GIBBSCAM | 2027 CAM for Production Machining

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2.5D Solids

GIBBSCAM

Contents

Introduction 6
About 2.5D Solids 6
What Is a 2.5D Solid Model? 6
Making Models 6
What Will the 2.5D Solids Option Model? 7
What Will the 2.5D Solids Option Cut? 7
What Is the Difference Between 2.5D Solids and the Other GibbsCAM Solids Modules? 7
Definitions 8
More About Solid Models 8

Interface 10
About the Interface 10
WorkSpace 10
Floating Toolbar 10
Command Toolbar 12
Main (Top Level) 12
Modeling Palettes 12
Bodies 13
History 13
Properties 14
Body Information 14
Part, Fixture, or Stock 15
Chord Height 15
Physical Properties 15
Multiple Body Properties 16
Body Bag 16
About the Body Bag 17
Body Bag Context Menu 18
Body Bag Color Display 19
Body Bag Pages 19
Body Bag Page Context Menu 19
Viewing Body Bag Pages 19
Selecting Body Bag Objects 20
Solid Menu Items 20
Edit 20

Modify 21
Solids Tools Menu 21
Plug-Ins 22
Context Menus 22
Body Context Menu 22
Face Selection Mode Options 25
Edge Context Menu 26
History Context Menu 27
Profiler Context Menu 27
Preferences 28
Display Tab 29
Render Faceting 29
Machining 30
Modeling 32
Introduction to Modeling 32
About Modeling 32
Solids 33
Sheets 33
Primitives/Atomic Solids 33
Workspace 34
Workgroups and Coordinate Systems 34
Boolean Operations 35
Recreate Mode 36
Rebuilding Solids 37
Modeling Reference 37
Surface Modeling Palette 37
Plane 38
Extrude Sheet 38
Revolve 38
Loft 39
Coons Patch 39
Sweep Sheet 40
Sheet from Face 40
Trim/Untrim Surfaces 40
Stitch Sheets 41
Unstitch Sheets 42
Untrim & Extend Surfaces 43
Solid Modeling Palette 43
Create Solid Palette 44
Advanced Solid Modeling Palette 51
Slice 57
Replace 57

Swap 57
Add 58
Subtract 58
Intersect 59
Separate 60
Geometry Creation from Solids 60
History List 62
Body Types 62
Body Names 63
Modifying, Recreating, and Rebuilding Bodies 63
Method 1: Create a New Solid 64
Method 2: “Locally” Edit an Existing Solid 65
Method 3: Replace/Swap and Rebuild 65
Method 4: History, Recreate, and Rebuild 66
Tips and Techniques 67
Machining 69
Introduction to 2.5D Solids Machining 69
2.5D Machining Details 69
Gen 3 Engine 70
Compatibility With Earlier Versions 70
Surface Tolerance 70
Selection Modes: Part, Constraint (Fixture), Stock 71
Stock Definition 71
Notes 72
Operation Stock Size 72
Fixtures 72
Contouring Process 72
Using the Profiler 73
Roughing Process 73
Material Only 74
Machining Preferences 74
Material Only Pockets 74
Optimizing Material Only for Solids 75
Solids Tab 76
Limitations of Create 2D Toolpath 81
Open Sides Tab 82
Appendix 84

Glossary 84

Conventions 88
Text 88
Graphics 88

Links To Online Resources 89

Index 90

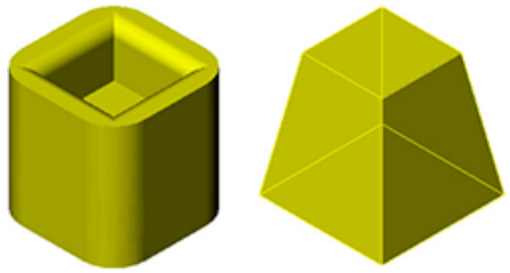
Introduction

About 2.5D Solids

This material in this guide describes machining 2.5D solids. You can define parts using solid and surface modeling techniques that can be directly machined using 2D, 2.5D and occasionally even 3D machining techniques. As a prerequisite, you should be familiar with the basics of geometry creation, coordinate systems, and basic machining. For information, see the *Geometry Creation*, *Mill*, and *Advanced CS* guides
Note: The capabilities and user interface described in this and other guides apply to GibbsCAM Industrial Edition with all product options licensed and active. GibbsCAM Viewer, GibbsCAM Operator and GibbsCAM Student Edition provide a subset of the full functionality.

What Is a 2.5D Solid Model?

A 2.5D solid can be cut with a series of 2D toolpaths at different Zs, producing analytic (lines and circles) toolpath feature output from underlying analytic model faces. A 2D solid is an XY shape extruded in Z – a circle produces a cylinder. In a 2D solid model all Z slices produce the same shape.



A “2.5D solid model” allows different Z slices to be different 2D shapes, but all slices must be 2D shapes. A taper on the walls of a 2D solid, makes it a 2.5D solid – cylinders are now cones but a Z slice still produces a circle. Every Z slice is different but slicing the corners still produces a circle. Adding top and bottom fillets or chamfers to a 2D model creates a 2.5D model.

Making Models

There are three primary methods of using 2.5D Solids to create part models that can be machined. The first method is creating solid models from part blueprints using the solid modeling functions contained in the system. There are many powerful modeling functions including adding, subtracting and intersecting solids, automatic chamfering and filleting, and several methods for generating solid bodies from geometry.
Secondly, the system can directly read solid model formats generated by other CAD packages. For example, more files can be directly opened by the system (some formats require the

purchase of an additional option). The system does not use an importation filter or any means of translation for these solid files but rather directly reads them.
The final method is through the importation of 3D surface files. The system recognizes and imports several surface entities. Once a surface file is brought into the system, it can either be made into a solid (using the Solidify functions) or 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 2.5D Solids machining capabilities. The standard Roughing and Contouring functions can be applied to solid bodies and sheets.

What Will the 2.5D Solids Option Model?

The 2.5D Solids option includes all the solid modeling functions useful to create 2.5D solid models, or to work from imported models, to model tooling like vise jaws, chucks, and fixtures, to correct and modify imported solids, and also to provide CAM focused modeling functions.

Modeling Functions in 2.5D Solids			
Union	Difference	Intersect	Separate
Create Spheres	Create Cuboids	Extruded shapes	Revolved shapes
Create Planes	Extract Sheets	Coons Patches (2.5D)	Shell and Offset
Simple Rounding	Chamfering	Unstitching bodies	Solidify sheets
Lofted shapes from 2 curves	Swept shapes (Drive Curve Plane 2D)	Swept shapes with one drive curve	Swept shapes with sharp corners

What Will the 2.5D Solids Option Cut?

The 2.5D Solids option will contour and pocket any solid – 2D, 2.5D, or 3D. It is optimized for, and will produce the best output for, 2D and 2.5D analytic faces. For 3D faces, it will start with the little line segments and attempt to fit arcs, replacing lots of lines. Using the Advanced CS option allows for rotary positioning that will make any side of the part the 2.5D side.

What Is the Difference Between 2.5D Solids and the Other GibbsCAM Solids Modules?

The 2.5D Solids option is far more powerful than the Solids Import option which allows you to open a solid model, extract geometry from the model and machine the geometry. 2.5D allows you to make the model or modify an existing model and machine it directly, with or without the use of geometry. The SolidSurfacer module provides more modeling techniques than 2.5D Solids, particularly for making 3D shapes, and provides greater solid model machining functions that produce 3D or 2D and 2.5D optimized toolpath.

Definitions

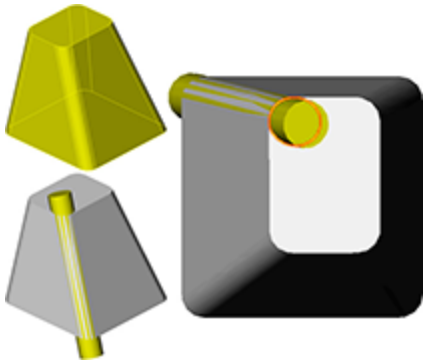
The terms and definitions provided below are used to describe objects and elements used by the system, and throughout this guide. More information can be found in Glossary.

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, while 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 while a solid only includes 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, group of faces (depending on how the surface was created) of a solid or 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.

More About Solid Models

In CAD terms, 2D simply means a shape lies in a plane. In CAM terms, a 2D toolpath does not change in Z. A 2.5D toolpath is a series of 2D toolpaths at different Z levels. Any 3D model can be cut as a series of 2D toolpaths at different Z levels – this is referred to as a 2.5D process. What makes the common CAM usage of the terms 2D solid and 2.5D solid different is the expectation of clean line, clean circle output – what is referred to as analytic toolpath features. As a CAM package, GibbsCAM uses the 2.5D CAM definition for the 2.5D Solids option. A 2.5D solid is, therefore, a solid that can be cut with a series of 2D toolpaths at different Zs, producing analytic toolpath feature output from underlying analytic model faces.
Let’s start with the CAM term “2D solid model”. We expect perfect analytic toolpath elements (lines and circles) from analytic solid faces (plane and Z axis cylinder). A 2D solid model (also known as a prismatic body) is an XY shape extruded in Z. A circle produces a cylinder face. In a 2D solid model all Z slices produce the same shape, and the slice of a cylinder face produces a circle segment.

A “2.5D solid model” extends this concept by allowing different Z slices to be different 2D shapes. But all slices must be 2D shapes, with analytic segments from analytic faces. Put a 10 degree taper on the walls of a 2D body, and it becomes a 2.5D body. The cylinders are now cones, but a Z slice still produces a circle. Every Z slice is different, but slicing the corners still produces a circle. A 2.5D solid model is still made up of analytic faces. This includes planes, spheres, orthogonal fillets/cylinders, Z axis cones, Z axis revolved shapes and XY base curve Z plane drive curve swept shapes. Adding top and bottom fillets or chamfers to a 2D model creates a 2.5D model. Since the 2D model definition is a subset of the 2.5D model definition, all 2.5D functions work equally well on 2D models, in the creation of analytic toolpath features.



What about “3D solid models”? Picture a cube and put a different angle taper on each wall, say 5, 10, 15, and 20 degrees. Then round the edges. The corners are no longer cylinders or cones. A slice will not produce a circle segment at the corner. This is not a 2.5D solid model, but a 3D solid model. (This example is illustrated to the right. The cylinder in the third image represents the filleted corner. If the cylinder is sliced along the XY plane the resulting profile is an ellipse.) We use the term 3D solid model for anything that exceeds the 2.5D definition. A 3D solid model can still be machined with the 2.5D Solids product.

Another common variation occurs with imported bodies. Some CAD systems output solids as all NURB faces (especially if the export options are not set properly). They look like planes and cylinders and look like 2.5D models, but they are not. They are 3D models with no analytic faces. The Simplify function attempts to detect and restore the analytic definition of suitable faces. The Solid Inquiry plug-in (see the Plug-Ins Guide) will display the nature of all selected faces for your review.

A 2.5D model has all 2D or 2.5D analytic faces. In reality many parts are not so pure, but contain a mix of 2.5D analytic faces and 3D faces. Not to worry. You will get the best output from 2.5D analytic faces, but the 3D faces will be cut just fine.

Differences Between 2.5D Processes and 2D, 2.5D and 3D Solid Models

2.5D Process	Machine 2 simultaneous axes. Control position of 1 axis point to point (G17=Z, G18=Y, G19=X)
2D Model	All faces are parallel or normal (perpendicular) to tool
2.5D Model	Setting tool normal to a face, the angle of the tool will not vary all the way around the part. This is anything that can be created with the “Wall Control” options in the process dialogs.
3D Model	All other parts. This includes parts with variable radii and fillets at an angle.

Interface

About the Interface

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

- WorkSpace on page 10
- Command Toolbar on page 12
- Bodies on page 13
- Context Menus on page 22
- Machining on page 30

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

WorkSpace

Floating Toolbar

The Floating Toolbar contains six items that are part of the SolidSurfacer interface. For more detailed information on these Toolbar items see the Interface section in the *Getting Started* guide.

Show Solids:
Show or hide all bodies, including sheets.

Render/Wireframe:

Render fully shaded objects, or simple wireframe.

Indicate Sheet Side:
Indicate the positive and negative sides of a sheet.

Face Selection:
Enable/disable face selection mode.

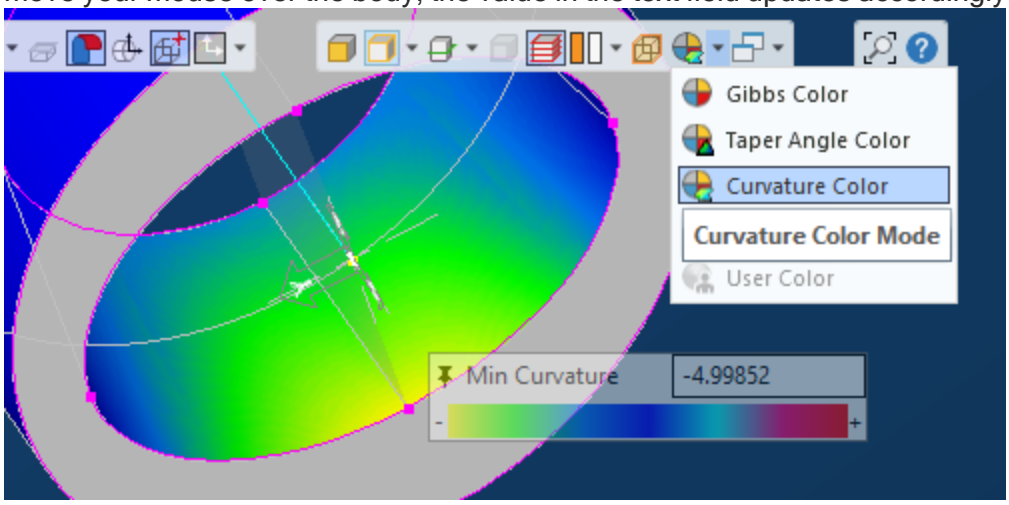
Edge Selection
Choose an edge selection mode (visible edges only, or all edges), or else disable edge selection.

Profiler:
Enable or disable the Profiler grid.

A seventh item on the floating toolbar, **Color Mode**, contains two items that are especially helpful for solids and surfaces.

Taper Angle Color and Curvature Color

These two modes make it quick and easy for you to visualize angles and curvatures. In either mode, the model displays its solids and surfaces using the spectrum of colors for this mode. You can hover your cursor over the body and wait for the floating dialog to appear. Then, as you move your mouse over the body, the value in the text field updates accordingly.

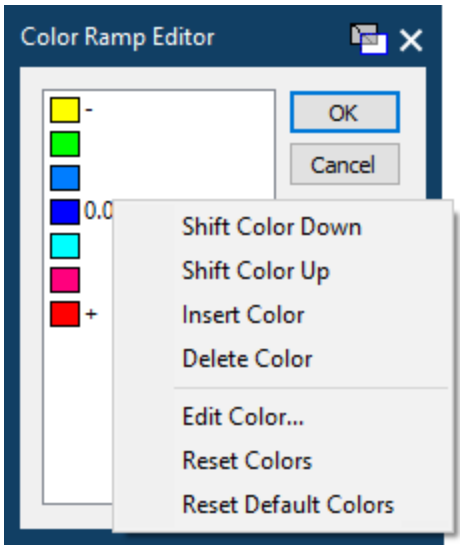


Both modes let you choose either an unsigned color ramp that runs from zero to maximum or else a signed color ramp that runs from maximum negative values on the left to maximum positive on the right.

To switch between signed and unsigned: Right-click the title bar of the Curvature dialog and use the context menu to toggle your preference, as follows:

- For **Taper Angle**, the choices are Taper Angle and ± Taper Angle.
- For **Curvature**, you can toggle Signed Curvature on or off, and you can also choose the type of curvature to measure: Min, Max, Mean, or Gaussian.

The default signed color ramp, - +, runs from yellow (maximum negative) through blue (zero) to red (maximum positive).
The default unsigned color ramp, 0 ±, runs from blue (zero) to yellow (maximum).
You can, if you want, modify a color ramp by right-clicking in it and selecting Edit. In the **Color Ramp Editor**, shown below, you can right-click any color to move it down (leftward on the ramp) or up (rightward), or to delete it. The editor's context menu also lets you insert a new color or edit an existing color. The color picker is described in “Appearance”.



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.

Main (Top Level)

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

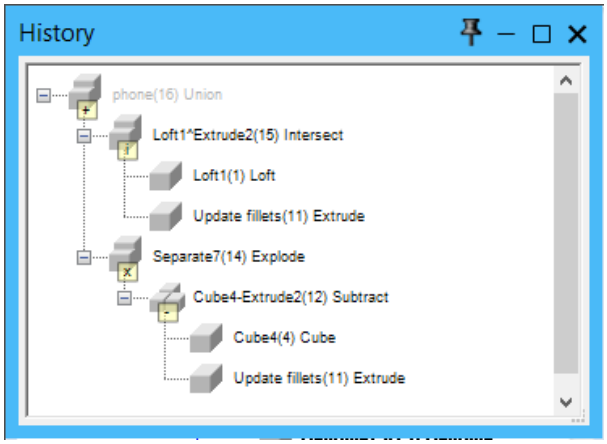
You use the Surface Modeling palette to perform sheet, or surface, modeling. You can extract surfaces from faces or create surfaces from geometry. Booleans from Solid Modeling palette are also used in sheet modeling. For more information, see Surface Modeling Palette.

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 .

Bodies

Each body (solid, facet body, 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 can even be rendered in wireframe and hidden.

History



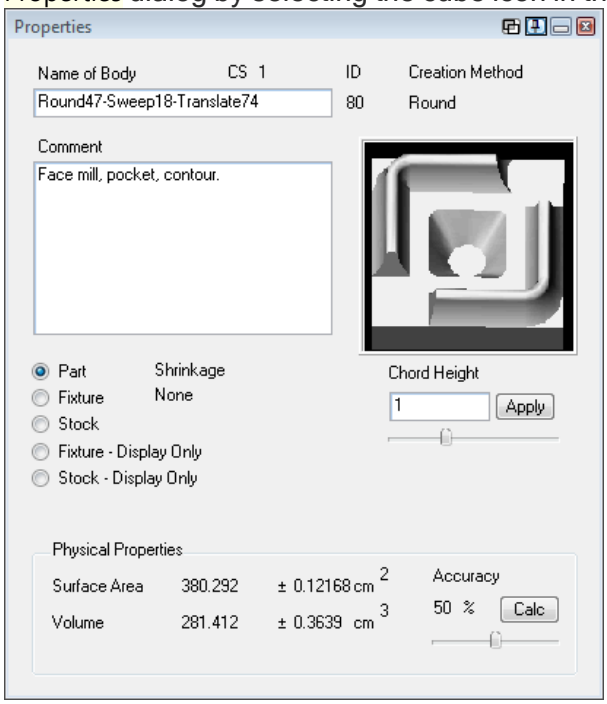
You access the History list from the body context menu. See Body Context Menu. The History list displays the creation list for any selected body. All of the bodies and functions that were used to create the selected body appear in the History list, even if they are no longer active bodies. The system 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 that contain information on the “parents.” “Parents” is the term used to describe the bodies (solids, facet bodies, or sheets) that were used to create the selected body. Solids and facet bodies 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

Body Information

You access the Properties dialog from the body context menu. For more information, see Body Context Menu.
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.

Physical Properties

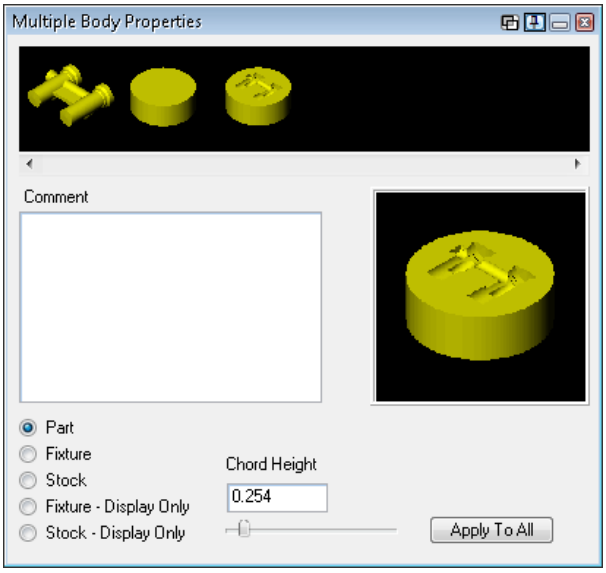
The Physical Properties section provides surface area calculations for solids and sheets, volume calculations for bodies, and surface periphery calculations for sheets. The Physical Properties section includes an Accuracy slide bar and a Calculate button. The Accuracy slide bar designates the amount 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

Multiple Body Properties

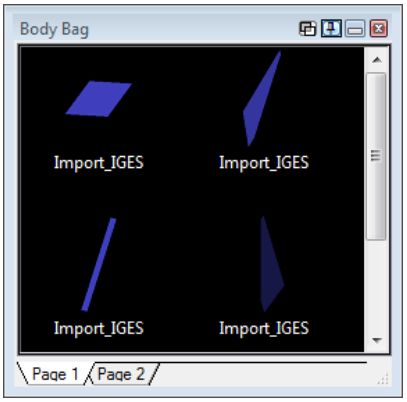
The Multiple properties dialog appears when more than one body is selected when the Properties command is chosen. This dialog helps to assign properties to many bodies at one time.



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 as well as their Chord Height and a Comment. Apply To All must be pressed in order 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 18. 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 18. 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.

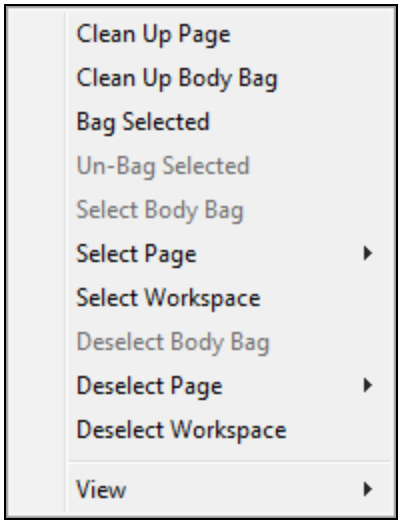
To increase or decrease the size of icons in the Body Bag:

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 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.

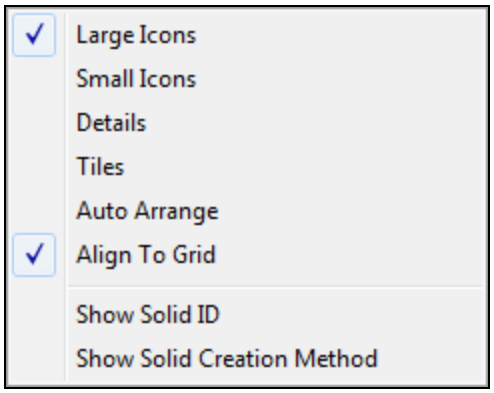
Un-Bag Selected:
Take any selected Body Bag icons and place the solids/sheets back into the drawing window from the Body Bag.

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 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 19. 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.

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 16. With this menu you can insert delete or rename pages.

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.

Solid Menu Items

Edit

Select Special

The `Select Special` menu items `Solids`, `Sheets`, and `Edges` enable you to select only the solids, sheets, and edges in a part file.

The Walls From Selected Edges item selects all faces that are tangent to the selected edges and perpendicular to the current CS. By Body Name and By Body Comment allow users to select bodies by entering their name or comment in the respective dialogs.

All Profiles selects all the available profiles found by the Profiler and Faces From Selected Profiles selects all faces that touch a selected profile.

Deselect Special

The Deselect menu contains the same items as the Select menu but act by deselecting (instead of selecting) entities.

Modify

Shrinkage

Shrinkage compensates for the rate at which an injection substance will shrink in a mold cavity. It performs a uniform or axial reduction or enlargements on selected solids. The range of shrinkage is -10% to 10%. Shrinkage can also be applied differently in each axis.

*Final Size = (100 - Shrinkage%) * Start Size /100*

Toggle Sheet Side

This item is useful when solidifying sheets into solids using the Offset solidify option. When sheets are converted into solids by offsetting, the offset must be calculated from one side of the sheet or the other. The Max and Min offset values are referenced from one side of the sheet. To offset the sheet from the other side, select the sheet and then select the Toggle Sheet Side item.

Solids Tools Menu

To access the Solids Tools menu:

From the Solids menu, look under Tools.

The Solids Tools menu provides the following options for diagnosing problematic solids. However, some items are strictly developer tools.

Check Body Validity

When this item is selected, the system checks to ensure that all selected entities are valid. If a sheet is not valid, it is deselected once the check is complete, allowing you to identify the problem. An error message identifying the specific problem also displays for each invalid entity.

Check Face Validity

This item runs a face validity check on the selected sheets. This function can also be performed by clicking on the Face Check button in the Stitch Utils dialog, and is useful for when stitching has failed to identify problem areas before attempting to stitch again.

Machining Face Check

This item checks the validity of selected faces to ensure they can successfully be machined. Machining Face Check is only necessary when using the Gen 2 Engine in surfacing operations. After validating the face(s), the system will display a message with information on the face(s) if the check passed or an error message on each of the bad faces.

Remove Unneeded Topology

This function attempts to merge any sheets that are defined by the same underlying surface definition into a single sheet, thus reducing the number of individual sheets and simplifying the entire part file.

Simplify

This function attempts to convert NURBS surfaces into analytic surfaces within a given tolerance amount. Often times when surface files are imported, analytic surfaces are converted to NURBS; this function converts those NURBS back into analytics.

Multipass Stitch

This option attempts to stitch all selected sheets together by performing a series of single passes. The Multipass Stitch option is the same as stitching using the Multiple Passes option in the Stitch Utils dialog.

Check Trimmed Surf. Polyline

This item verifies the validity of trimmed surface polylines to ensure proper machining. Check Trimmed Surf. Polyline is only necessary when using the Gen 2 Engine in surfacing operations.

Check Trimmed Surf. Edges

This item verifies the validity of trimmed surface edges to ensure proper machining. Check Trimmed Surf. Edges is only necessary when using the Gen 2 Engine in surfacing operations.

Plug-Ins

Many plug-ins are available for solids. For details on body-specific plug-ins, see the *Plug-Ins* guide.

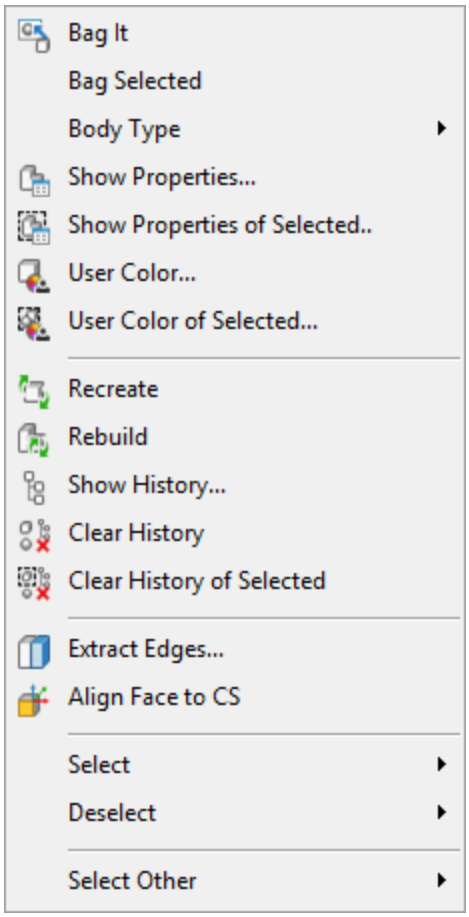
Context Menus

You access context menus by right-clicking on certain items. You can access the following context menus.

- Body Context Menu on page 22
- Edge Context Menu on page 26
- History Context Menu on page 27
- Profiler Context Menu on page 27

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 16.

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 "Multiple Body Properties" on page 16.

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 16.

User Color:

Enables custom color and material display of individual edges or faces. See "Properties" on page 14 and "Multiple Body Properties" on page 16.

User color of Selected:

Change the color and material of all currently selected faces or bodies. See "Properties" on page 14 and "Multiple Body Properties" on page 16.

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.

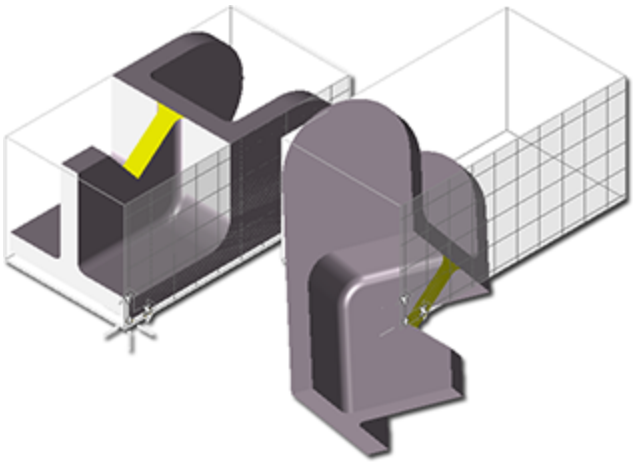
Extract Edges:

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.

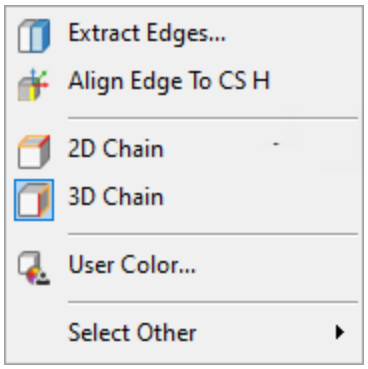
Transition Faces:

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.

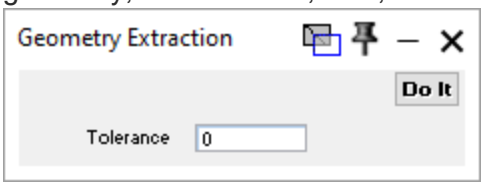
Edge Context Menu



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.

Extract Edges:

When you select this option, the **Geometry Extraction** dialog opens, allowing you to enter a value for Tolerance. When you click Do It, all edges in the selection set are extracted and copied to geometry, such as lines, arcs, and circles.



Note that, for solids that were stitched with a large tolerance, such as solids imported from other solid modelers through IGES or STEP, or imported from PTC Creo Parametric (Pro/E) or from Catia, gaps between faces might still exist in the solid that are not visible in the solid representation because of the tolerances of the solid modeler. To extract geometry from such solids, we recommend using a plug-in: On the **Plug-Ins** menu, under Solids, click Extract Edges.

Align Edge to CS Horizontal:

This option is available only when the selected edge is linear. Selecting it causes the solid to rotate in 2D so that the selected edge is parallel to the horizontal (H) axis of the CS, without shifting the origin

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:

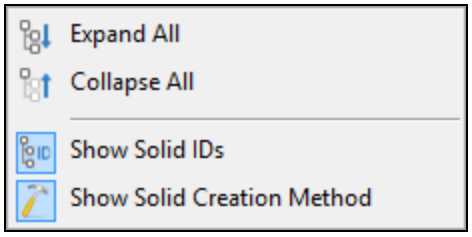
This opens the **Set Color** dialog, allowing you to assign colors, material and transparency values to individual edges.

Select Other:

This displays a list of edges that intersect with the selected edge. 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:

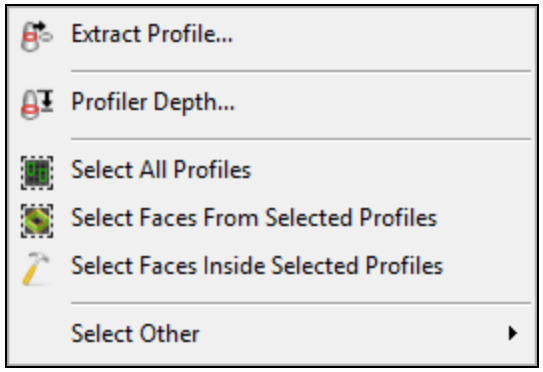
The Solid ID is a system-assigned positive integer that uniquely identifies each body.

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.

Profiler Context Menu

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

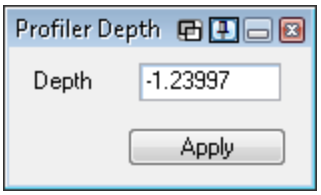


Extract Profile:

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:

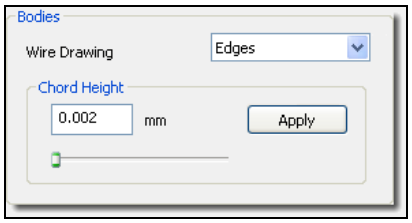
Selects all faces that are contained fully within the boundary of the selected profile and that are not tangent to the profile. Any face within the boundary is selected, regardless of its depth.

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 29.



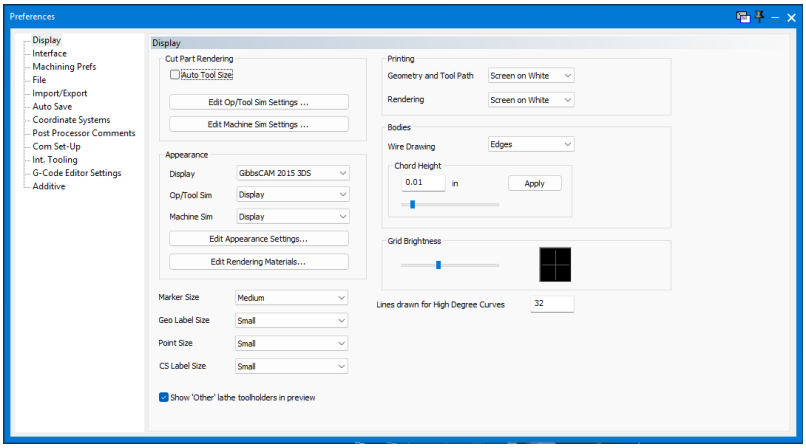
You can display bodies as rendered solid objects or as wireframe drawings. The Render/Wireframe button in the Floating toolbar determines whether solids and sheets are rendered as objects or wireframe drawings. The Wire Drawing setting defines whether it displays edges or facets of bodies. For more information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

Display Tab

To access GibbsCAM preferences:

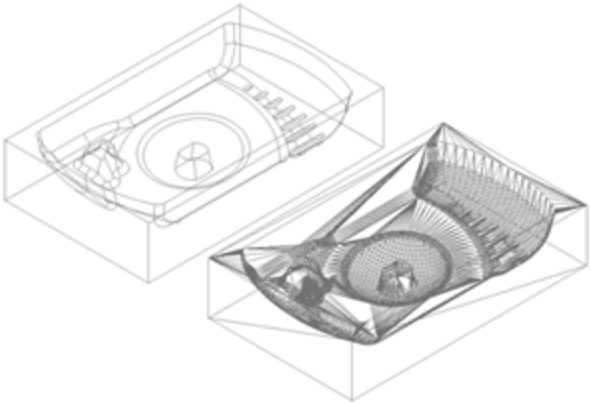
From the File menu, select Preferences.

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.



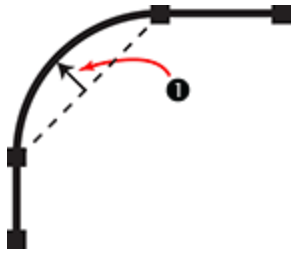
Chord Height:

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 29.

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

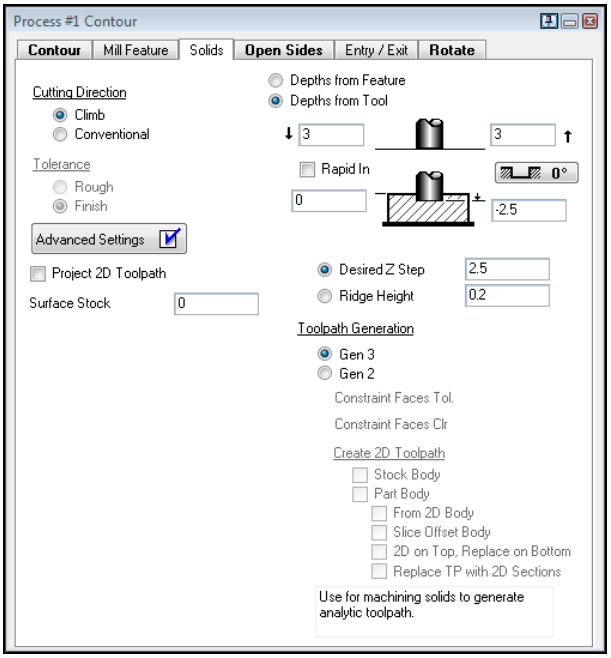
The system uses a global faceting chord height which is applied to the entire part model. The global chord height is applied to all solids and sheets that are created or imported.
To set the global chord height:

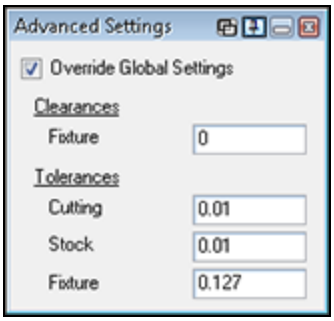
1. Click File > Preferences. The Preferences dialog appears.
2. Click the Display tab.
3. Under Chord Height, type a number in the text box or drag the slider to change the value.

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.

Machining

For Contouring and Roughing machining processes, you can set default values and options on the process dialogs Solids tab. The Solids tab contains options for overriding global settings for each operation type. For more information, see “Solids Tab” on page 76.


Process #1 Contour
Contour
Mill Feature
Solids
Open Sides
Entry / Exit
Rotate
Cutting Direction
Climb
Conventional
Tolerance
Rough
Finish
Advanced Settings
Project 2D Toolpath
Surface Stock
0
Depths from Feature
Depths from Tool
3
3
Rapid In
0°
0
-2.5
Desired Z Step
2.5
Ridge Height
0.2
Toolpath Generation
Gen 3
Gen 2
Constraint Faces Tol.
Constraint Faces Clr
Create 2D Toolpath
Stock Body
Part Body
From 2D Body
Slice Offset Body
2D on Top, Replace on Bottom
Replace TP with 2D Sections
Use for machining solids to generate analytic toolpath.


Advanced Settings
Override Global Settings
Clearances
Fixture
0
Tolerances
Cutting
0.01
Stock
0.01
Fixture
0.127

Modeling

Introduction to Modeling

This section includes the following topics:

- About Modeling on page 32
- Solids on page 33
- Sheets on page 33
- Primitives/Atomic Solids on page 33
- Workspace on page 34
- Workgroups and Coordinate Systems on page 34
- Boolean Operations on page 35
- Recreate Mode on page 36
- Rebuilding Solids on page 37

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. Facet bodies can also be used to model parts.

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 can 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.

A facet body, created from a solid object or imported from a different CAD/CAM system, can be manipulated in ways relating to the body's faceting, meshing, and smoothness. It can also be manipulated by most items in the Solid Modeling palette, such as Boolean, offsetting, trimming, and others.

Solids

A solid is an object that has volume and is not a facet body. A solid can consist either of a single body (a lump) or a collection of bodies (multi-lump body). You can view solids either as wireframe drawings or as rendered objects. However, for performing modeling functions, you can only select rendered objects. Selecting a solid will show if it is a single lump or a multi-lump body. A multi-lump body can be separated to create multiple bodies.
Unselected solids are rendered in grey. Selecting a solid changes its color to yellow. A solid that is defined as stock is rendered in blue. A solid that is defined as a fixture is rendered in red.

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 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 further information on these subjects, please refer to 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.

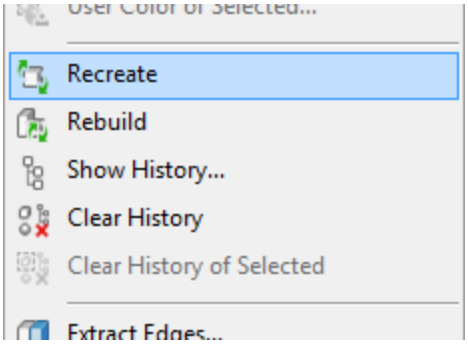
Intersection:

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.

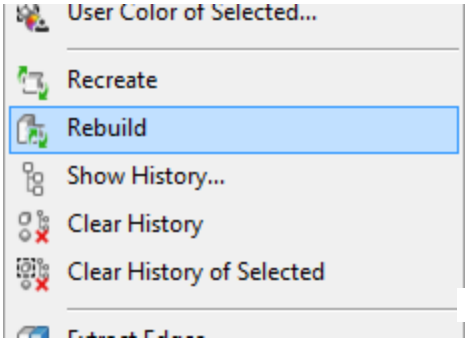
When in Recreate mode, the body selected to be recreated is drawn in red and placed in the Body Bag. Exit Recreate mode by selecting the body drawn in red or selecting the Exit Recreate item from the body context menu. If applicable, the parent body or bodies that were used to create the body to be recreated 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.

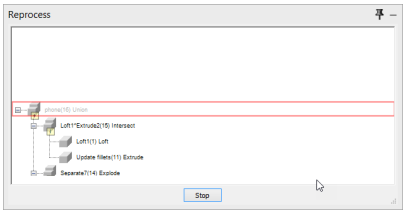
Recreate only affects the creation of the selected body. To recreate its parents, they must be brought back from the History list.

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. Bring a previous body back 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.

1. Plane
5. Coons Patch
8. Trim / Un-Trim

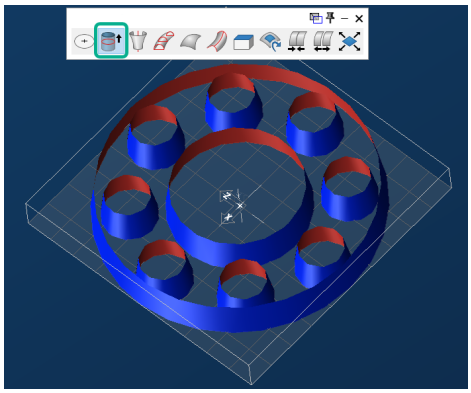
2. Extrude Sheet
3. Revolve
4. Loft
6. Sweep
7. Sheet From Face
9. Stitch
10. Unstitch
11. Untrim and Extend

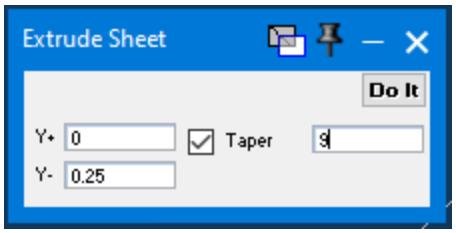
Plane

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

Extrude Sheet

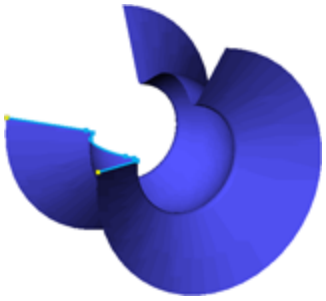
This button opens the Extrude Sheet dialog which lets you extrude 2D geometry or a closed or open profile in the Z+ and Z- directions to form a sheet.
The function for extrude sheet performs similarly to extrude solid. For more information, see " Extrude " on page 45

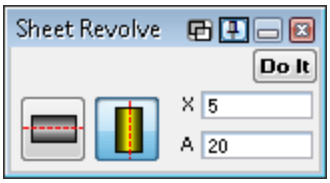




Revolve

This button opens the Sheet Revolve dialog which lets you revolve a shape around the horizontal or vertical axis by a specified number of degrees to create a sheet.



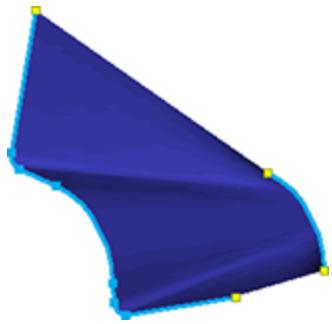


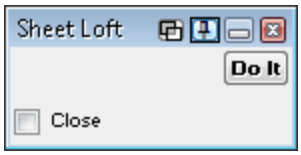
Sheet Revolve dialog

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 counterclockwise 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.

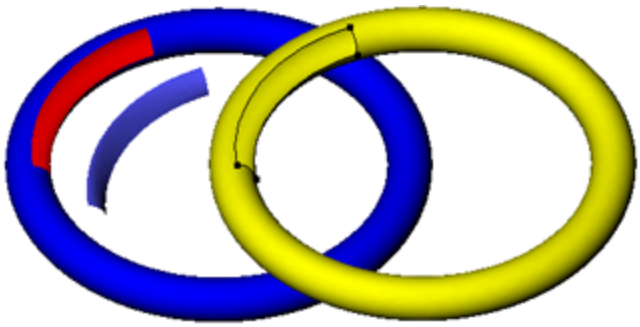




Sheet Loft dialog

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

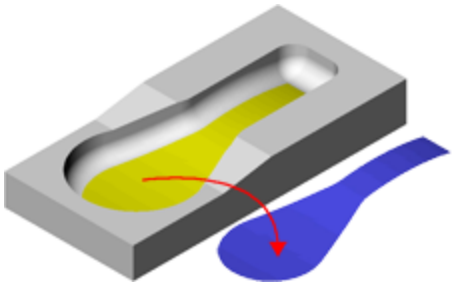
patch surface can be created if there are more than three or four line segments but the connected splines have three or four distinct corners.

Sweep Sheet

The Sweep Sheet function is nearly identical to the corresponding function in Solid modeling; see " Sweep Solid " on page 48. 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. For additional information, see " Sweep Solid " on page 48.

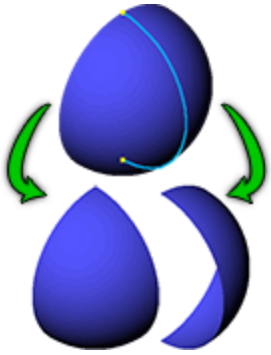
Sheet from Face

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 Toolbar, 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

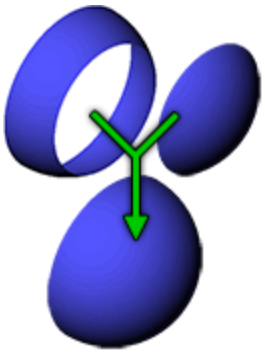
This button performs both the trim and untrim functions depending on the entities selected when the button is clicked. If a sheet and geometry are selected, the system will attempt to perform the trim operation. The trim function breaks a single sheet into two separate sheets at the selected trim geometry. 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

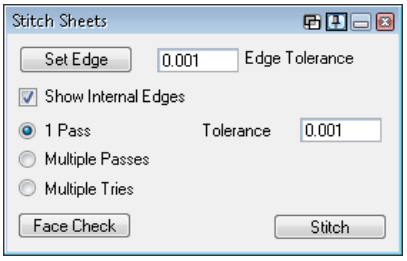


This button will open the Stitch Sheets dialog. This dialog provides different methods for stitching sheets together as well as tools to analyze stitched sheets. In order to stitch sheets, the user must select all the sheets to be stitched, choose a stitching method from the Stitch Sheets dialog and 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". You cannot specify a tolerance 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

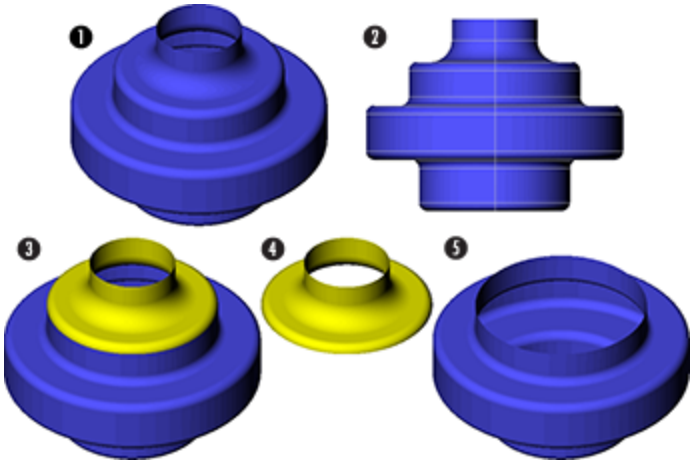
This option is similar to Multiple Passes in that it takes incremental passes ranging from the minimum tolerance (0.00002 mm or 0.00000079") to the maximum tolerance, which is the Tolerance value entered in the dialog. The system will attempt to stitch all the sheets together at each tolerance increment, 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

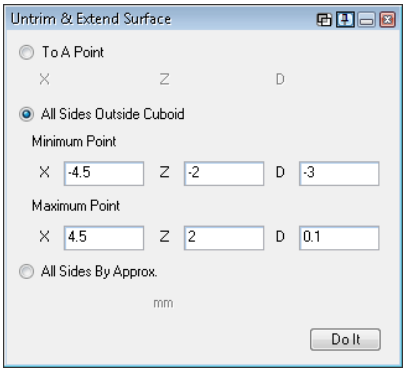
This button will unstitch or detach faces of a sheet. This will also convert solids into sheets. The faces will be unstitched at the edge loop which bounds the selected face or faces. An example of unstitching is illustrated below. The top row of pictures shows geometry revolved to create a single multi-faced 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.

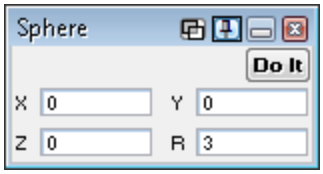
Create Solid Palette

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

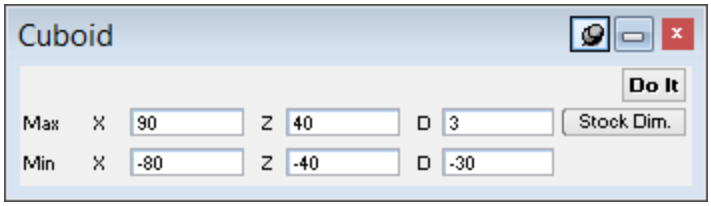
Sphere

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.



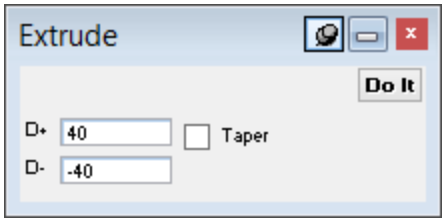
Cuboid

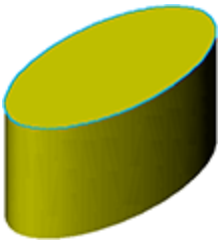
Opens the Cuboid dialog which you can use to create cubes and rectangular bodies. Enter a minimum and maximum horizontal, vertical and depth value to define the volume of the cuboid. These values are measured from the origin of the current coordinate system. The labels used in the dialog may vary when 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.



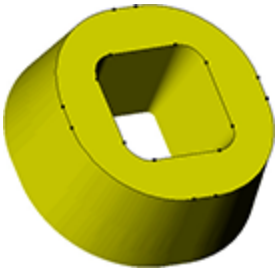
Extrude

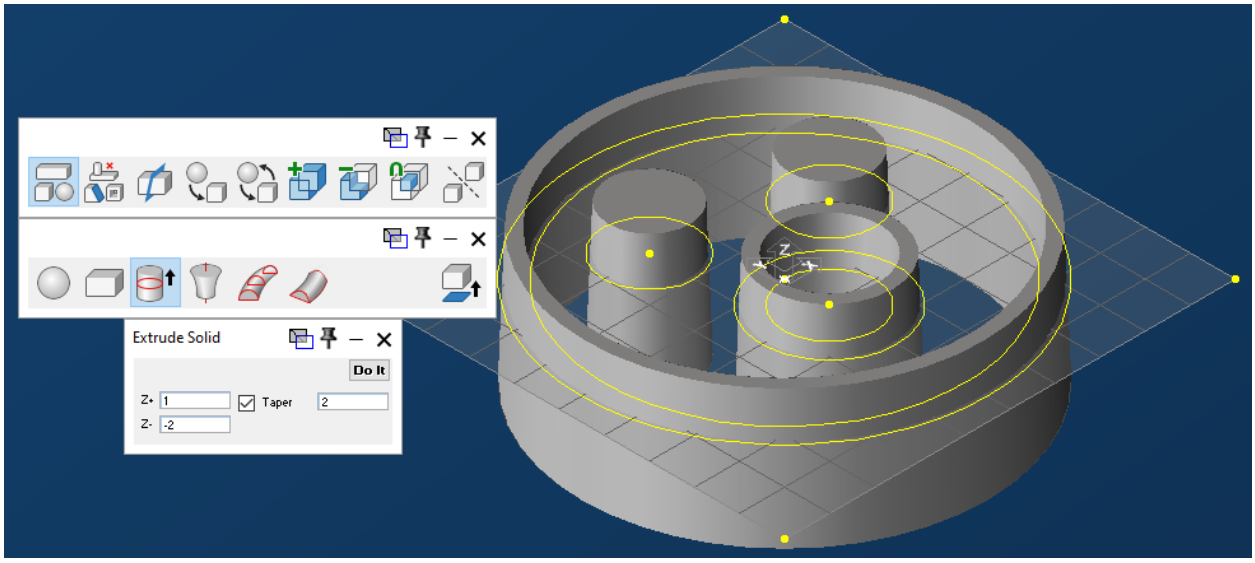
Click this button to open the Extrude dialog, which you can use to create solids by selecting one or more closed shapes and extruding along the depth axis. Closed 2D shapes can be extruded along the depth axis of the active CS in the positive and/or negative direction based on the values you enter. The extrusion starts at the depth location of the selected geometry. Click the Do It button to create the extruded solid or solids.
If you select multiple closed shapes, you can extrude them all in the same direction and extent. When extrusions overlap or intersect, nesting behavior is applied when possible, with a default towards union in ambiguous selections.





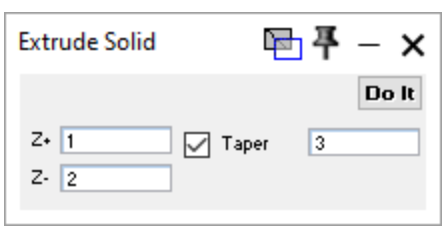
Straight and Tapered extrusions work with multiple geometry loops. The order of selection is important because the system uses the loop that is selected first as the outer profile.





Tapered Extrusions:

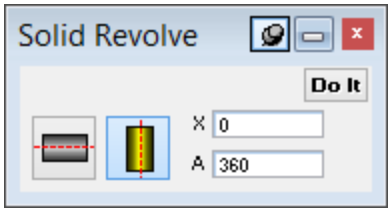
You can taper an extruded solid. Type a value in the Taper box to specify the angle of the taper. You can only taper an extrusion in one direction along the depth axis in order to properly calculate the taper on the solid. When you enter a value in the Taper box, the negative depth specification is grayed 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.

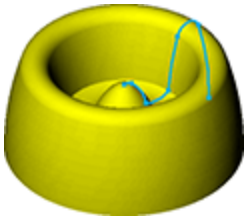


Revolve

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.

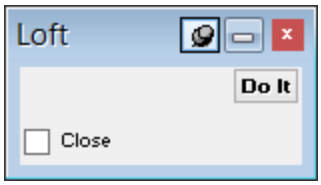
Likewise, if the vertical axis is the axis of revolution, you must enter a horizontal value to specify the position of the vertical axis that will be the revolution axis. The value entered in the A text box is the angle (specified in degrees) the selected shape is revolved around the selected axis. A 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.

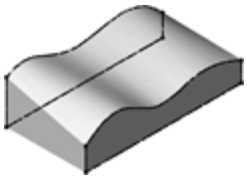




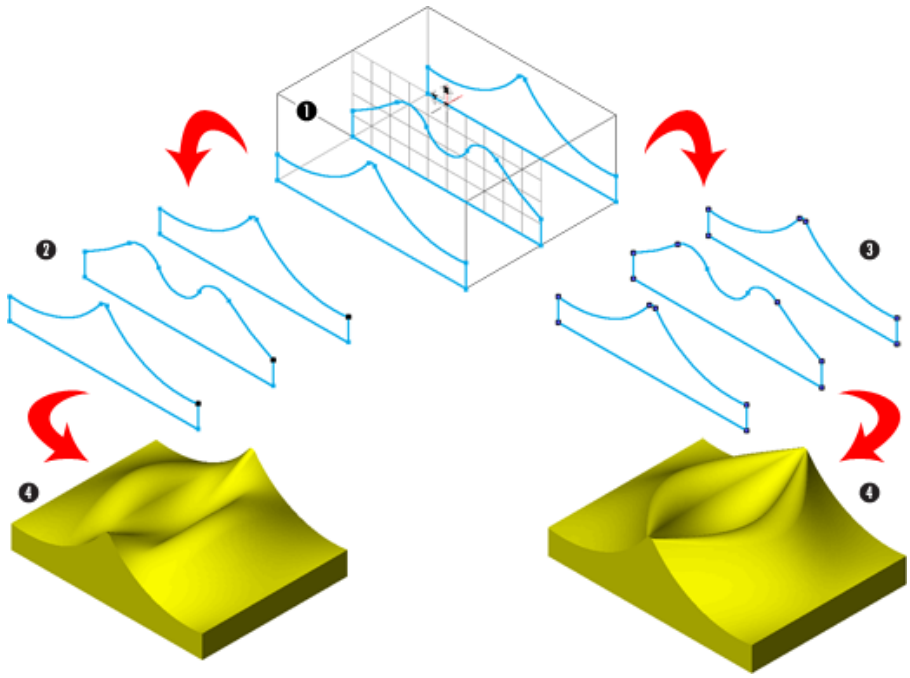
Loft

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.

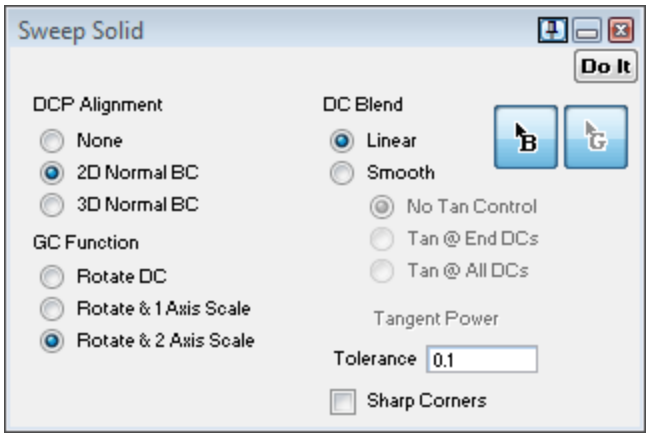


1. Curves to be lofted.
2. 1 alignment point is selected.
3. 6 alignment points are selected.
4. The resulting lofted solid

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.



Drive Curve Plane (DCP) Alignment:

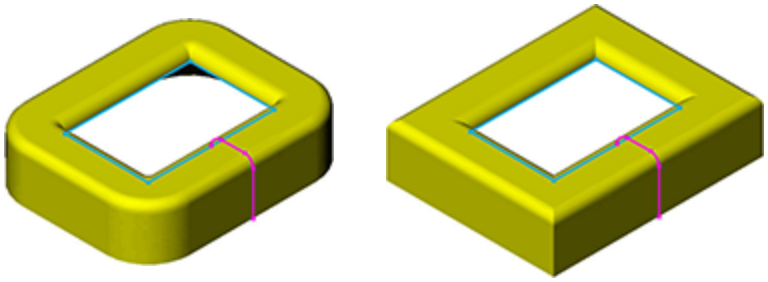
The DCP Alignment setting determines the alignment of the drive curve plane in reference to the base curve, that is, how the drive curve is swept around the base curve.

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.

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 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.

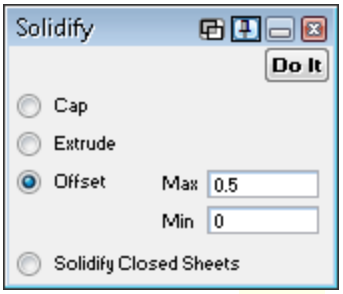
- Drive curves must be planar.
- All drive curves must be closed shapes. Open terminated shapes can be used for the drive curves only when these open shapes can be capped by a single plane.
- Select drive curves by choosing alignment points on each of the shapes.
    - Alignment points must be connectors 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.

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.

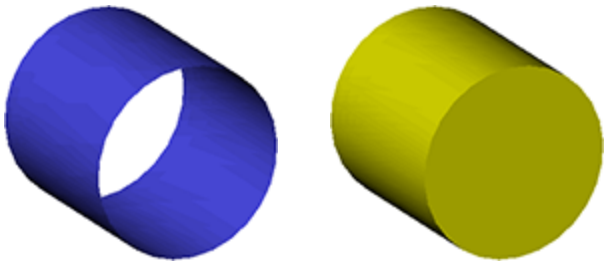
Solidify



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.

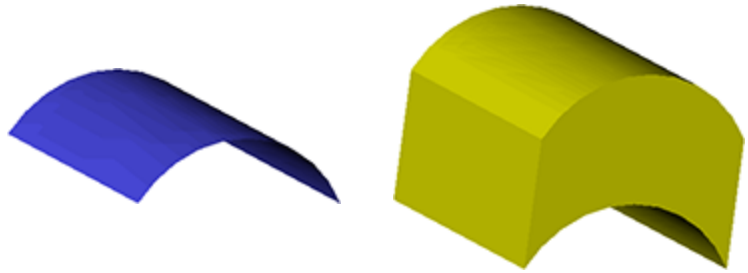


Solidifying a sheet using Cap

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.

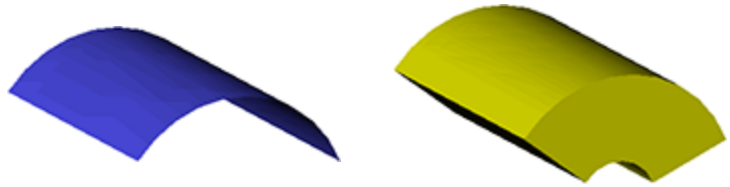


Solidifying a sheet with Extrude

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.



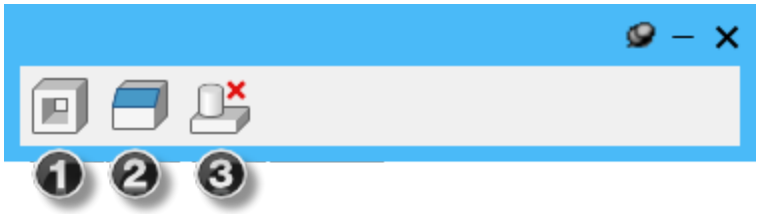
Solidifying a sheet with Offset

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, and Unstitch Solid functions. Each function is described below.



1. “Offset/Shell” on page 52
2. “ Blending ” on page 54
3. “ Unstitch Solid” on page 54

Offset/Shell

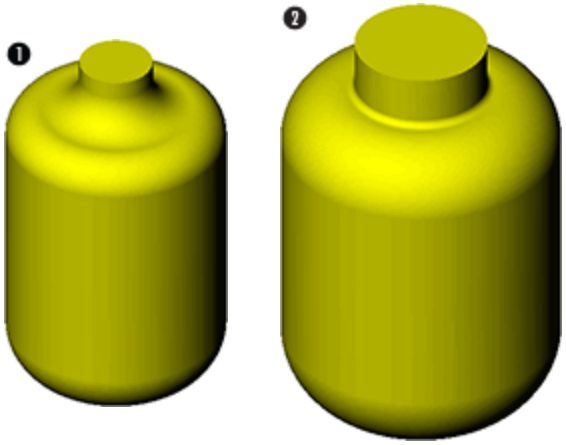
This button is located in the Advanced Solid Modeling palette and accesses the Offset/Shell dialog. There is an Offset button and a Shell button within the dialog. Each is described below.

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.

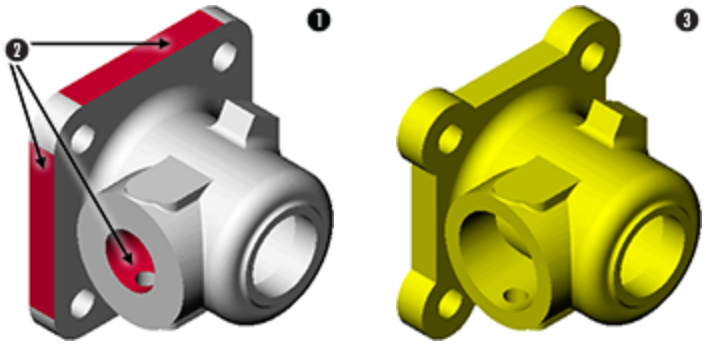
In Example of an Offset solid, the original solid is a canister and the offset is enlarged by a given amount. Note the change in the size of the fillet at the top of the bottle – it is a much smaller fillet than on the original solid. This is due to the fact the system must extend the other faces to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

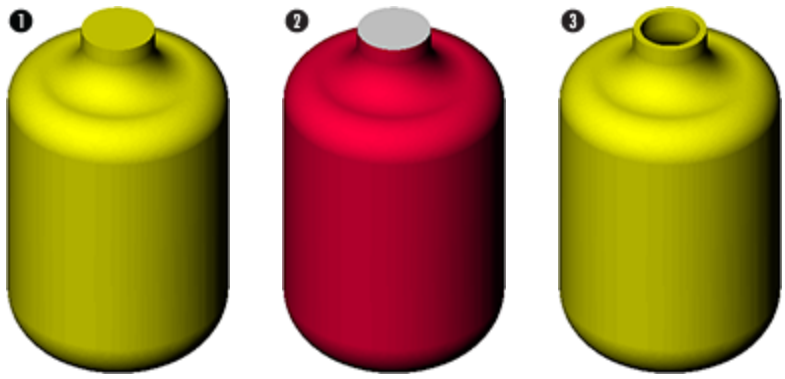
In certain instances, the offset function will not succeed because the specified offset amount creates excessive topology changes. Topology is the term in solid modeling for the manner in which specific faces of a solid are positioned relative to each other. Modeling functions that change the shape of 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.

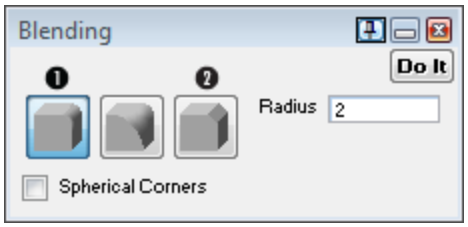


1. Original Solid
2. Solid selected with top face unselected
3. Result

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. 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.

Constant Chamfer:

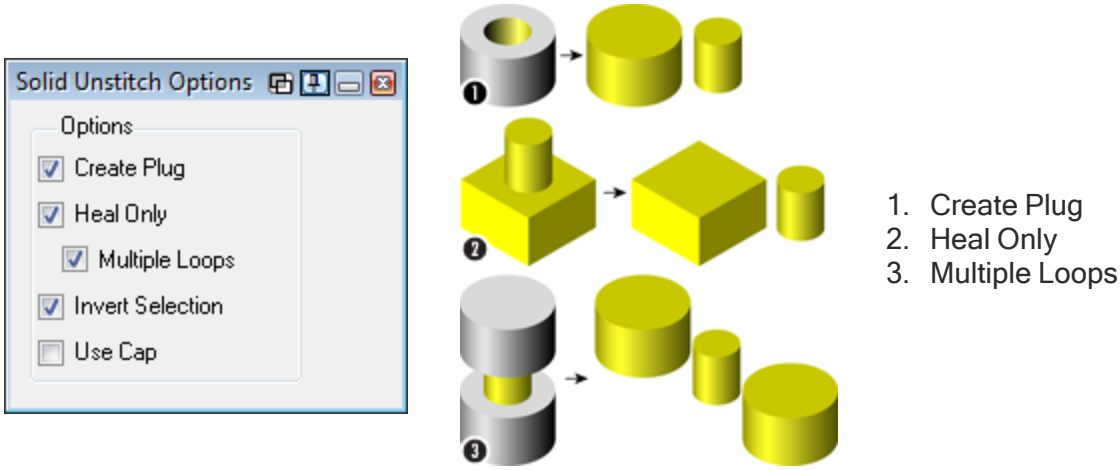
When you click the Constant Chamfer button, selected edges are chamfered based on the Length value entered. The chamfer is calculated by offsetting both faces joined at the selected edge by the length value specified and then finding the intersection of these offset faces. From the intersection, normals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

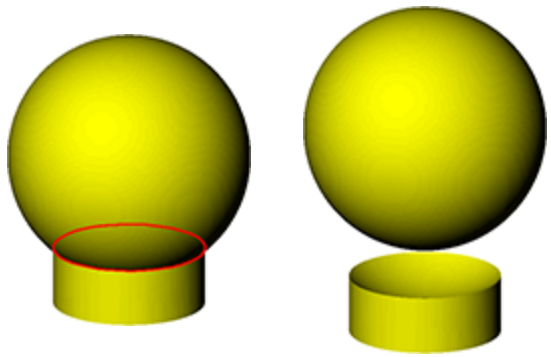
If unstitching cannot be done based on the selection, the selection will invert and reattempt the unstitch.

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

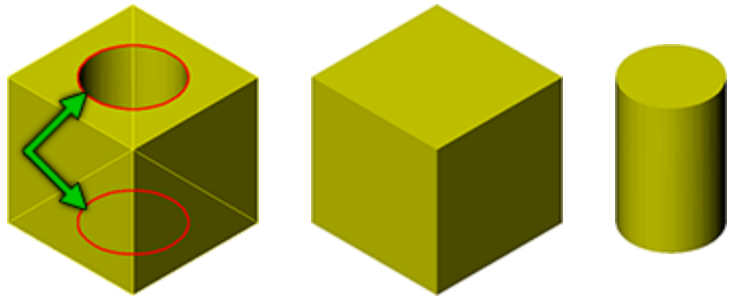
In Example of unstitching the original solid is a sphere with a cylindrical base. Selecting the intersecting edge to unstitch with Create Plug enabled will create two solids. One will be the sphere, which will not have a hole where the cylinder was previously attached. The other solid will be a cylinder: 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

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.

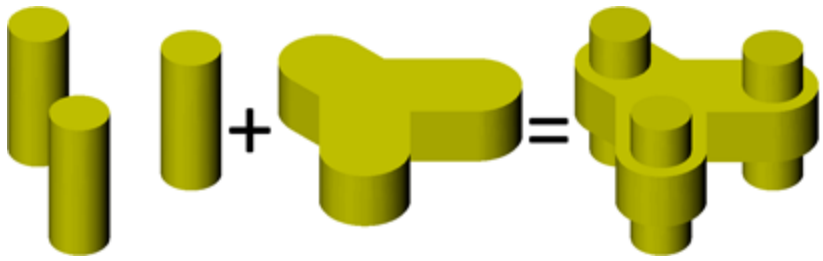


Figure 1: Solid + Solid addition

The figure below indicates the edges of the overlapped sheets.

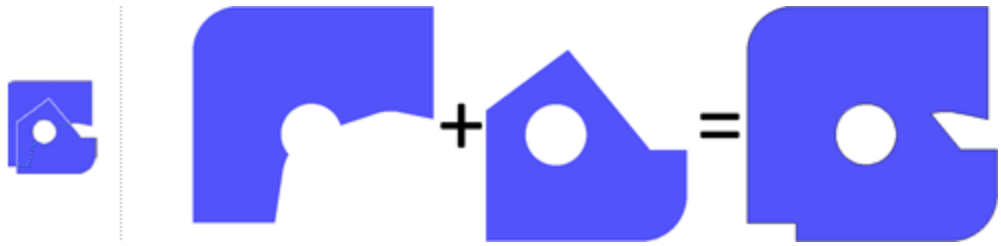
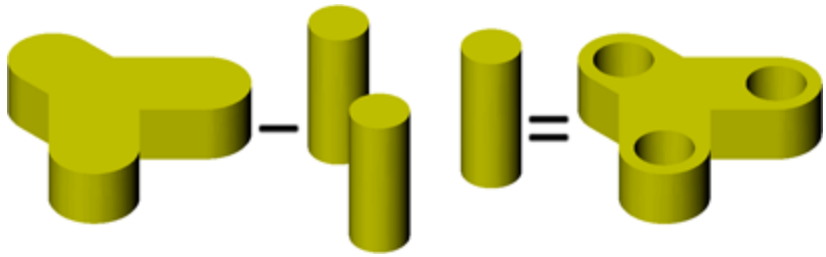


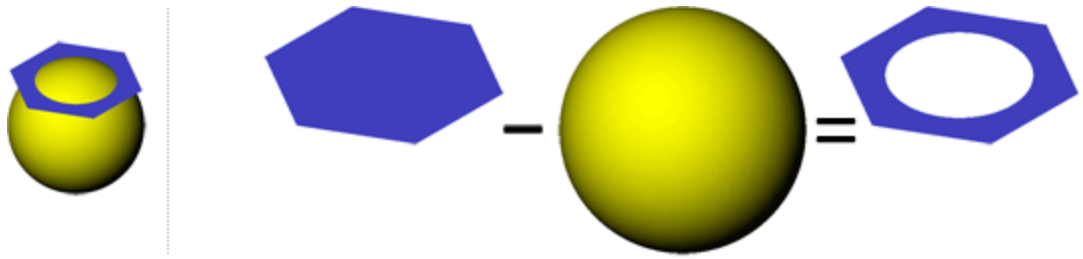
Figure 2: Sheet + Sheet addition.

Subtract

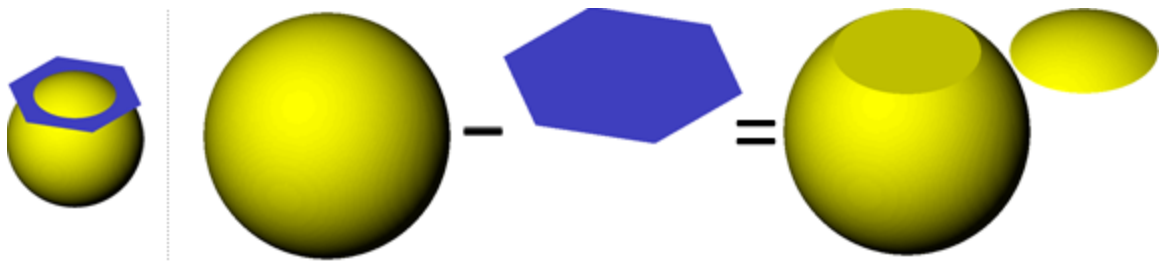
The subtraction boolean operation subtracts the common area of one body from another. The order in which you select the bodies is important because the second body you select is subtracted from the first body you select. The second body is deleted after the operation is complete. All bodies must be coincident 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.



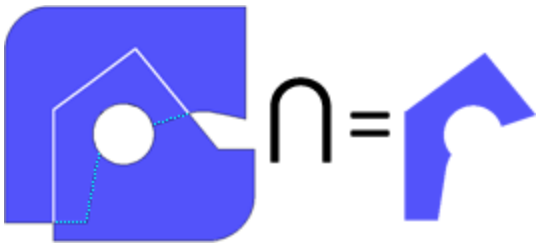
Solid - Sheet = a multi-lump body if separated becomes two individual bodies

Sheet - Sheet = removing the common area of a sheet

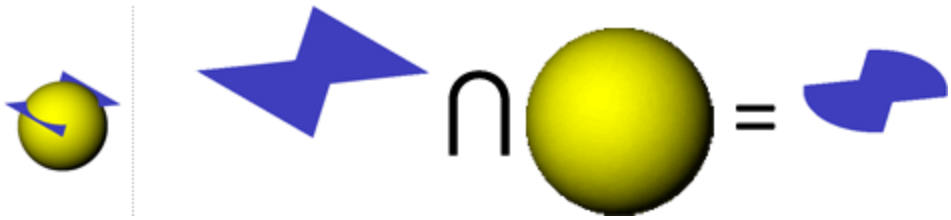
Intersect

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



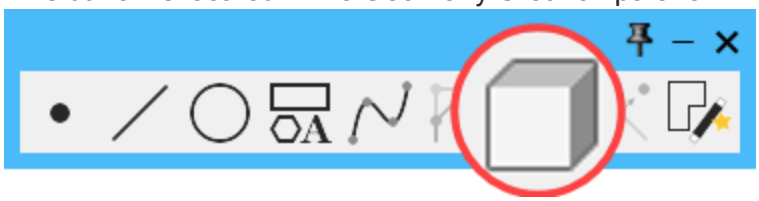
Sheet intersect Solid, resulting in a sheet trimmed by the solid

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.

Parting Line. Each of these functions is described below.

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

Geometry Extraction

The geometry extraction function creates geometry from selected edges of solids and sheets. In order to view edges of a solid or sheet, the system must be in Edge Selection mode. Connected shapes will be created if the selected edges create a closed loop. Typically, this function will extract 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 will consist of lines and circles. A tolerance of zero is recommended when extracting geometry that is definitely a circle or a line.

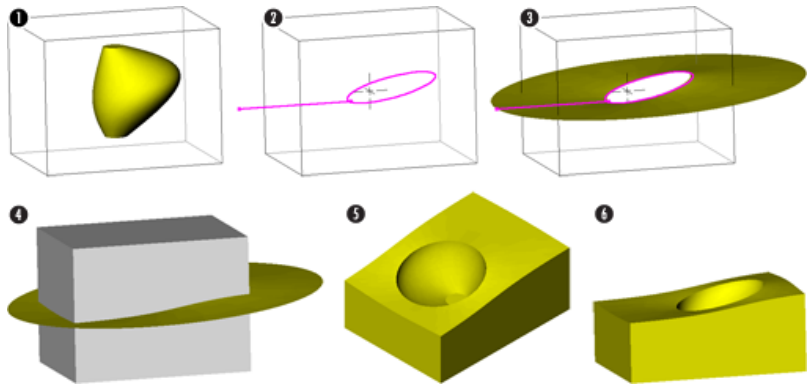
Hole Extraction

This function is used to create circles from holes in solids or sheets. This function allows the user to 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 will be based on the bottom of the hole(s), making it easy for the user to determine the depth for the drilling operation.

Parting Line

This function 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 will be 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.

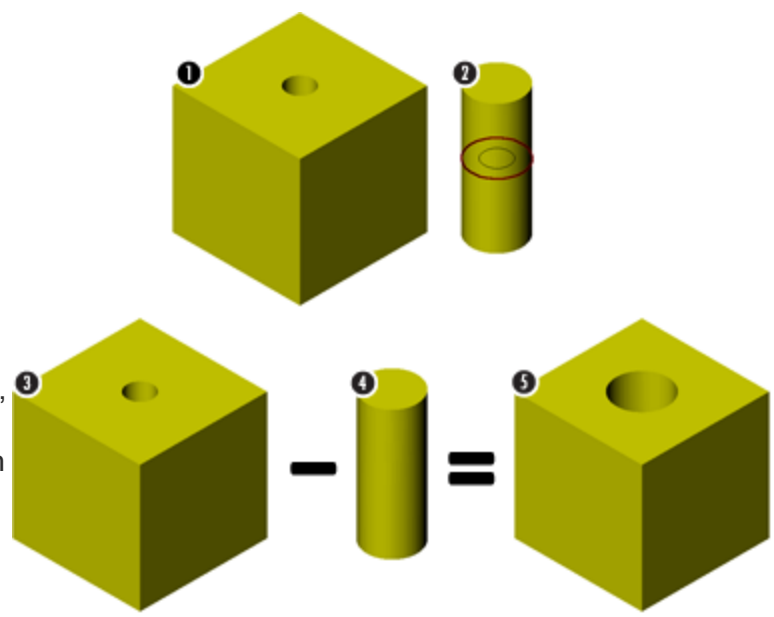
Rendered facet bodies, lump and multi-lump bodies contain a symbol on the icon to clarify what operation was performed in order to reach its state in the History list. The following is a list of characters that appear on lumped and multi-lump bodies.

Symbol	Function	Symbol	Function	Symbol	Function
+	addition	h	Stitch	t	Translate
–	Subtraction	i	Intersect	T	Duplicate And... Translate

Symbol	Function	Symbol	Function	Symbol	Function
–	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
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.

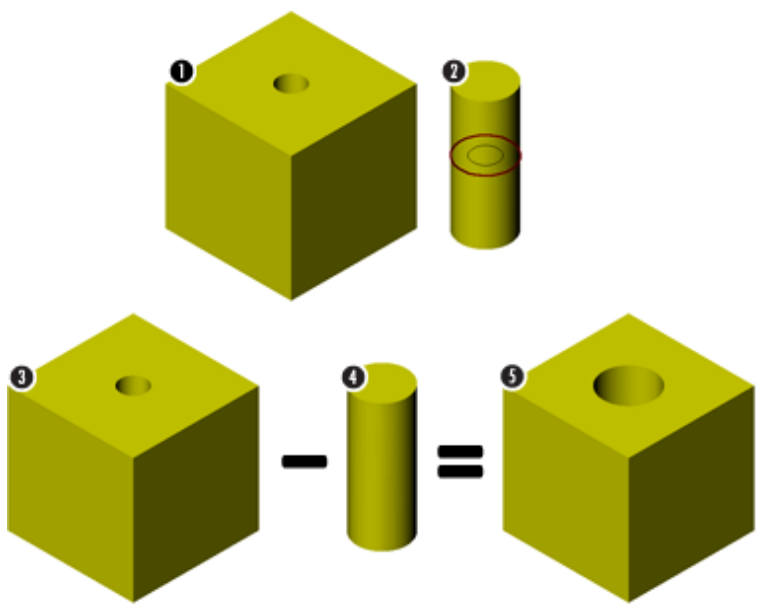
- Create a completely new solid (component) and manually recreate the final part to include the new component.

- 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.



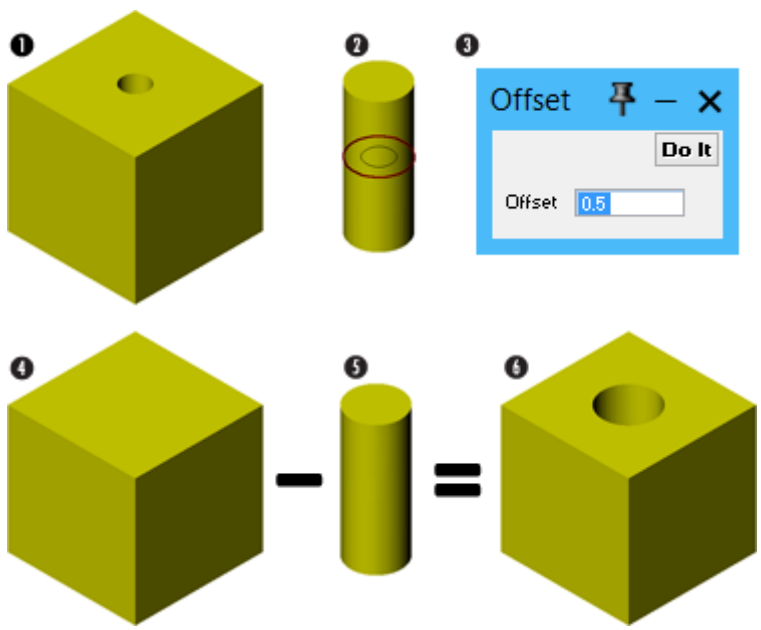
1. Original model needs larger hole
2. New extrusion
3. Original cube
4. Larger cylinder
5. New model with larger hole

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.

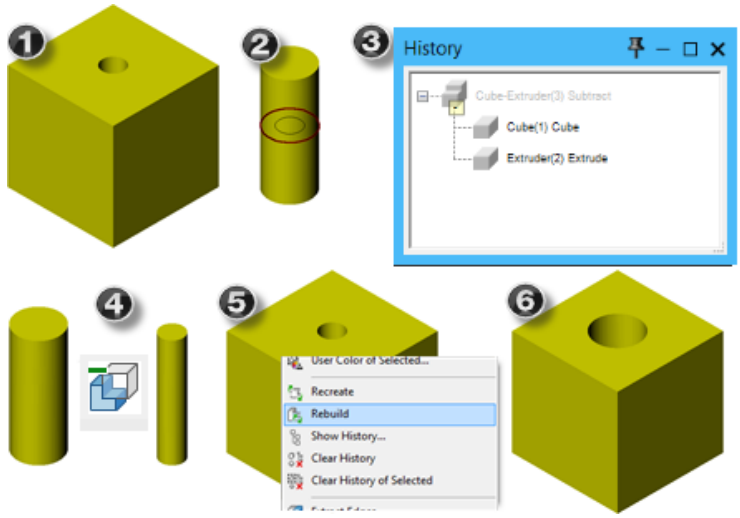


1. Original model needs a larger hole
2. Original cylinder
3. Value by which to offset the cylinder
4. Original cube
5. Offset cylinder
6. New model with larger hole

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.

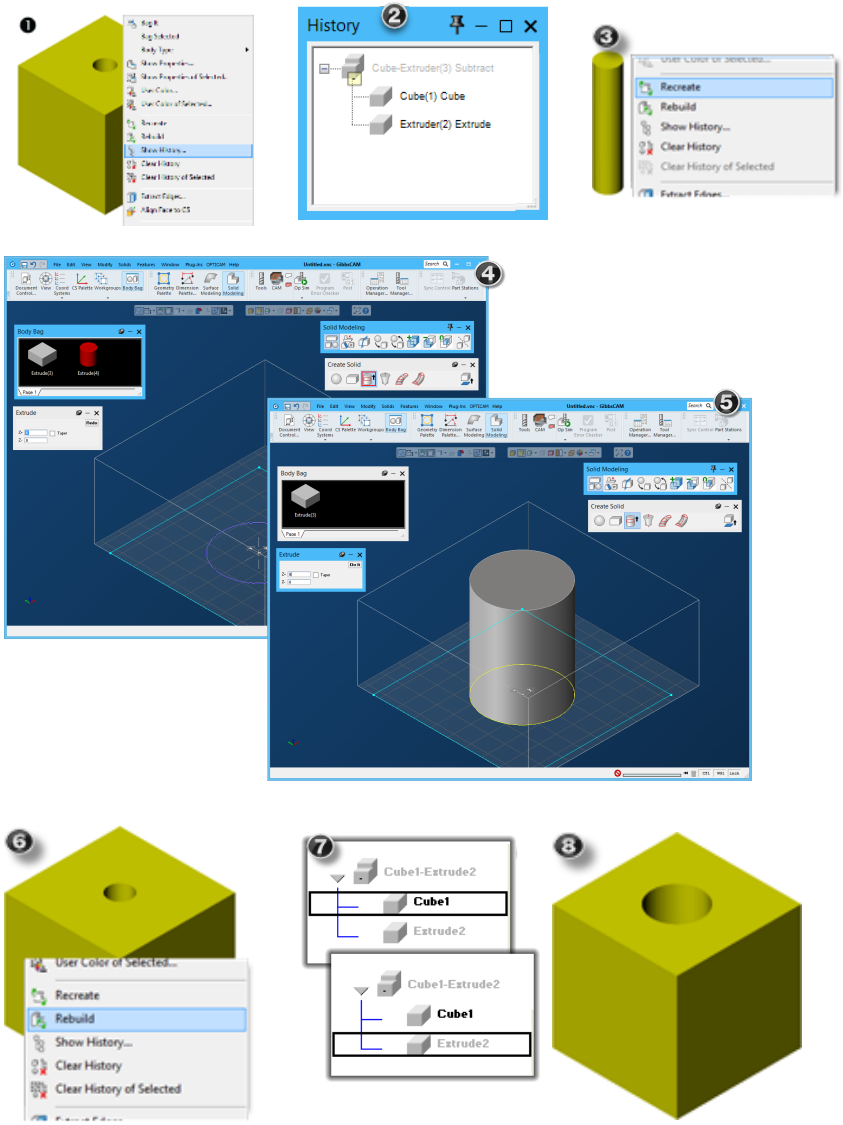


1. Original model needs a larger hole
2. New extrusion
3. Original extrusion is extracted from History list
4. History is Replaced by the new Extrusion
5. The model is Rebuilt
6. The new model

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.

Machining

Introduction to 2.5D Solids Machining

This chapter contains reference information on the multi-surface machining functions in the system. This first section explains some of the terms and concepts that the user will need to know in order to utilize the multi-surface machining capabilities. Refer to the Mill manual for information on the standard machining functions contained in the Production Milling module.

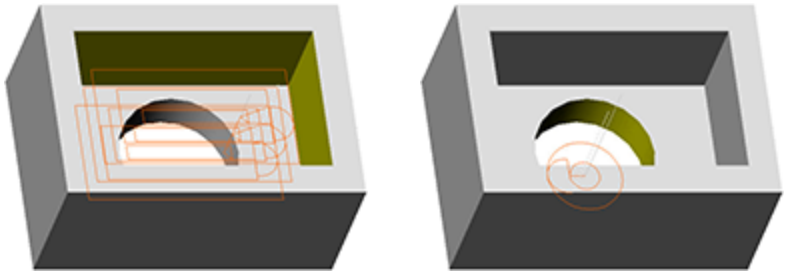
2.5D Machining Details

Contour and Roughing are the primary toolpath processes for the 2.5D Solids option. Contour and Roughing can be used in several ways. Both can cut selected geometry shapes. Both can project these toolpaths on top of selected bodies. Both can machine Profiler loops (see Using the Profiler) and edge selections, with or without Projection. Both can directly machine selected faces.

Contour's primary solid machining function is to machine selected faces, with a constant Z toolpath. This is easy to visualize as slicing the selected faces, at various Z depths. Unselected faces are not machined.

Roughing/Pocketing is a 2D area clearance function. Its primary solid machining function is to remove all the material in the region represented by the selected faces. This region concept is especially important when cutting at Zs above the part body, or above the selected faces. The tool will cut on centerline on a region boundary above the model. In the model, the faces control the toolpath.

Large through holes and empty areas (see the image below) make the selection and machining of their regions difficult. You cannot select a face that is not there. You can always make a process model, one without the hole, if you so desire. Or you can use the advanced short cut of selecting the walls of the hole. Selected walls on the edge of a region, will extend the region where ever possible. In general, select faces to machine more, deselect faces to machine less.



Picture a rectangular pocket, 35mm deep in a part whose total depth is 50mm. It has a 50mm diameter through hole in its floor. For the first pocket, the walls of the pocket are selected to extend the "clear" region across the hole. When pocketing down to Z-35 we will have a simple rectangular toolpath, as the through hole's region is part of the region that was selected to clear. For the second operation the wall of the through hole is selected so that when pocketing down to Z-50 we do not waste time finishing the floor across the top of the through hole. Do not confuse this "through hole" in the finish part model, with where there is, or is not material.

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

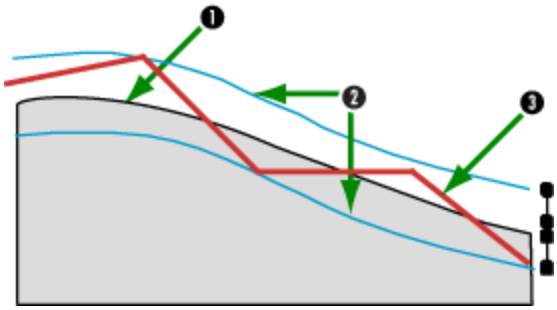
Data imported from earlier versions of the system does not have all of the functionality of the Gen 3 engine and the new features. To accommodate this, the process is rebuilt with whatever data can be gathered from the old process and the new options use default values.

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 (Fixture)
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. These methods are the workspace stock in the Document Control dialog, a workgroup that is defined as stock and a solid that is defined as stock. Stock definitions have different effects on different operations. The primary effects are to extend or reduce the 2D area being machined. They can also reduce the area being machined in Z.

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 2.5D Solids 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.

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 will track 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 will generate 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 dialog box 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 will perform the necessary calculations for a Material Only operation even if the calculations will not be applied; this information will also be saved with the part file.

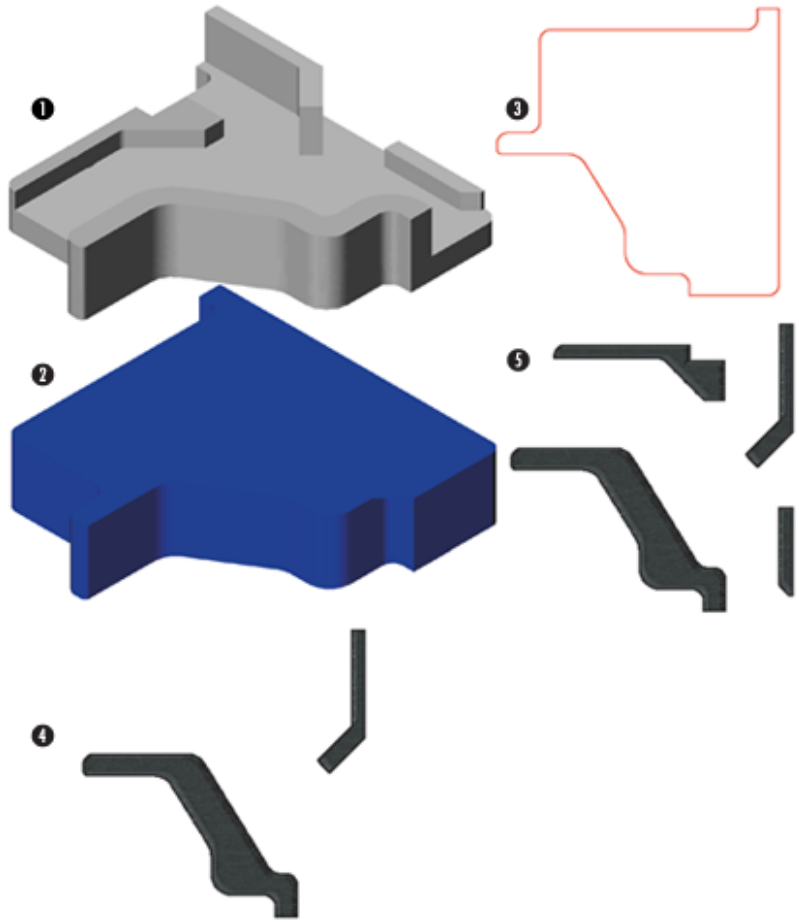
Material Only Pockets

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. More information on Material Only and cutting geometry can be found in 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, 2.5D Solids 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.



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

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.

Optimizing Material Only for Solids

- Avoid full solid selections. Only select the area (faces) to be cut.
- Use 2.5D toolpath optimization. This will produce better toolpath (not just G1s but G2s and G3s) and will also allow for a tighter Surface Tolerance setting. Avoid undercuts when using the 2.5D toolpath optimization 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.

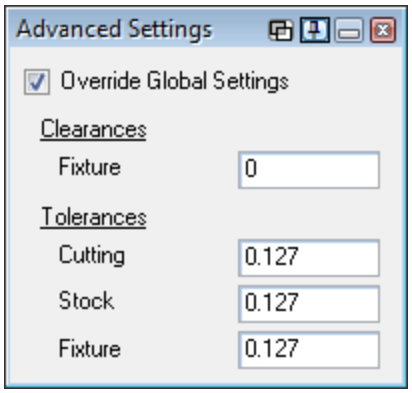
Cutting Direction

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



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 overriden.

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.

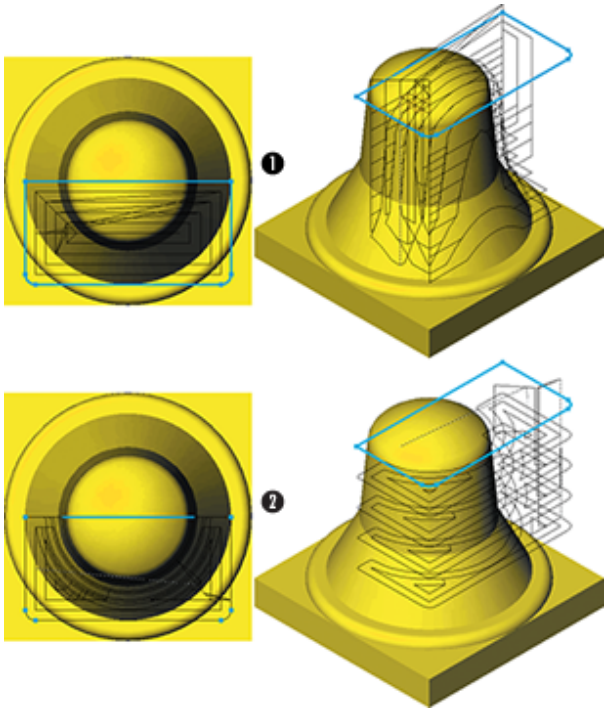
There is a text box for the clearance value from a Fixture. This value is the additional distance the toolpath will be offset from the object.

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 below 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. The user must specify a tolerance for the constraint faces as well as the Create 2D Toolpath settings (described in the section below) 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 wish 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.

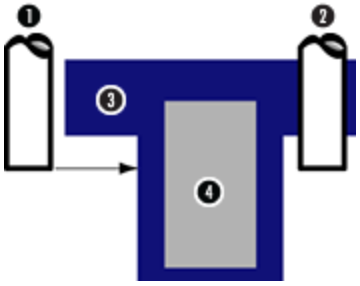
A 2D toolpath contains lines and arcs and does not vary with surface tolerance. A 3D toolpath is usually a large number of small line moves that vary from the true surfaces by the surface tolerance. A 3D toolpath is created when solids and surfaces are being machined.

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. They are documented below and are noted as they apply to each option.

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 moves.

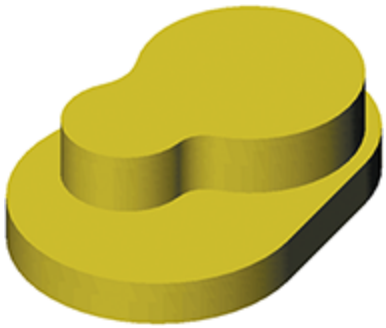


1. Clear Entry with undercut protection
2. Clear Entry without undercut warning
3. Stock
4. Part

Undercut protection in a T-shaped part

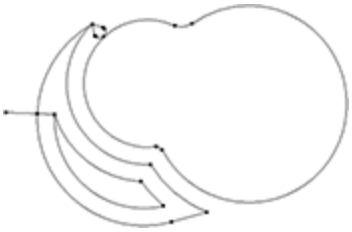
Part Body

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.



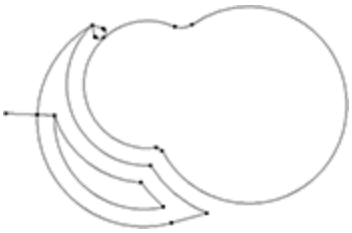
Model that is ideal for 2D toolpath	Standard 3D Toolpath

From 2D Body



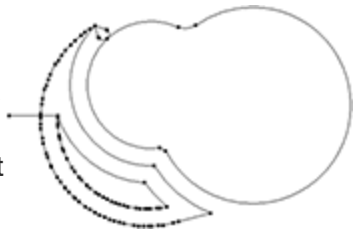
This option will generate 2D toolpath (lines and circles) without the surface tolerance deviation, provided that all of the faces selected are 2D. It makes high-quality 2D toolpath very quickly. Note that a horizontal chamfer or fillet is not 2D.
In order for the From 2D Body option to work, either the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

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

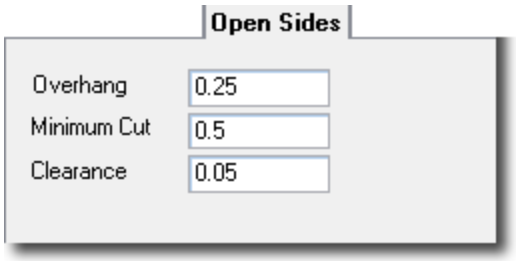
Constraint Face Protection

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.

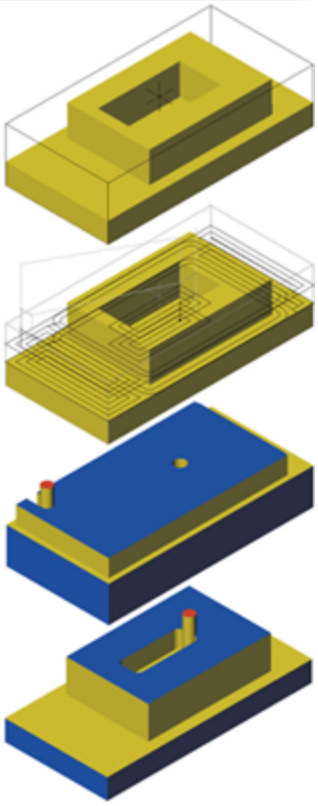
Fixture Protection

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.
Despite these limitations (listed on the following pages with the appropriate function), there are many parts that do not need this protection, and the advantages of a 2D toolpath for prismatic solids are significant.

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 pocketing 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 pocketing 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.

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.
- 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 or Contouring 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.

Appendix

Glossary

2D Solid Model	Also referred to as a prismatic body. A 2D solid is an XY shape extruded in Z, for example a circle produces a cylinder. All Z slices produce the same shape, and the slice of a cylinder face produces a circle segment.
2.5D Solid Model	A solid that has all 2D or 2.5D analytic faces and can be cut with a series of 2D toolpaths (of varying shapes) at different Zs, producing analytic toolpath feature output from underlying analytic model faces.
3D Solid Model	A term used for any model that exceeds the 2.5D solid model definition including parts with variable radii and fillets at an angle.
Alignment Points	Alignment points are also referred to as sync points. The selection of alignment points is used with the lofting and sweeping modeling functions. 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	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, while a sheet body is more like a balloon with an infinitely thin wall.
Body Bag	The Body Bag is used as a storage place for bodies 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 Commands palette. Bodies

	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. Bodies 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 bodies 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 tree. 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 bodies 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 > Graphics dialog. Different chord heights can be applied to individual bodies 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 tangencies, 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 line or curve between two adjoining surfaces. An edge of a solid must have exactly two faces connected to it. 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 bodies or

	sheets using the Edge Selection button located in the Floating Taskbar. 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. A Sheet face includes the positive and negative sides while a solid only includes the positive side. 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. A simple face is bounded by one loop.
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 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. 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	A surface is either a face group of faces (depending on how the surface was created) of a solid or side of a sheet. Sheets have two surfaces.
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.
Lynch 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. Bodies 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.

Links to Online Resources

Please contact your reseller for support.

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

Index

#

1 Pass Stitch 41

2.5D Solid Model 8

2.5D Solids product 6

2.5D Toolpath 8

2D Chain 26

2D Curve 49

2D Normal Base Curve 49-50

2D on Top, Replace on Bottom, part body option 81

2D Solid 6, 8

2D Toolpath 8, 78, 82-83

3D Chain 26-27

3D Curve 49

3D Faces, Body Select context item 25

3D Normal Base Curve 50

3D Toolpath 78, 81

A

Add
- button 43, 58

Adding
- sheets 43, 58
- solids 43, 58

Addition
- boolean operation 58

Addition Boolean operation 35

Addition function 35, 58

Advanced Settings 30, 77
- Clearances 77
- Tolerances 77

Advanced Solid Modeling
- palette 12

Advanced Solid Modeling button 51

Advanced Solid Modeling palette 43

Advanced Solid Modeling Palette 51

Air Walls 82

Align Edge to CS H 26

Align Face To CS
- Body context menu 24

Align to Grid 18

aligning solids 26

Alignment
- DCP 48

Alignment Points 47
- definition of 84
- drive curve 49

All Sides by Approximately, Untrim & Extend 43

All Sides Outside Cuboid, Untrim & Extend 43

Allow Mill Material Only 74

Analytic Geometry 6

Analytics, definition of 84

Arranging items, Body Bag 17

Atomic Body 13, 24, 32-33, 44, 57, 62
- definition of 84

Auto Arrange 18

Axis of Revolution, revolve solid 46

B

Bag It 16
- Body context item 23

Bag Selected 16, 18, 23

Base Curve 48

Base curve
- swept shapes 49

Blend edges, tip 68

Blending
- button 51
- see also Lofting 47
- solid edges 54

Blends
- remove 55

Blue
- stock 33

Bodies
- collection 33
- comments 14
- cuboid 44
- history 62-63
- intersecting 59
- modifying 65
- multi-lump 60, 62
- naming 14
- rebuilding 68
- rectangular 44
- slicing 57
- spherical 44
- subtracting 58
- swapping 57

Body 12
- atomic 32, 62
- Definition of 8, 84
- history 66
- lump 62
- multi-lump 33
- names 68
- replacing 57
- substituting 57
- yellow 33

Body Bag 13, 18, 34
- arranging items 17
- button 12
- clean-up 19
- definition of 84
- detailed list 17
- icons 17
- List 17
- pages 19
- selecting bodies 20
- Surface file imports 68
- Tiles 17

Body Bag context menu
- Bag Selected 20
- Deselect Page 20
- Select Page 20
- Show Properties of Selected 20
- User Color of Selected 20

Body Bag items
- deselecting 68

Body Bag objects
- colors 19

Body Bag page
- context menu 19

Body Bag pages
- adding 19
- arranging objects 19
- clean-up 19
- deleting 19
- inserting 19
- moving objects 19
- renaming 19
- view settings 19

Body Type 23

Body Validity, Check 21

Boolean
- subtraction 64

Boolean Operations 13, 35
- addition 35
- definition of 85
- intersection 35
- new solid 64
- replace 35
- separation 35
- subtraction 35
- swap 35

Boolean operations
- addition 58
- destructive 35
- non-destructive 35

Bottle volume
- calculate 55

Boundary 25

B-pointer marker 49

Buttons
- Add 43, 58
- Advanced Solid Modeling 51

Blending 51
Intersect 43
Offset 52
Part 71
Replace 43, 57
separate 43
Separate 60
Shell 52
Shell/Offset 51
Slice 43, 57
Solid Modeling 43
Subtract 43
Subtraction 58
Swap 43, 57
Unstitch Solid 51

C

Calculate bottle volume 55

Calculate volume 55

Cap, solidify sheet 50

Chamfers
remove 55

Chord Height 15-16, 29-30
definition of 85
global setting 30

Chord height
faceting 30

Clean Up Body Bag 18

Clear History 24

Clearance, open sides 82

Climb Cut 76

Closed Shape 49

Closed Surfaces
Solidify 51

Co-edge Modeling 67

Coincident, definition of 85

Collapse All (History) 27

Colors
blue 33
Body Bag objects 19
red 33
solids 33
yellow 33

Comments
bodies 14

Congruent Face Modeling 67

Constant Chamfer, solid 54

Constant Radius, solid 54

Constraint button 71

Constraint face 71

Constraint Face Protection 81

Constraint Faces
Clearance 78
Tolerance 78

Constraints, body machining 71

Container
volume 55

Continuity, definition of 85

Contour
Profiler 73

Contour Operations 73

Contour Process 72, 76

Contouring operations
fixtures 72

Contouring Process
Dialog 76

Conventional Cut 76

Coons Patch 39, 41

Coordinate Systems 34

Corner Cleanup 83

Corners
blending 68
mitred 49
sharp 49

Create 2D Toolpath 78, 82
Limitations 81

Create Plug, unstitch 55

Create Solid palette 33, 43-44

Create Solild palette 12

Creation method
solilds 27

Cubes
creating 44

Cuboid bodies 44

Cuboid dialog 44

Cuboid, solid 33, 44

current coordinate system
- slicing with 57

Cut Part Rendering
- Stock 72

Cut Shape 71

Cutting Direction 76

Cylinder
- new 64

D

DCP (Drive Curve Plane) Alignment 50

DCP Alignment 48

Deselect 21
- Body Bag 18
- Tangent Faces 25
- Wall Faces 25
- Workspace 18

Desired Z Step 78

Destructive process 35

Detail 18

Diagnosing problems in solids 21

Dialogs
- Loft 47
- Solid Revolve 46
- Solidify 50
- Sphere 44

Difference, see Subtraction 35

Disjunct, definition of 85

Display preferences 29
- setting 28
- viewing 28

Dividing
- multi-lump bodies 60
- multi-lump sheets 60

Document Control dialog 72

Dormant Bodies in History 24

Draft
- topology 53

Drive Curve
- Plane Alignment 48

Drive curve
- alignment points 49
- open terminated shapes 49
- swept shapes 49

Drive curves
- swept solids 48

E

Edge 10, 41
- definition of 8, 85

Edge Drawing 29

Edge Loop 26, 40, 42
- definition of 86

Edge Selection 41
- 2D & 3D 26

Edge Tolerance 41

Edges
- selecting 10

Edit
- solid faces 65
- solid locally 65

Edit menu
- Deselect 20
- Invert Selection 20
- Select All 20

EDM electrodes 55

Electrodes
- EDM 55

Engine
- toolpath 70

Enlarge
- face 52
- solid 52

Enlarge Solid 21

Expand All (History) 27

Extend Sheet 43

Extract Edges 26

Extract Edges plug-in 26

Extract Edges, using Edge context menu 26

Extract Profile 28

Extract profile as geometry 73

Extrude 34
- Solid 45
- Solidify Sheet 50

Extrude dialog 45

Extrude Sheet 38

Extruding shapes 45

Extrusion
- new 64

Extrusions
- tapered 46

F

Face 10
- Check 42
- Definition of 8, 86
- enlarge 52
- shrink 52
- Validity, Check 21

Face Selection 25, 40

Faces
- edit 65

Faces Above, Body Select context item 25

Faces Below, Body Select context item 25

Facet Drawing 29

Faceting
- chord height 30

Faceting Tolerance 29-30

Facets 29-30

File imports
- Body Bag 68

Fillets
- remove 55

Fillets, Body Select context item 26

Fillets, creation tip 68

Finish Tolerance 76

Fixture
- Display Only 15
- red 33

Fixtures
- Contouring operations 72
- designating body as 15-16
- Protection 82
- Roughing operations 72

Flat Face 25

Floor Faces, Select
- Body context menu 25

Floor Z 74

Floor/Wall Angle Tolerance 25

From 2D Body, part body option 80

Functions
- Offset 52

G

G-code 76

Gen 2 Engine 21-22, 70, 78

Gen 3 Engine 70, 78

Geometric Modeling 32
- definition of 86

Geometry
- As Stock 72
- Boundary 78
- extract from profile 73
- Extraction 60
- From Solids 60

Geometry Extraction 61

Global Settings for Solids 76

Graphics Preferences 29

Grey
- rendered solid 33

H

Heal Only 56

Heal Solid 54

Healing Components 56

History 13, 24, 27, 33, 36, 66
- bodies 63, 66
- body 62

Characters 62
model 62
Names 63
symbols 62

History list 24, 62, 64
replacing bodies 57
solid name 65

History List 62

History tree 37

Hole Extraction 60-61

Holes
remove 54

Hollow container
volume 55

Hollow model 55

Hollow solid 53

I

Icons, Body Bag 17

IGES Files 40

Imported surface files
Body Bag 68

Indicate Sheet Side button 53

Internal Edges 41
Definition of 86

Intersect
button 43

Intersecting
bodies 59
sheets 43, 59
solids 43, 59
surfaces 68

Intersection Boolean operation 35

Intersection function 36, 59-60

Invert Selection, unstitch 56

L

Large Icons 18

List, Body Bag 17

Loft 34
Definition of 86
Sheet 39
Solid 47

Loft dialog 47

Loop
Definition of 8, 86

Lower boundary 25

Lump 33

Lump body 62

M

Machine Partially Selected Solids, Pocketing 74

Machining
Preferences 74
Selections (part, fixture, stock) 71
Tips 82

Machining Face Check 21

Machining Markers 28, 76

Machining Surface 70

Main palette 12

Material Only 74, 76, 79
3D operations 74
Multiple Shapes Method 75
Parameters 74
Tips for Solids 75

Minimum Cut, open sides 82

Mitred corners 49

Model
hollow 55

Modeling
co-edge 67
congruent faces 67
geometric 32
solid 32
surface 32

Modeling, definition of 86

Models
history 62

Modifying Bodies 63, 65-67

Mold
cavity 55
core 55
Multi-lump Bodies 35
Multi-lump bodies 36, 58, 62
definition of 86
dividing 60
icons and symbols 62
separating 60
Multi-lump body 33
multi-lump icons and symbols 62
Multi-lump sheets
dividing 60
separating 60
Multi-lump solids
separating 60
Multiple Loops, unstitch 56
Multiple Passes Stitch 22, 41
Multiple properties dialog 16
Multiple Tries Stitch 42

N

Name
solid 65
Naming Bodies 14
Naming bodies 68
Neighboring faces 25
New cylinder 64
New extrusion 64
Non-Atomic Bodies 13
Non-destructive Booleans 35, 68
NURBS 22

O

Object colors
Body Bag 19
Offset
Shelling amount 53
Solid 52-53
Solidify sheet 51
Solidifying 21
Offset button 52
Offset function 52, 65
Open Pocket Settings 74
Open Sides Tab 82
Open Terminated Shapes 46, 49
Open terminated shapes
drive curve 49
Open-Sided Pockets 82
Outline 62
Overhang 82
Override Global Settings 77

P

Page tab, Body Bag 19
Pages, Body Bag 19
Palettes
Advanced Solid Modeling 12, 43, 51
Create Solid 12, 43-44
Main (Top level) 12
Solid Modeling 12, 43
Surface Modeling palette 12
Top Level 43
Parametric, definition of 86
Parent Body 13
Part Body
Create 2D Toolpath option 79, 81
Part button 71
Part Stock 72
Part, body definition 15
Part, designating body as a 16
Parting Line 61
Physical Properties of a solid 15
Planar Sheet 38
Plane, create 38
Planes, see Coordinate Systems 34
Plug-Ins 22

Preferences
display 29
setting 28
viewing 28

Primitive Body, see Atomic body 33

Primitive Solids 32, 44

Prismatic Shapes 78-79

Profile
extract as geometry 73
extracting 28

Profiler 11, 73
Context Menu 27-28
Contour 73
Select All Profiles 28
Selection options 21

Profiler Depth 28

Project 2D Toolpath 77-78

Properties
Body context menu 23

Properties dialog 14-15, 30, 72

R

Radius
sphere 44

Rebuild Body 24, 36-37

Rebuild function 57, 64-67

Rebuilding
bodies 68

Recreate Body 24, 36-37

Recreate function 64, 66-67

Red
fixture 33

Red Body 24, 36

Reduce Number of Sheets 22

Reduce Size of Solid 21

Remove blends 55

Remove chamfers 55

Remove fillets 55

Remove holes 54

Remove Remaining Material, see Material Only 74

Remove Unneeded Topology 22

Render Shaded Objects 10

Render/Wireframe button 28-29

Rendered solid
grey 33

Rendering of bodies 29

Replace
button 43

Replace Boolean operation 35

Replace button 57

Replace Solid function 35, 57, 64-65

Replace TP with 2D Sections, part body option 81

Replacing
bodies 57
sheets 43
solids 43

Retangular bodies
creating 44

Revolve 34
Sheet 38
Solid 46-47

Revolving
horizontal axis 46
shapes 46
vertical axis 46

Ridge Height 78

Rough Tolerance 76

Roughing operations
fixtures 72

Roughing process 73, 76
Z step 74

Rounding Solid Edges 54

S

Scallop height, see Ridge Height 78

Select
All Profiles 21, 28

Body Bag 18
Faces From Selected Profiles 21
Tangent Faces 25
Wall Faces 25
Workspace 18

Select Faces From Selected Profiles, Profiler 28

Select Faces Inside Selected Profiles, Profiler 28

Selecting
By Body Comment 21
By Body Name 21
Edges 20
edges 26
Sheets 20
Solids 20
Walls From Selected Edges 21

Separate
button 43

Separate button 60

Separate function 36, 60

Separating
multi-lump bodies 60
multi-lump sheets 60
multi-lump solids 60
sheets 43
solids 43

Separation Boolean operation 35

Shape
sweep 48

Shapes
extruding 45
revolving 46

Sharp Corners 49

Sharp Swept Corners 49

Sheet 10
convert to solid 50
Definition of 8, 33, 87
from Face 40
Toggle Side 21

Sheet modeling 12

Sheet Side 10

Sheets 32
adding 43, 58
intersecting 43, 59
replacing 43
separating 43
slicing 43, 57
subtracting 43, 58
swapping 43
Transform to solids 44
trimming 59

Shell
wall thickness 53

Shell button 52

Shell, Solid 52

Shell, solid 53

Shell/Offset button 51

Show
Solids 10

Show Internal Edges 41

Show Properties of Selected 23-24

Show Solid Creation Method 18

Show Solid Creation Method (History) 27

Show Solid ID 18

Show Solid IDs (History) 27

Shrink
face 52
solid 52

Shrinkage 21

Simple solids
slicing 68

Simplify (NURBS Surface) 22

Skinning, see Lofting 47

Slice
button 43

Slice button 57

Slice function 57

Slice Offset Body 79
Part body option 80

Slicing
bodies 57
current coordinate system 57
sheets 43, 57
simple solids 68
solids 43, 57

Small Icons 18

Solid
creating 66
Definition of 8, 33, 87
enlarge 52
heal 54
hollow 53
modifying 66
rebuild 65
replace 65
selected 33
shell 53
shrink 52
swept 48
Unstitch 54
unstitching 55

Solid faces
editing 65

Solid ID
showing 27

Solid Modeling 32, 43
advanced 12
button 43
Definition of 87

Solid Modeling palette 12

Solid Modeling Palette 43

Solid Models 6

Solid name
History list 65

Solid object 32

Solid Revolve dialog 46

Solid Unstitch 65
Options 55

Solidify 34
Closed Sheets 50
Closed Surfaces 51
Sheets 50-51

Solidify dialog 50

Solidify sheet
cap 50
Offset 51

Solidify Sheet
Extrude 50

Solidify sheets 50

Solidify Sheets 51

Solids
adding 43, 58
colors 33
combining 63
creation method 27
edit locally 65
intersecting 43, 59
modifying 65
primitive 32
replacing 43
separating 43
slicing 43, 57
subtracting 43, 58
swapping 43, 57
swept 48
switching 57
toolpath 70
trimming 59
viewing 33

Solids Button 12

Solids tab 76, 82

Solids toolpath engine
Gen 3 70

Sphere
radius 44

Sphere dialog 44

Sphere, solid 33, 44

Spherical bodies 44

Splines 39

Stitch
Multiple Passes 41
Multiple Tries 42

Stitch Sheet 41-42

Stitch Utilities 21-22

Stock 71
blue 33
button 71
Defining 71
Designating body as 15-16
designating body as 15
Display Only 15
Hierarchy 83
Ignoring 74
Local 71
Multi-lump Body 72
size 72
Temporary 71-72

tolerance 72
Toolpath confined to 74
Workgroup 72

Stock Body 72
Create 2D Toolpath option 79

Stock Shape 71

Substituting
bodies 57

Subtract
button 43

Subtracting
bodies 58
sheets 43, 58
solids 43, 58

Subtraction
Boolean 64

Subtraction Boolean operation 35

Subtraction button 58

Subtraction function 35, 58

Surface
Definition of 8, 87
machining 70
swept 48

Surface Area, calculating 15

Surface Entities 7

Surface file imports
Body Bag 68

Surface Machining Tolerance 30

Surface Modeling 32
Palette 37

Surface Modeling palette 12, 37

Surface Normals 53

Surface Stock 70, 72, 78

Surface Stock Allowance 72

Surface Tolerance 70-71

Surface Trim
Definition of 87

Surface Z 74

Surfaces
intersecting 68

Surfaces button 12

Surfacing Process 83

Swap
button 43

Swap Boolean operation 35

Swap button 57

Swap function 35, 57

Swap Solid 64

Swapping
bodies 57
sheets 43
solids 43, 57

Sweep 34
shape 48
Sheet 40
Solid 48, 50

Sweeping Plane 50

Swept shapes
base curve 49
drive curve 49

Swept solid 48

Swept solids
creating 48
drive curves 48

Swept surface 48

Switching
solids 57

Synchronization Points 47
See also Alignment Points 87

T

Tab, Body Bag page 19

Tangent Faces
Body Select context item 25

Tapered Extrusion sheet 38

Tapered Extrusion solid 46

Target Face 25
Definition of 87

Taskbar 10

Temporary Stock 72

Tiles 18

Tiles, Body Bag 17

To A Point, Untrim & Extend 43

Tolerance
- Solids Tab setting 76
- surface 70
- toolpath 70

Tool, moving past stock 74

Toolpath
- Deviation 70
- Not cutting entire part, only the stock 74
- solids 70
- tolerance 70

Toolpath engine
- gen 3 70

Toolpath, Trim 77

Tools menu 21

Top Level palette 12, 43

Topology
- Definition of 87
- draft 53

Transition Faces, Body Select context item 25

Trim Surface 40

Trimmed Surface Edges, Check 22

Trimmed Surface Polyline, Check 22

Trimming
- sheets 59
- solids 59

U

Un-Bag It 16
- Body context item 23, 68

Un-Bag Selected 16, 18

Undercut Faces 83

Undercut protection 79, 81

Union, see Addition 35

Unstitch
- Components 56
- Solid 54, 56
- Surfaces 42

Unstitch Solid button 51

Unstitching solid 55

Untrim & Extend Surfaces 43

Untrim Surface 40, 43

Upper boundary 25

Use Cap, unstitch 56

Use Global Settings for Solids 30

Use Stock 74, 83

V

Variable Radius Rounding 54

Vertex 26
- Definition of 8, 87

View 18

View items, Body Bag 17

Viewing solids
- rendered solid 33
- wireframe 33

Volume
- calculate 55

Volume, calculating 15

W

Wall Faces, Body Select context item 25

Wall thickness
- shell 53

Wire Drawing 29

Wireframe View 10

Workgroups 34

Workspace 34-35, 57
- As Stock 72
- Definition of 87

Y

Yellow body 33

Z

Z Step 78