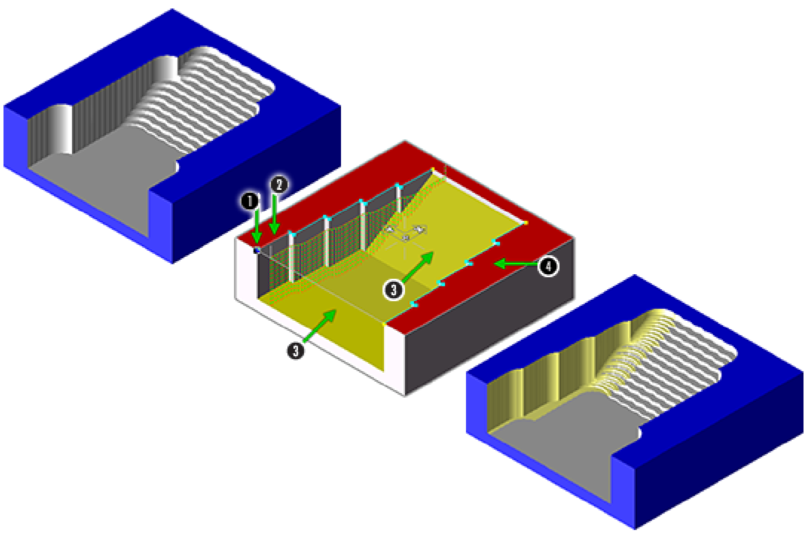
Mode:
The option chosen in this pull-down menu sets the area to be machined and governs what other controls are available.

- Stock Bounding Box specifies that the machined area is bounded by the stock, offset by the value specified for Side Offset. To load the values for H-, H+, V-, and V+, click the Get Stock Value button.
- Part Bounding Box specifies that the machined area is bounded by the stock, offset by the value specified for Side Offset. To load the values for H-, H+, V-, and V+, click the Get Stock Value button.
- User Defined causes the Get Stock Value button to become available and allows you to specify specific values for horizontal and vertical offsets.

Get Stock Value:
Clicking this button (not available when Mode is set to User Defined) loads the values from the Document Control dialog into the fields for H-, H+, V-, and V+.

Circular Machining Area:
Offers settings that are available only when Toolpath Layout is set to Circular. The parameters in this section control which way the tool will cut. By setting values for Start Radius and End Radius, you define the size of the area to be machined and whether to machine outside-to-inside or from inside outside.

Contour:
Offers settings that are available only when Toolpath Layout is set to Contour. Steps for some Toolpath Layout cut types can leave large amounts of material on the walls. The usual reason to set Toolpath Layout to Contour is to rough out additional material along vertical walls that are parallel to the cutting direction of a previous Linear cut. To create this process, you must select a contour and the floor faces that you wish to machine The wall faces need not be selected.



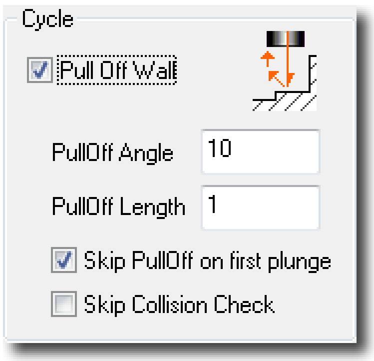
1. Selected Start Point
2. Selected First Feature
3. Selected Faces to Machine
4. Constraint Face

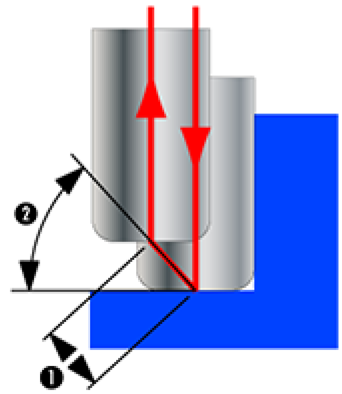
An example of using a Contour Toolpath Layout to clear excess material.

Side:
You must specify which side of the contour — Left, Right, or Center — will be cut using this selection. The direction of the cut can be controlled by selecting the starting point and the first line or arc in the direction you want to cut.

Offset:
Available only if Side is set to Left or Right. You can specify an additional offset that will be added to the tool radius to offset the selected contour. This is commonly used to set the amount of stock you wish to keep on the walls.

Cycle:
The parameters in this section are available only if you select the Pull Off Wall checkbox. They provide you with the ability to retract the tool with a pulloff motion, so as to minimize tool wear and to avoid scraping the walls. When Pull Off Wall is in effect, you specify an additional move before the tool retracts at each drill position. This additional move is defined by specifying the PullOff Angle measured in a vertical plane (0 = horizontal) and a PullOff Length, as illustrated below. The intent of a PullOff move is to avoid retracting the tool while it is in contact with the material.





1. Pull Off Length
2. Pull Off Angle

Skip PullOff on first plunge:

This option allows you to avoid the collision that would occur if the tool were to pull off from an initial hole start position is inside the part and a drill hole smaller than or equal to the Plunge Rough diameter has been pre-drilled.

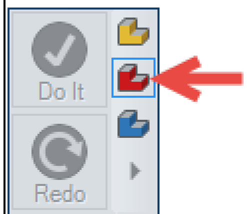
Skip Collision Check:

By default, this checkbox is *not* selected, and therefore the system *will* perform collision checking on the pulloff. However, if you are sure that collision checking is unnecessary and you want to reduce toolpath calculation time, you can select this checkbox. ***WARNING!*** You should only use this option if the forward step is smaller than the tool radius and the pulloff length is smaller than the tool radius. If you want to skip collision checking, you should rough the part with using the Up Hill choice, or in a direction that will prevent gouging.

Controlling Where Plunge Rough Machines

The results of a Plunge Rough process will depend upon what is selected as well as the part's stock definition. Interactions with part components, selections and the results.

Workgroup Stock	Plunge Rough processes will recognize stock as defined by geometry in a workgroup. The Plunge Rough toolpath will plunge in rapid to the stock and then feed to the part taking into account any Surface Stock value. The toolpath will also be trimmed to the stock.
Stock Solids	Plunge Rough processes recognize bodies defined as stock. This will optimize the tool motions. The Plunge Rough toolpath will plunge in rapid to the stock and then feed to the part's surface taking into account any Surface Stock value.
Geometry Constraints	You may define a shape to limit the Plunge Roughing to the area within the shape. The geometry must be a closed shape. The roughing will be limited to the inside of the selected shape. The tool axis will stay inside the 2D projection of the selected shape. Only one selected closed face will be recognized as a constraint. Contour Plunge Rough processes will use one open geometry shape while Guide Curve Plunge Rough processes require two open shapes.
Selected and Unselected Faces	In addition to selecting a solid to apply a Plunge Rough process to, you may select individual faces to machine. The system will keep the tool within the bounds of the selected faces. Unselected faces act as constraint faces so the toolpath calculation process will avoid these faces. No roughing will be done of top of these unselected faces.

Constraint Faces	In addition to selecting a solid to Plunge Rough, faces of the solid body may be set as constraint faces. This is done by clicking on the Constraint button on the Machining palette and then choosing faces in Face Selection mode. Constraint faces are shown as red rather than yellow or gray. These faces, and the area above the faces, will be avoided during the toolpath calculation process. While these faces will not be considered in toolpath calculation, the tool may machine over these faces by up to a tool radius, to fully machine selected faces. 
Fixture Bodies	Plunge Rough processes recognize bodies that are set as fixtures. No roughing will be performed on top of these bodies.

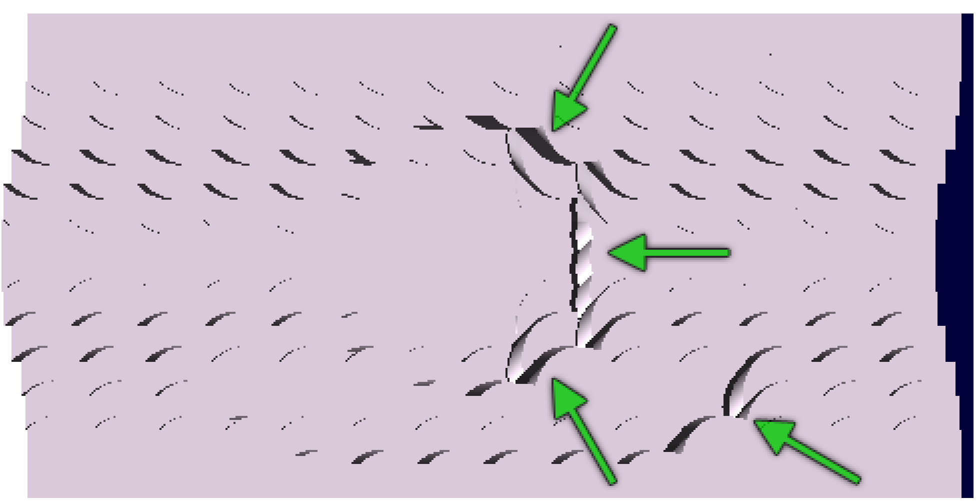
Samples

Following are some examples of common problems and solutions. These include:

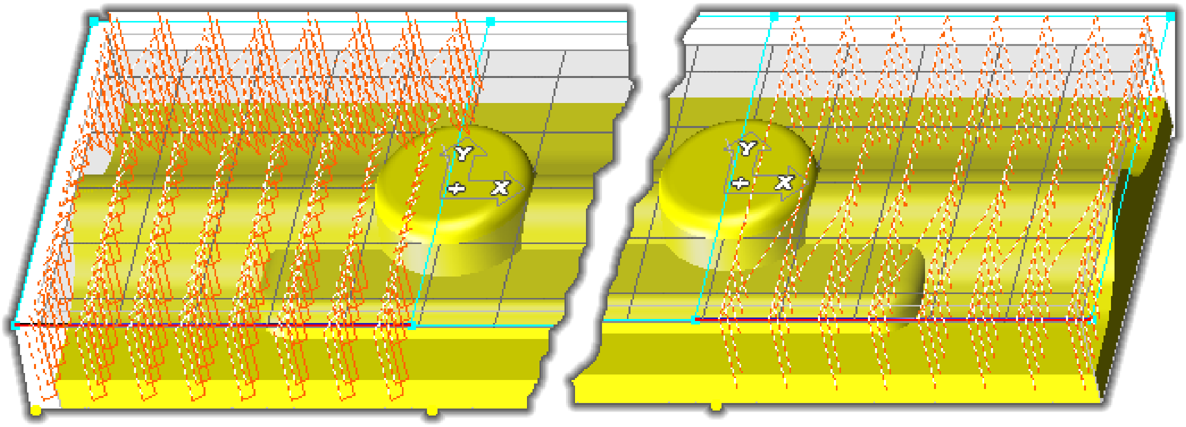
- “Uphill Cutting and Gouge Checking” on page 177
- “PullOffs with Stock” on page 178
- “Contouring” on page 179
- “Guide Curves” on page 180

Uphill Cutting and Gouge Checking

In this part we have an uphill linear toolpath with a pulloff. The process has the Skip Collision Check option enabled. With this part these options will cause the tool to collide with the part during the pull off. There are several solutions to this problem.



- Take advantage of the part's topology and break the toolpath into separate operations machining in opposite directions so the tool is always cutting uphill. This is accomplished using containment geometry. This maintains the fast toolpath generation time which disabling collision checking allows.

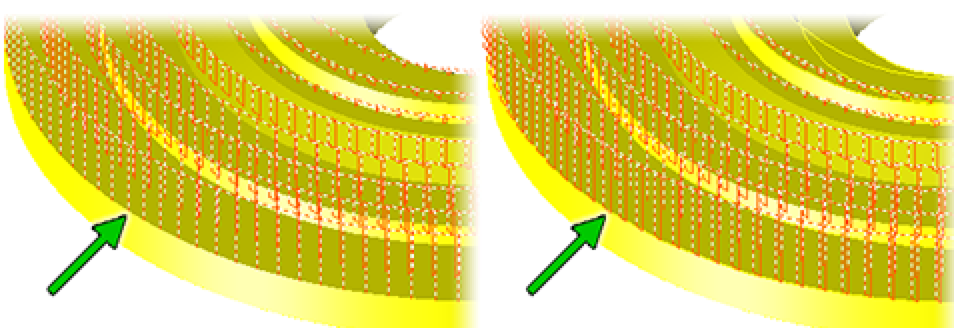


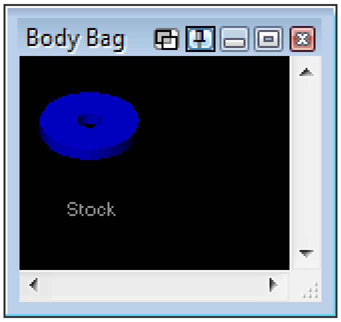
- Disable the Skip Collision Check option. On larger or more complex parts this can mean slower toolpath generation but it is safe. In this case we don't lose any appreciable calculation time and the process trims out pulloffs for some plunges in the areas machined after the central boss.

See `LinearPlungeRough.vnc` in the part files for these samples.

PullOffs with Stock

In this sample we have a circular part in which the first cut does not have any pulloff moves. The software is considering the part's stock all the way to the rectangular definition in the Document Control Dialog. The software is purposely excluding the pulloffs to avoid collisions with the stock. To fix this problem we created a cylinder with an identical OD to the part and set this body's properties to "stock".



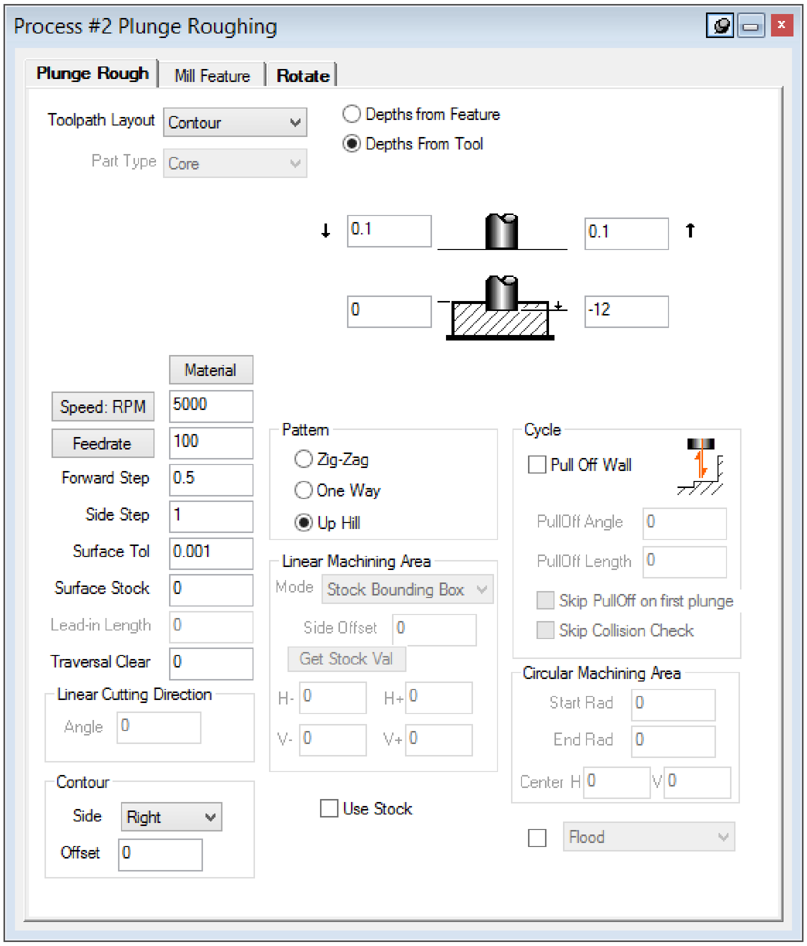


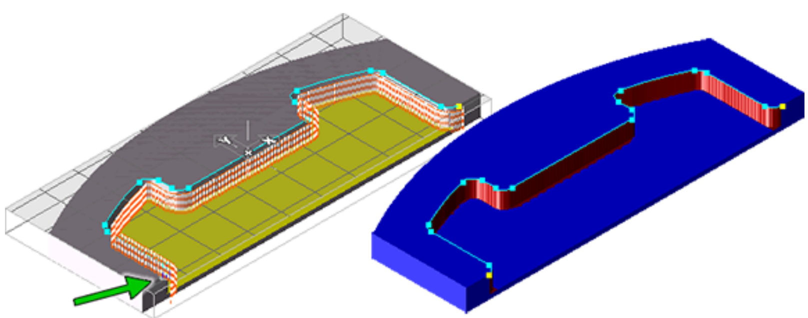
This part also includes a sample of using the circular toolpath definition to control the cutting direction, resulting in the tool always cutting “uphill. The first operation cuts from the outside to the inside while the second operation cuts from the inside to the outside. The two operations meet in the middle at the highest point on the part.

See `CircularPlungeRough.vnc` in the part files for these samples.

Contouring

In this sample part we start with a part that has already had plunge roughing processes applied to it. The results were saved as a facet body and that is used as our stock. This part demonstrates how to use the Contour Plunge Rough process. There is geometry that defines the contour we wish to cut. The point that designates the starting position for the contour (designated by the green arrow in the image below) and the floor face are selected. The process will cut on the right side of the geometry and will be offset by 0.3mm.

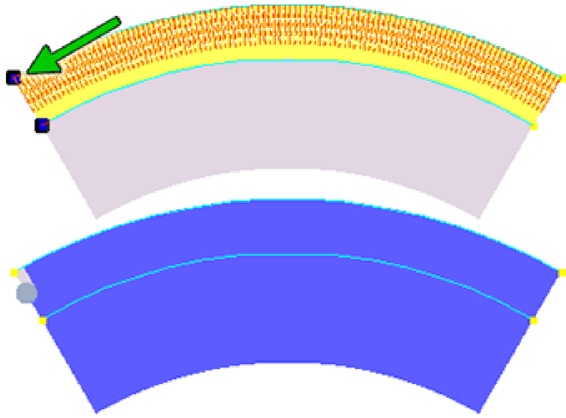




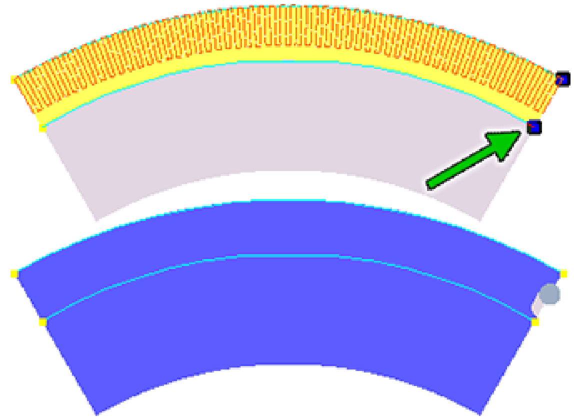
See `ContourPlungeRough.vnc` in the part files for these samples.

Guide Curves

The Guide Curves functionality requires the selection of 2 curves designated with the starting points, and a selected solid or faces to machine. The direction of cut is set by the selected points. In the following examples we have a One Way cut designated as starting at the top left point and a ZigZag cut designated to start at the bottom right point. The One Way cut will continue to cut in a downward direction while ZigZag will alternate.

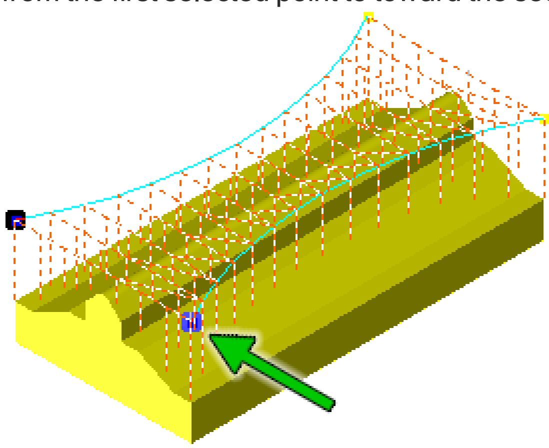


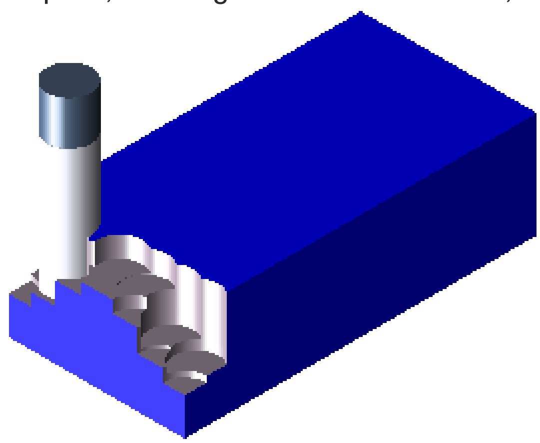
One Way Plunge Rough



Zig Zag Plunge Rough

A Guide Curves Plunge Rough operation using an Up Hill pattern behaves similarly in that it will go from the first selected point to toward the second point, breaking the motion as needed, to go up hill.





See `ContourPlungeRough.vnc` and `ContourPlungeRough.vnc` in the part files for these samples.

Glossary

The following is a list of terms and concepts used throughout the 5-Axis documentation.

Alignment Points	Alignment points are also referred to as sync points. The selection of alignment points is used with the lofting, sweeping modeling functions and 2 Curve Flow machining. Alignment points indicate how the system will blend the selected shapes into a solid or sheet.
Analytics	This term is used to describe surfaces that are defined by a precise mathematical equation. Some examples of analytic surfaces include spheres and cylinders. Analytic surfaces are less mathematically complex than parametric surfaces and are therefore easier for the system to handle. Because analytic surfaces are completely defined by simple equations, it is much easier and faster to perform modeling functions such as rounding and boolean operations on analytic bodies. Often times when bodies or surfaces are imported into the system, they are converted from analytic surfaces to parametric surfaces. The Solids > Tools > Simplify option will attempt to convert any parametric surfaces back into analytic surfaces within a tolerance amount.
Atomic Bodies	Atomic bodies are also referred to as primitive bodies. Atomic bodies are bodies that were created using the standard modeling functions available in the Create Body palette. Atomic bodies are not created from the combination of other bodies using boolean operations. Examples of atomic bodies include spheres, cuboids, revolved bodies and extrusions.
Body Bag	The Body Bag is used as a storage place for solids and sheets in order to keep the Workspace as uncluttered as possible. Double-clicking on a body or sheet in the Workspace will place that entity in the Body Bag. The Body Bag is accessed by clicking the Body Bag button in the main palette. Solids and sheets are represented as icons when they are in the Body Bag. Similar to standard desktop icons, they can be dragged within the Body Bag and also dragged back into the Workspace. Solids and sheets contained in the Body Bag are considered active bodies, in that several functions (such as Boolean operations) can be performed on them while they are in the Body Bag.
Boolean Operations	Named for G. Boole, an English mathematician, Boolean operations are used to combine two entities (either solids or sheets or some combination of the two) to create a new, single body or sheet. The Boolean operations contained in the system are addition, subtraction and intersection. Boolean operations are destructive, in that the initial two bodies selected for the Boolean operation are deleted and only the resulting body remains active in the Workspace. The deleted bodies used in the Boolean operation become dormant bodies and can be retrieved from the History list. Non-destructive Booleans can be performed by holding down the `Alt` key. Non-destructive Booleans will generate the new body and place the two original bodies used in the Boolean operation in the Body Bag.

Chord Height	This term describes the method by which solids and sheets are rendered on the screen. Solids must be faceted when they are rendered. Facets are small planar surfaces that compose the rendered modes. The chord height setting determines the number of facets that will be used to render a solid model. The smaller the faceting chord height, the more facets that will be used to compose the model and the better the model will look on the screen. The overall chord height used by the system is specified in the File > Preferences > Display tab. Different chord heights can be applied to individual solids and sheets using the Chord Height specification found in the Properties dialog. The chord height can either be designated with the sliders or by entering a numeric value.
Coincident	When two features (from points to surfaces) are located in the same position in space, they are said to be coincident. For example, when two surfaces overlap and all points on one surface also lie on the other surface within the area of overlap, these surfaces are coincident. Also, if two points are in the identical position in 3D space, these points are coincident.
Continuity	This is a mathematical concept that the system uses to evaluate curves, usually used in reference to lofting and sweeping. Continuity refers to the smoothness of the curve. C0 continuity signifies that there are sharp corners in the selected curves. C1 continuity signifies that there are tangency's, but no corners.
Disjunct	This term means that items are disjoined or separated, not touching at all. Multi-lump bodies are composed of disjunct solid components. Tiles that are disjunct are not a continuos order.
Edge	This term refers to the curve between two adjoining surfaces. In order for a body or sheet to be considered a valid solid object, it must have a single edge between all adjoining faces. The user can view and select edges of solids or sheets using the Edge Selection button located in the Toolbar. Several modeling and machining functions require the selection of edges, such as blending, drafting, stitching/unstitching, and intersection machining.
Edge Loop	What bounds a surface definition into a finite bounded surface.
Face	This is the term used for a single surface of a body or sheet. Faces, however, contain more information than merely the surface definition. Faces have knowledge of all neighboring faces and they are adjoined. For example, one side of a cube would be considered a face. Every face is bound by a loop, which is composed of all of the connected edges that bound the face.
Geometric Modeling	The process of defining a model with simple geometric constructions such as points, lines, circles and splines. Geometry can be defined in either two- or three-dimensional space.
Internal Edge	An internal edge is one that is viewed from the inside of the model looking outward. The concept of internal versus external edges is useful when performing stitching operations on sheets. All edges that are successfully stitched become internal edges so that when the Show Internal Edges checkbox is unselected, the only edges that will be displayed are external edges. It is these external edges which still need to be stitched together. When performing stitching operations, turning off internal edge display

	provides the user with a method of determining which edges were not successfully stitched together. All stitched edges become internal edges.
Loft	A lofted shape is created by selecting a series of shapes that will be blended together using selected alignment points. Also referred to as skinning or blending
Loop	This term refers to the bounding curve of a face. A loop is the series of connected edges that provides the boundary or trim for the face. A face is composed of a surface bounded by a single loop. The faces of a body or sheet must have an adjoining edge in order for it to be a valid entity.
Modeling	The process of defining a part's shape and dimensions on a computer. Common types of modeling include geometric modeling, solid modeling, and surface modeling.
Multi-lump bodies	These bodies are composed of disjunct solids—solid pieces that do not intersect at any point. Multi-lump bodies are considered one entity by the system and can be identified by selecting the body. If all or more than one of the disjunct pieces becomes selected, then it is a multi-lump body.
Parametric	This term is used to describe more complex surfaces that are defined within a given set of parameters and not merely an equation. Parametric surfaces are often referred to as free-form surfaces. The system utilizes B-splines, which are a class of parametric surfaces. When modeling functions such as lofting or Coons patch are used, the resulting entity is composed of parametric surfaces.
Primitive Bodies	See "Atomic bodies"
Sheet	A sheet is the modeling entity that represents a surface. A sheet contains more information than a surface because a sheet has knowledge of the neighboring surfaces that surround it. A sheet is represented as a single object. Sheets do not have any thickness or volume. A sheet is the graphical representation of a surface or collection of surfaces.
Solid	A solid is an object that has volume. Solids can either be single bodies (lumps) or a collection of bodies (multi-lump body).
Solid Modeling	The process of defining a part as a solid object. The process begins with the creation of a simple solid known as an atomic or primitive body. Boolean operations can then be performed on an atomic body to create a new, distinct body.
Surface Modeling	The process of creating sheets as the foundation for a model.
Surface Trim	The edge of an island or a cavity that is within the faces selected to be cut.
Synch Points	See "Alignment points"
Target Face	A selected face on a body. This term is often used for a face that will be used in the selection of other faces or for modification.
Topology	This is the term used in solid modeling to refer to how specific faces of a body are positioned relative to other faces. Modeling functions that change the shape of a face do not necessarily affect the topology, unless the function requires that a change be made to the manner in which the faces are connected together at their edges. For example, the number of edges of the body is changed if the modeling function creates new faces, and therefore

	the topology is changed.
Vertex	A vertex is an endpoint of an edge.
Workspace	The Workspace consists of the drawing window, which is the main portion of the screen, and the Body Bag. Solids and sheets that are contained in one of these two locations are said to be active bodies. Modeling functions can only be performed on active bodies. Bodies that are no longer in the Workspace can often be retrieved from the History list.

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

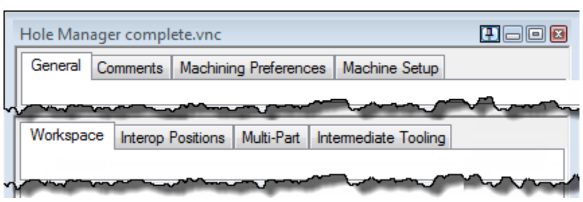
Screen text. Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

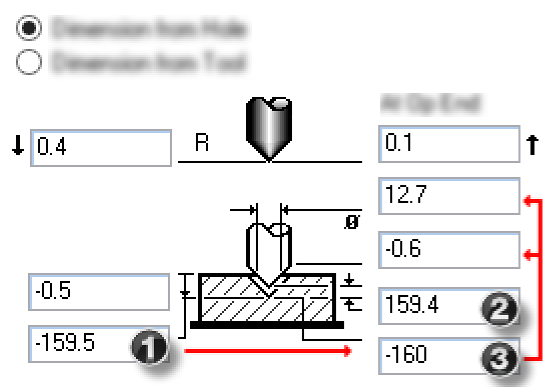
`Keystroke/Mouse`. Text with `this appearance` indicates a keystroke or mouse action, such as `Ctrl+C` or `right-click`.

`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

B

C

D

E

F

G

H

I

N

O

P

R

U

Z