GIBBSCAM | 2027 CAM for Production Machining

Version 2027 : September 2026

Common Reference

GIBBSCAM

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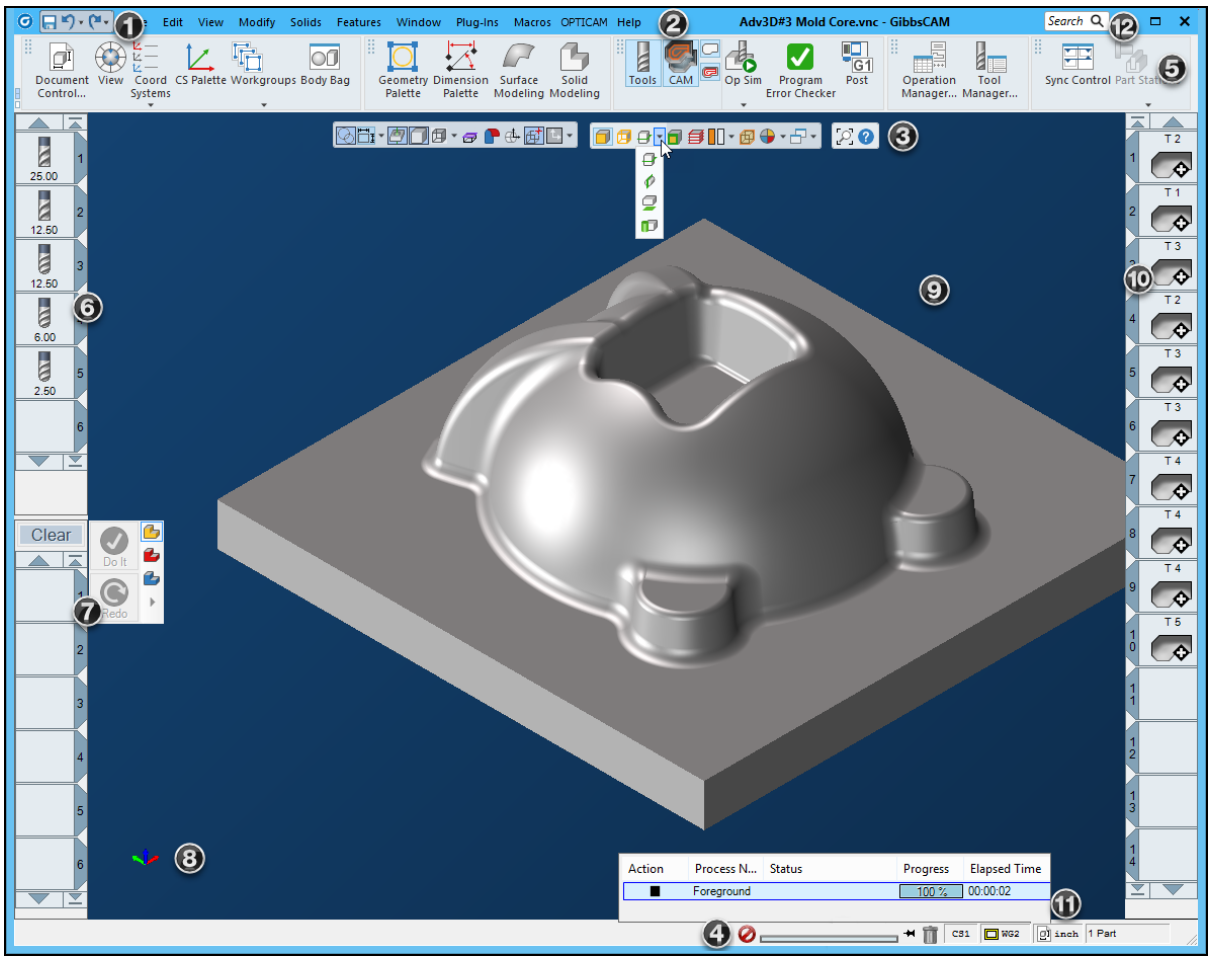
Introduction

Welcome to the GibbsCAM *Common Reference* guide. GibbsCAM is a modular application, meaning that different customers will have different functions depending on their need. In other words, not everybody needs Mill/Turn functionality or the SolidSurfacer module.
But much of GibbsCAMs interface and functionality is shared or is common across the many modules. This guide documents most of the common interface items found throughout GibbsCAM. Some options or third-party packages create their own Menu Bar entry. These packages are not covered in this guide. Intermediate Tooling, Reporter, and Plug-Ins are all covered in separate guides.
It is recommended that you also read the *Getting Started* guide in conjunction with this guide.
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.

Main Screen Components



1. Quick Launch Bar
2. Main Menu
3. Floating Toolbar
4. Status Bar
5. Commands toolbar
6. Tool List
7. Process List
8. Axis Block
9. Workspace
10. Operations List
11. Trash
12. Command Search

Main Menu

The Main Menu is available to the user at all times. The Menu provides access to many of the system's functions through individual sub-menus.

Some items might or might not appear depending on the current license, installed options, and machine type. For example, GibbsCAM Viewer offers a subset of capabilities provided by GibbsCAM Industrial Edition.

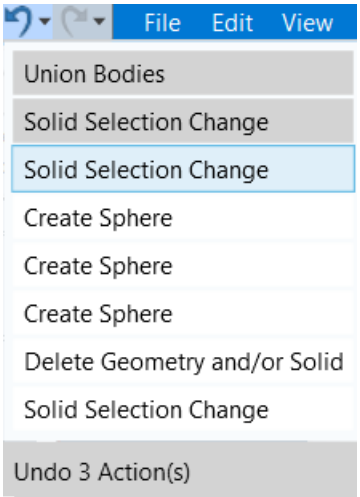
1. Quick Access Toolbar
2. File Menu
3. The Edit Menu
4. The View Menu
5. The Modify Menu
6. The Solids Menu
7. The Features Menu
8. The Window Menu
9. The Plug-Ins Menu
10. The Macros Menu
11. OPTICAM Menu
12. The Help Menu
13. Current filename display
14. Command search.

Quick Access Toolbar

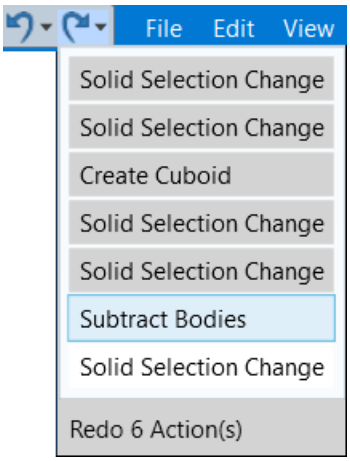
By default, the Quick Access Toolbar contains only the **Undo**, **Redo**, and **Save** commands, but it can be customized and extended if required: See Customizing the UI.htm

In the Quick Access toolbar (, just to the left of the **File/Edit/View**/... main menu, the **Undo** and **Redo** commands are pulldown menus. Each pulldown provides a list of all the items that can be undone or redone. Dragging the cursor down the list provides a count of the number of items that will be undone or redone if a selection is made.

This function greatly speeds and facilitates navigating to a particular action instead of having to press `Ctrl+Z` or `Ctrl+Y` repeatedly to undo or redo many actions.



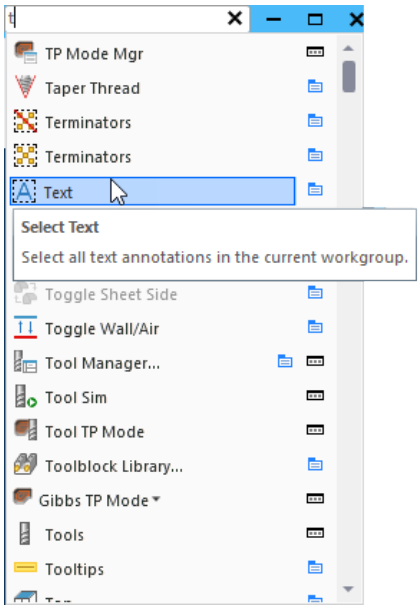
Undo pulldown menu (sample)



Redo pulldown menu (sample

Command Search

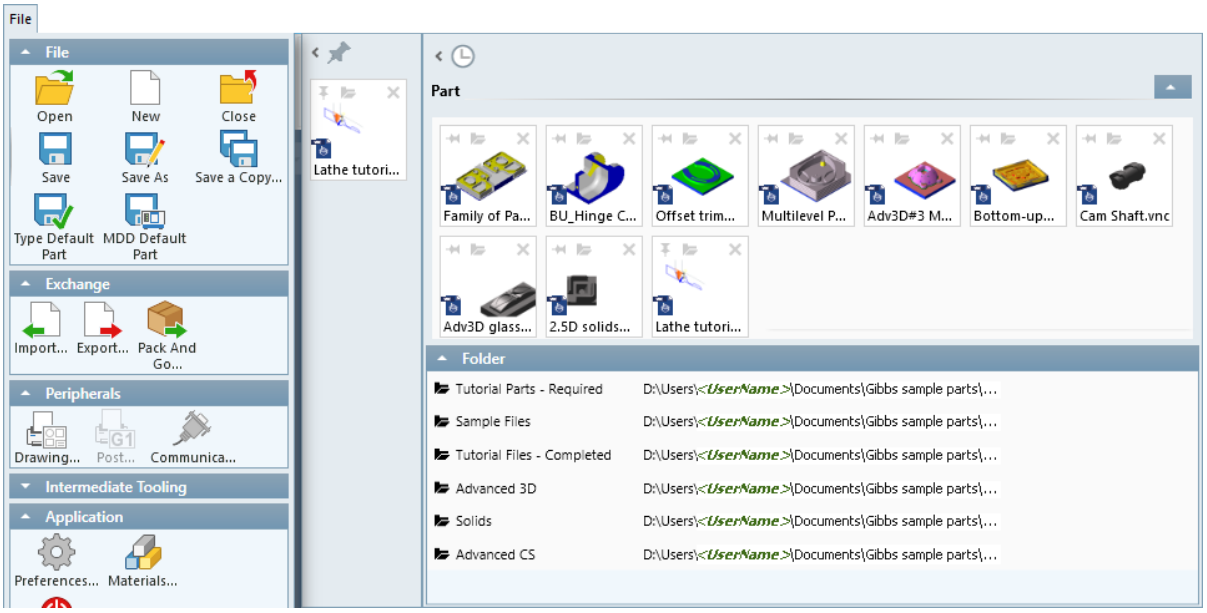
Command Search lets you find a command just by typing in a portion of its name.



You can hover your cursor over a list entry to see tooltips (illustrated left). Click to execute the command. Click either the main or sub-menu icon to highlight the command in the UI, (it flashes yellow) but not execute it.

If a menu item is grayed out, it cannot be selected. Depending on the item, selecting geometry or some other appropriate item on the screen can change the menu item so that it becomes black and can be selected.

File Menu



The system will keep track of the parts that have been most recently used and will display information on folders. Moving the cursor over a recently opened file displays a larger preview of the part.

Clicking the pin icon on the part view saves the part file onto the pinboard. This is a scrollable expanding area which enables parts to be grouped together and will allow more than the most recent parts to be saved. Use the large pin icon at the top of the pinboard to hide/unhide the pins. All the last opened part information and pinboard is saved for your next session when you exit GibbsCAM.

In addition to the standard Windows file controls, there are additional options as detailed below:

File Items

Open

Displays the Open dialog. Change to the filetype required, or choose All Files (*.*). Browse to the file you want to open and click Open. If a file is currently open, it is closed and the file you select opens.

Save a Copy

This will save the current file under a different name or version and let you continue working with the original file. “Save as type” pull-down menu provides an ability to save the file into older versions of GibbsCAM.

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

Type Default Part

You can designate that a particular part file is the default part associated either with one of two machining types (Mill or Turning) or with an MDD (Machine Definition Document):

- To save the part as the default for its type, choose File > Type Default Part.
- To save the part as the default for its MDD, choose File > MDD Default Part.

Result: The part is saved in a special folder (the Default Parts Folder) with a special filename, such as (for Type Default Part) `New Mill Part.vnc`, or (for MDD Default Part) `New 3 Axis Vertical Mill.vnc`. The name and location of the Default Parts Folder are available through Pathfinder — typically `C:\Users\<username>\AppData\Roaming\Gibbs\GibbsCAM\<version>`**DefaultParts**\. When a new part is created, the MDD Default Part will be copied in if it exists for this MDD; otherwise, the Type Default Part will be copied in; or, if there is no default part set for either the MDD or the machining type, then the new part will be blank.

Definition:
In GibbsCAM terminology, the MDD (Machine Definition Document) is where all aspects of a particular machine are organized and stored, including its linear and rotary axes, its toolgroups, spindles (part stations), and utility stations, and how these are associated and organized into Flow Axis Sets, Interop Moves, etc. It also specifies the post processors, coolants, extended cycles, and simulation bodies available to the machine, and it records preferences for work areas, limits, clearances, and many other items.

Exchange

Import/Export

The Import and Export dialogs allow a variety of file types to be exchanged into and from the current file. Each of the import types can be directly opened through the File > Open command. For specific information on how each file will be interpreted and translated, see the *Data Exchange* guide.

Pack and Go

You can easily create and use package files (`*.gcpkg`) containing a GibbsCAM part and the transportable portion of the environment that the part requires or expects.

Please Note: Be aware of proprietary considerations before you share parts and environments with others. For example, you might have a part file that is not proprietary, but its associated post, or Machine Sim part files, might be subject to a license or nondisclosure agreement.

To create a package: On the main menu, File > Pack and Go.

The following are all included in the package:

- part file (always)
- MDD (always)
- VMM (always, if present)
- Templates for Basic Utility Operations (always, if present)
- Machine Simulation files (optionally, if present)
- Intermediate Tooling files (optionally, if present)
- Postprocessor file (optionally, if present)
- Macros for custom drill cycles (always, if present)
- system diagnostic file

To use a package: Simply drag the `*.gcpkg` file into GibbsCAM. A temporary "sandbox" environment will be created that will be automatically deleted when you close the part.

Peripherals

Print Drawing

Selecting Drawing... (`Ctrl+P`) will print the geometry, toolpath, and the rendered image as they currently appear on the screen. The exact output will vary depending on the settings in the Printing preferences; see " Display Preferences " on page 16.

Post

Post... becomes active when a file has been post processed. Posted output can also be printed from the Post Processor dialog.

Communication

Communication dialog allows the user to specify how to send and receive text or VNC files to and from the machine control. The Communication dialog can also be accessed from the Post Processing dialog. For more information on the use of this dialog and Post Processing see the *Getting Started* guide and the guides for *Mill* or *Turning*.

Intermediate Tooling

UKM provides the ability to support a Toolblock Library (a generic term encompassing adapter blocks, holders for toolholders, right-angle heads and adjustable heads, etc.) and a Fixture Library (chucks, tailstocks, steadyrests, etc.). For a part setup, you can see locations and orientations of toolblocks and adjust their offsets, and you can control the display of tools and toolblocks in the toolgroup. For more information, see " Intermediate Tooling Preferences " on page 57. You can also change the machine configuration using Machine Data.

Application

Preferences

Preferences opens a dialog that allows you to set all the preferences and behaviors for GibbsCAM. This includes communications with a CNC, establishing default file extensions, and details on how to save files and set up the display output, as well as interaction with the mouse or digitizer. All the preference settings are described in " Preferences" on page 15.

Materials

This choice opens the Materials dialog which allows you to view, add and modify material properties and cutting speeds. For a full description of this function, see " Materials" on page 60.

Exit

This closes GibbsCAM.

Preferences

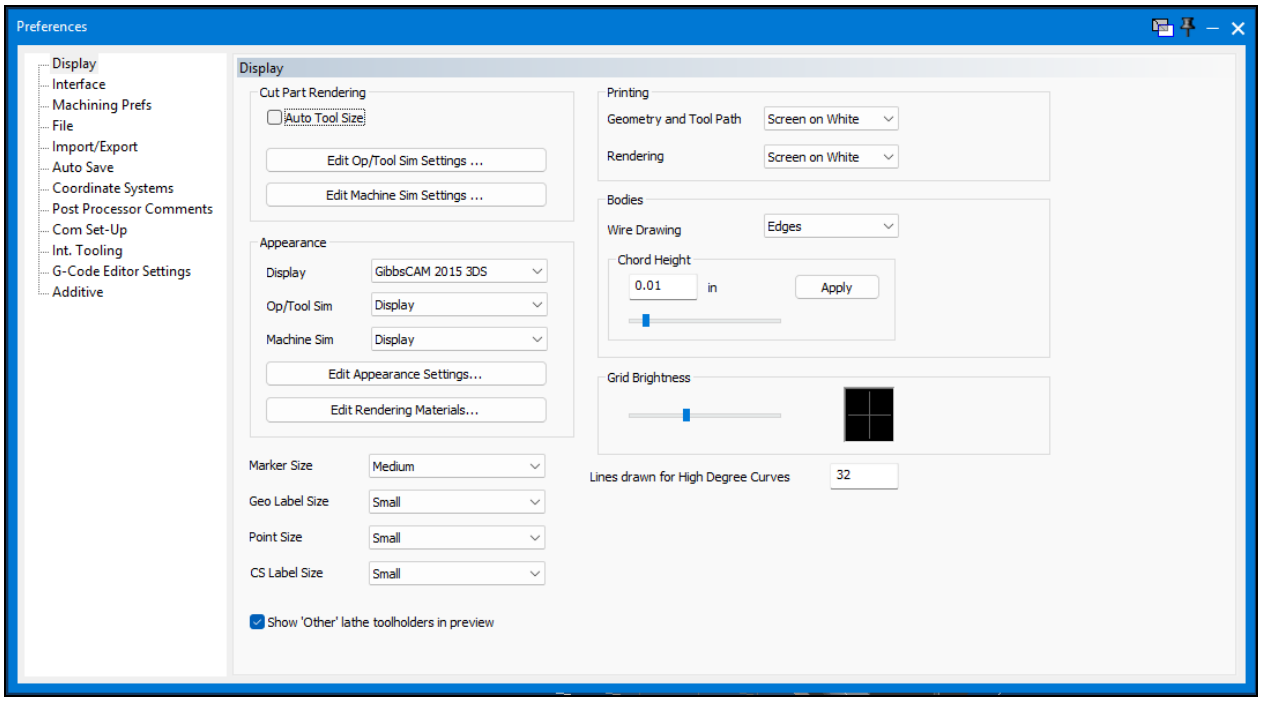
The **Preferences** dialog presents tabs containing controls that affect different parts of the system. The items in the **Preferences** dialog for GibbsCAM Viewer are a subset of those in the dialog for GibbsCAM Industrial Edition, as shown below.

1. Display Preferences , next.
   Available in Viewer and Industrial.
2. "Interface Preferences " on page 35.
   Available in Viewer and Industrial.
3. " Machining Preferences" on page 43.
   Available in Viewer and Industrial.
4. " File Preferences" on page 47.
   Available in Viewer and Industrial.
5. " Import/Export Preferences" on page 49.
   Available in Viewer and Industrial.
6. " Auto Save Preferences " on page 50.

Available in Industrial only.

7. “Coordinate Systems Preferences” on page 50.
   Available in Viewer and Industrial.
8. “ Post Processor Comments ” on page 52.
   Available in Industrial only.
9. “ Communication Setup Preferences” on page 55.
   Available in Industrial only.
10. “ Intermediate Tooling Preferences ” on page 57.
    Available in Viewer and Industrial.
11. “ G-Code Editor Settings ” on page 58.
    Available in Viewer and Industrial.

Display Preferences



The Display preferences affect what you see on screen or what you print out.

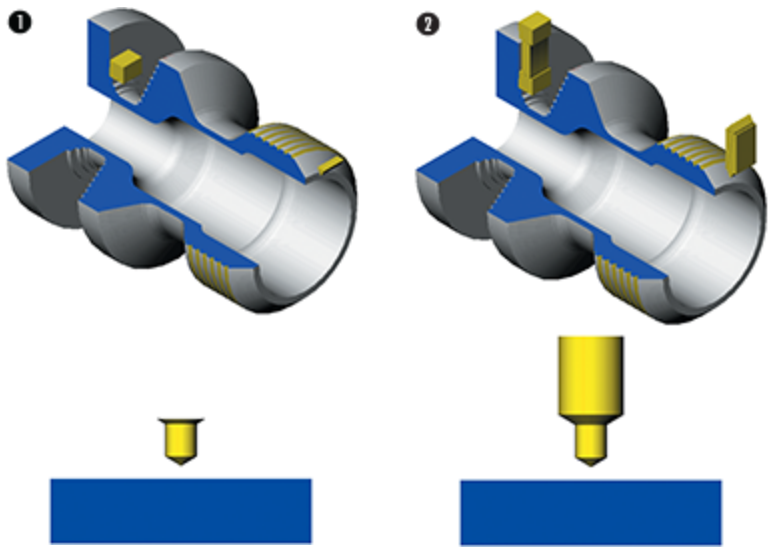
Cut Part Rendering

The Cut Part Rendering section specifies preferences for the display when a part is being virtually machined, including how tools are displayed.

Auto Tool Size

This modifies the length of some tools to maximize the visibility of Cut Part Rendering. Very long tools are shortened and short tools are lengthened to just above the stock to ensure visibility. Thus, a short tool will not be hidden inside a pocket when being rendered. When Auto

Tool Size is disabled the system will display tools as they actually look. This is particularly useful for Turning threading and grooving operations and some milling operations. Activate Auto Tool Size if you have an older computer or are not concerned with tool length, as this will increase Cut Part Rendering speed.



1.Enabled
2.Disabled

Auto Tool Size enabled and disabled examples.

Edit Op/Tool and Machine Sim Settings

Clicking this button opens the Settings dialog that allows you to define how Part and Tool Rendering should look and act. This includes the quality and appearance of the rendering and how collisions should be handled. For more information, see Settings for Op Sim, Tool Sim, and Machine Sim

Appearance

The Appearance settings directly affect the look of the GibbsCAM workspace. Beyond just the "look" of the workspace, these settings can directly affect the system's performance. There are three categories which can each have different settings.

Display

The Display category affects the immediate workspace and solids.

Op/Tool and Machine Sim

This affects how simulation will look and behave.

You can choose from several preset themes, including GibbsCAM 2005 No OpenGL, GibbsCAM 2006, GibbsCAM 2005 OpenGL, and many others. You can have different settings for Op/Tool and Machine Simulation. Or, you can set Op/Tool or Machine Simulation to use the same settings as Display by choosing "Display" from the drop-down list.

GibbsCAM 2005 No Open GL uses the legacy GibbsCAM cut part rendering and behavior. Older systems may work best under this setting. Note that Op Sim and Tool Sim still work when using this option, but it closely resembles the legacy ("CPR") rendering. GibbsCAM 2005 OpenGL has the legacy look but uses OpenGL to accelerate solids and Op/Tool Sim. GibbsCAM 2015 3DS is

the standard display. GibbsCAM 2006 CR and GibbsCAM 2006 SW are variations on the GibbsCAM 2006 appearance.

Edit Appearance Settings...

Clicking this button opens a dialog that lets you create your own appearance setting and choose options. This can help speed up the system display.

Appearance Settings Dialog

Configurations

This list shows the available presets. A red arrow is next to the preset the display is currently set. These presets cannot be changed but you may choose one, duplicate it and modify the duplicate. It is recommended that you change the name of your custom preset by double clicking the new preset and changing the name to something you prefer. The items discussed in this section affect the appearance and performance of GibbsCAM. There are other factors that can have an effect on performance and appearance. Please see "Appendix" on page 228 for more information on video cards, video drivers and various recommendations.

Common Tab

The items in this tab control very basic elements of the interface, which apply to all parts of the system.

Enable OpenGL Acceleration

This item turns OpenGL on and off. Most of the system (from display to toolpath to rendering) is accelerated by OpenGL. Therefore, this checkbox should remain selected, except for older or low-end computers or video cards that do not handle OpenGL well.

Activate Dialog on RMB

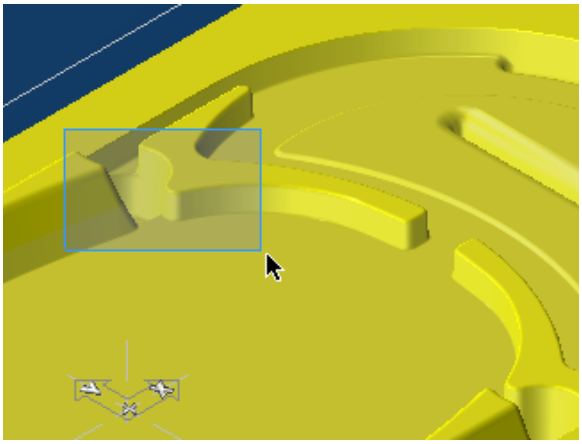
When this option is active, you can bring up the Appearance preferences dialog by pressing and holding the right mouse button when the mouse is positioned over an empty area of workspace — in other words, not over solids, toolpath, or geometry.

Unzoom on View Change

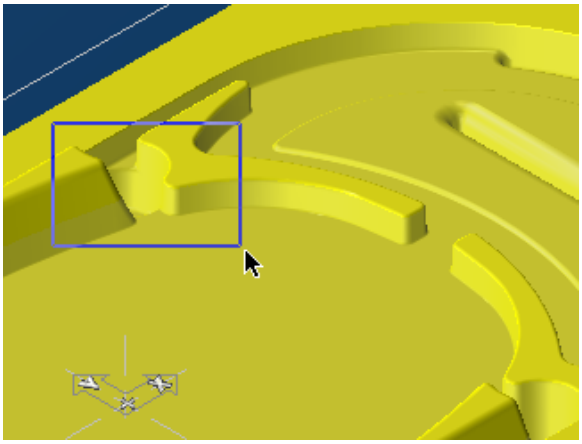
When this item is active, the system will automatically unzoom the part when you change views using any of the trackball buttons or keyboard shortcuts for the standard views. The system will not unzoom if you perform a pan or rotate.

Semi-transparent Drag Rectangle

When this item is active, the system will softly shade the area surrounded by the mouse-drag rectangle.



Semi-transparent Drag Rectangle



Drag Rectangle with no semi-transparency

View Animation Speed

This item controls how fast the transition is between standard views.

Plane Opacity

This item controls how "solid" the CS plane is. The CS plane fills in the grid of the current coordinate system. The plane can be toggled on and off using the Show CS button in the floating Taskbar.

Stock Opacity

You can control the Machined stock transparency in workspace.

Tool Opacity

You can control the Tool transparency in workspace.

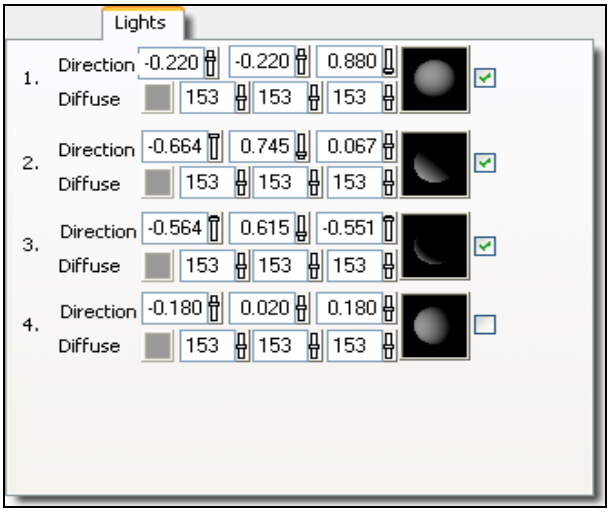
Machine Opacity

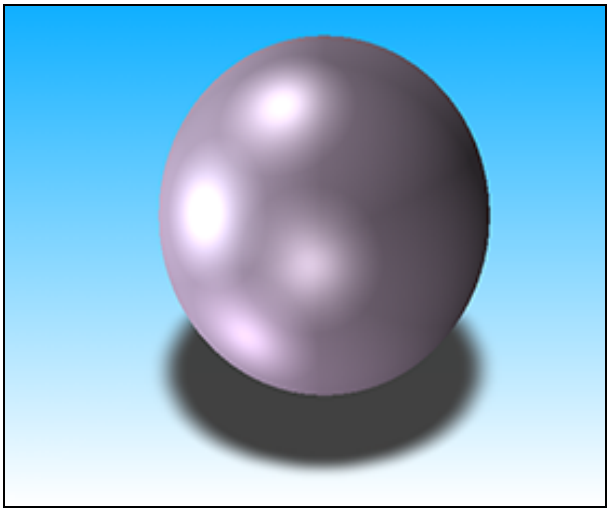
You can control the Machine Assembly transparency in workspace.

Lights Tab

The model is lit by up to four lights. Each light's position around the part can be controlled through XYZ vector values ranging from `-1.0` through `1.0`. Alternatively, you can control the direction by dragging the light across the preview. Each light has a color defined by RGB (Red Green Blue) values ranging from `0` through `255`. Or, you can click on the color swatch and select a light color from the color picker, described in "" on page 18.

Here you see an example of a body lit with four lights. The Faux Shadows option is active.





Colors Tab

You can control the color/transparency or metallic appearance of stock, parts, fixtures, solids, surfaces, and many other items.

Color Swatch

Clicking an item's swatch (color square) opens the color/transparency picker, described in “” on page 18.

Metallic

Clicking an item's pulldown menu allows you to choose a metallic appearance for the item, such as Iron, Gold, Brass, Silver, Copper, Steel, Aluminum, Titanium, TiN (titanium nitride), etc. Or you can choose Custom to open the **Custom Rendering Material** dialog, described in “” on page 18.

Material Properties

The Material Properties of bodies can be modified as well. This is what kind of material a body looks like. By controlling Specular, Ambient light, and Shininess values, a body can look like plastic or aluminum or stainless steel.

Specular

The Specular value is the amount of reflectivity bodies have and color of the reflections. A low value (color) means that the surface is dull. A high value (a brighter color) means the surface is very reflective. You can select a color or define the RGB values for this attribute.

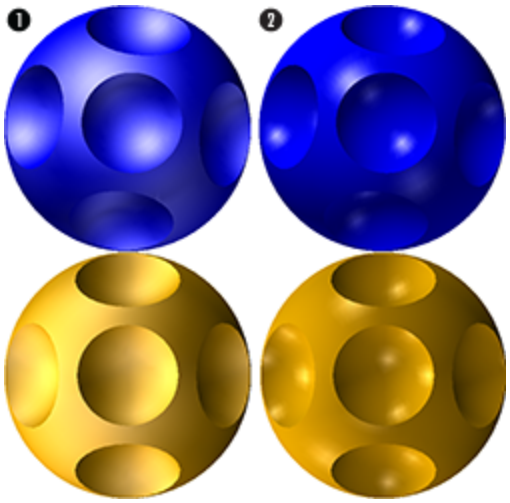
Ambient

The Ambient value is color of the non-directional lighting on the part. You can select a color or define the RGB values for this attribute.

Shininess

Shininess is the size or intensity of reflections. This is a value from 0 through 255. Low values create larger highlights (which are less intense). High values create smaller, very intense bright spots. Here you see two examples of a part with low shininess and high shininess. The lower shininess settings create larger, less-intense reflections. The high values create small, more-intense reflections.

Below are some examples of lighting modes to show you how the different lighting can affect the shininess. The models have two different settings for the part color. The top two bodies are based on the default settings. The bottom two bodies have a lower specular value and four light sources.



1. Low shininess
2. High shininess

An example of shininess, using custom light settings.

Pre-selection Properties

The items in this section control the appearance of items on mouseover or mouse hover when Toggle Pre-Selection Highlighting is in effect:

Solid Edges When active, the edges of a preselected body will be shown in the pre-selection.

Halo Transparency Sets the transparency of the pre-selection. A value of less than 20 will be very subtle, whereas the maximum value, 255, will make the preselection opaque.

Halo Thickness Sets the width in pixels of halos and edges that surround pre-selected elements.

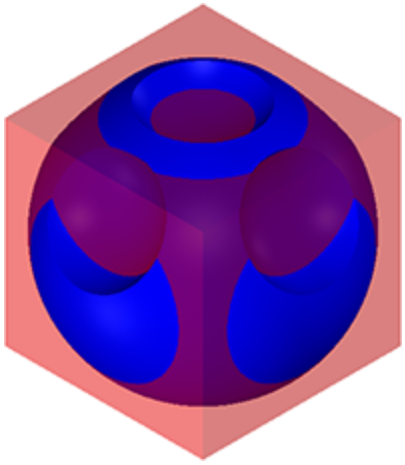
Sim Tab

The items in the **Sim** tab allow you to define the color of the various items displayed by Machine Sim, Op Sim, Tool Sim, etc. You can change the default color of rendered objects by setting the RGB values by key-in or slider. Or, you can click on the color swatch and select a light color from the color picker, described in “” on page 18. Separate colors can be applied to selected bodies (Target), toolpath (Cut), interference (Clash), the primary color of the cutting tool (Tool) and its Shank and Tool Holder), the currently selected operation (Selected Op), errors (Tool Error and Holder Error), and tool color while rapiding (Rapid Tool).

Metallic

Clicking an item's pulldown menu allows you to choose a metallic appearance for the item, such as Iron, Gold, Brass, Silver, Copper, Steel, Aluminum, Titanium, TiN (titanium nitride), etc. Or you can choose Custom to open the **Custom Rendering Material** dialog, described in “” on page 18.

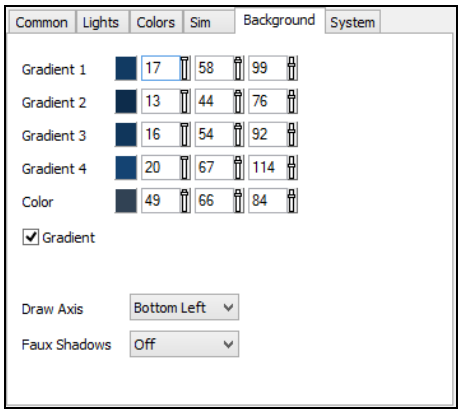
You can set the transparency level for fixtures, stock, and tools. A value of 0 turns the transparency off (the item is solid); the maximum value of 255 makes the item completely transparent. Whether the fixture and/or stock is transparent is controlled by the right-mouse menu of the Render Control palette. The tool transparency is set using the Invisible, Transparent, and Solid buttons on the Render Control palette.

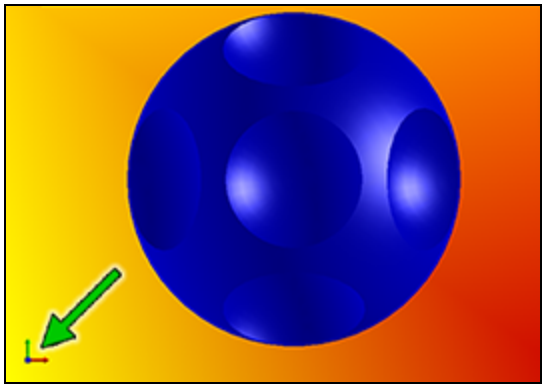


An example of a transparent fixture.

Background Tab

The items in the Background tab allow you to define the colors of the cut part rendered background. Both the single color background (defined by the Color setting) and the gradient background can be customized. The gradient consists of up to four colors, one per corner of the workspace.





Gradient

This option switches the single color workspace to a gradient, making it easier for you to distinguish between standard rendering and the OpenGL rendering modes, which may be slower. The colors of the background may be customized (as seen above) by changing the default colors.

Draw Axis

This enables/disables the display of a small Red (horizontal), Green (vertical) and Blue (depth) axis marker in any of the four corners of the workspace. This marker can help visualize the part's orientation, and hovering over the marker with the mouse enables a temporary trackball. Choose the location to display this marker, or turn it off.

Faux Shadows

This item enables the drawing of shadows for bodies. The shadows are not created from light sources but are instead just to the Bottom, Back or Side of the model. The shadow can help with part orientation but can slow the display.

System Tab

These options normally do not need to be modified. The items found in this tab can help improve the performance of Op Sim or Tool Sim, depending on your CPU, the amount of memory available, the video card, the type of monitor and its display. Some experimentation may be necessary to determine the optimal settings for any given computer. As a general rule, computers with higher quality/faster cards will not need to change these options.

Display Lists

This is a speed optimization. It is recommended to turn this on if you have a fast video card with 128+MB of memory. This option outputs solid geometry to OpenGL display lists. Using display lists can improve the performance of Simulation and, on some graphics adaptors, provides performance improvements for rendering all solids. Generally, newer cards with hardware geometry acceleration will take advantage of Display Lists. Using this option can require large amounts of video adaptor memory.

There are two instances where turning off Display Lists may be useful.

a. If you find that Legacy CPR or Simulation crashes with some regularity (this is likely the result a data-management issue with a MachineWorks/graphics card manufacturer).

b. Sometimes using display lists can result in fairly slow rendering. This situation is hard to gauge but it has to do with inefficient display list management inside the graphics card manufacturer's OpenGL implementation, typically where the card shares video RAM with the computer's memory. A good example is rendering a large contouring pass will be slower with display lists than without. This is because certain video card manufacturer's OpenGL implementations are bad at managing display list memory. All the tiny changes to contoured stock and the data added to the display lists can adversely affect the simulation speed through inefficiencies.

Partial Updates

This is a speed optimization. Partial Updates lets the system redraw only sections of the display that have been modified. Among newer video cards that use Display Lists, some will benefit from Partial Updates, others will not. Partial Updates is particularly useful in optimizing display updates in Simulation/CPR during animations and especially when the Steps Per Update is smaller and animation runs more smoothly. Please note that some video cards do not support partial update behavior. The trade-off of using Partial Updates is that on some video cards there may be situations where a part of the display is not drawn correctly.

Vertex Arrays

This option structures solid topology as an efficient point-reduced mesh and uses optimized OpenGL functions to send data to the graphics adaptor. This option requires additional memory to store meshes and additional processor bandwidth to pre-calculate the meshes. This can be a big speed improvement on systems with a adequate video card, i.e. a lot of memory.

Soft Shadows

Applies a blurring filter to the projected shadow. In some parts with extreme bounding volume aspect ratios (like MTM), soft shadows may eliminate all projected feature details. On some video cards this can have a large impact on solid rendering performance.

Driver

Select the type of OpenGL rendering that will be used in the next graphics session — either the next part that is opened or the next GibbsCAM session that is launched. The options are:

Hardware

This is fully accelerated OpenGL. This option uses OpenGL to render all geometry, which includes solids and toolpath. This mode requires a redraw whenever an overlapping window is moved over the geometric window.

Software OpenGL

This is a software simulation of OpenGL. This is a standard implementation across all Windows hardware platforms so it should work the same on every machine, but provides no acceleration. This mode does not support shadows and uses the non-OpenGL method for rendering non-solid geometry. This option does not require a redraw whenever an overlapping window is moved over the geometric window. This option provides for a more compatible result but sacrifices speed and power. This option should only be used if your video card has poor or no OpenGL support, or if you experience unacceptable graphical issues.

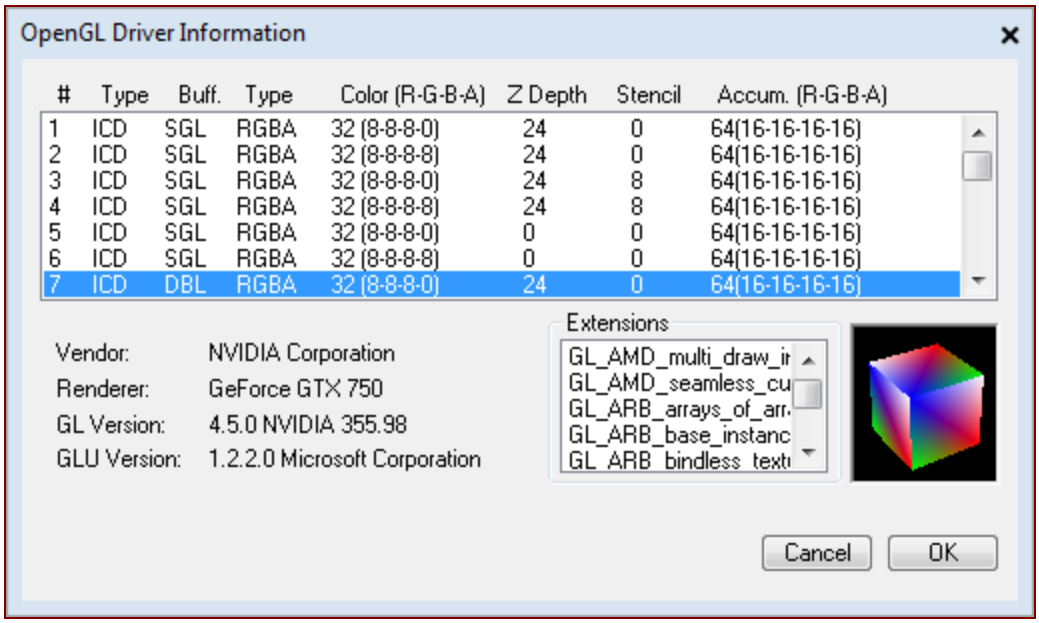
P-Buffer

This is a partially accelerated OpenGL. This option uses accelerated OpenGL to render solid geometry and shadows but, like the Software mode, uses the non-OpenGL method for rendering non-solid geometry and does not require a redraw whenever an overlapping window is moved over the geometric window. This mode is not supported on all graphics adaptors but appears to work well on Nvidia. This option provides for a highly compatible rendering system but sacrifices speed.

Facet Body Opacity

This item sets the visibility of facet bodies. Facet bodies are created from STL files. See the Mill manual for more information.

Driver Info

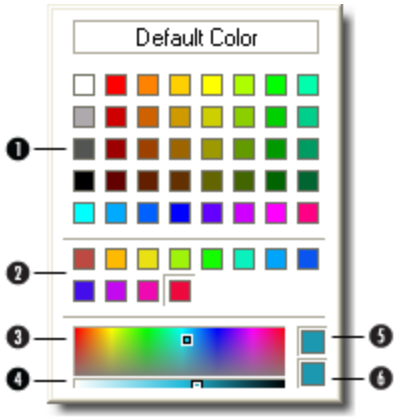


This item opens a dialog that displays the currently available OpenGL drivers on the machine, their supported extensions, and the amount of currently used and available graphics memory.

Test Frame Rate

This item tests the number of times per second that the current display is redrawn.

Color Picker (Select Color dialog)



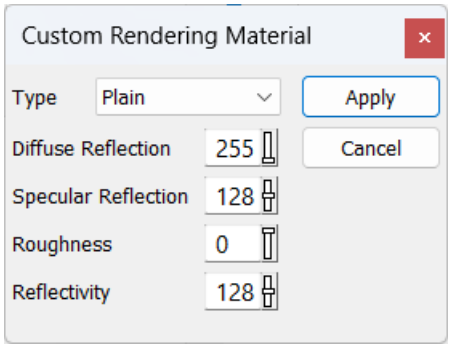
1. Standard Colors
2. Custom Colors
3. Color Mixer
4. Saturation Level
5. Add Custom Color
6. Use Custom Color

The **Select Color** dialog lets you select a color from one of the forty standard choices. Clicking the Default Color button is the same as clicking Escape, inasmuch as the dialog will close and the color selection will not change.

To create a custom color, click near the color you wish to use, adjust the slider to select the saturation of the color, and click one of the two buttons to the right of the color mixer and slider. The upper color button ("Add Custom Color") selects the specified color and adds it to the current custom palette. The lower color button ("Use Custom Color") selects the color but does not add it to the custom colors palette.

Custom Rendering Material

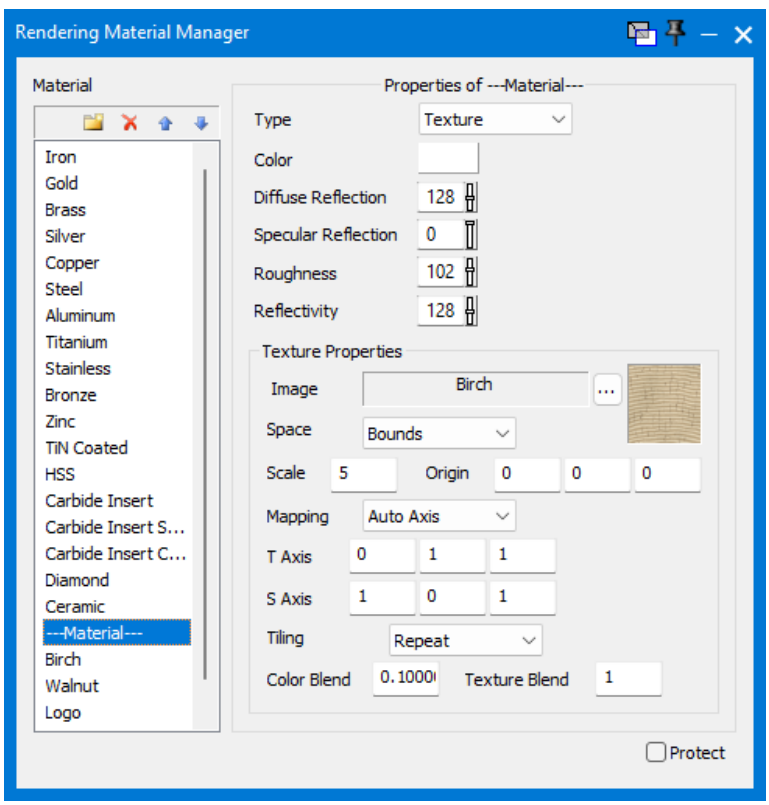
The **Custom Rendering Material** dialog lets you design your own Plain or Metallic custom appearance by specifying values from 0 to 255 for each of the four parameters Diffuse Reflection, Specular Reflection, Roughness, and Reflectivity.



Edit Rendering Materials...

Clicking this button opens a dialog that lets you create your own rendering material setting and choose options. This can help speed up the system display.

Rendering Materials Dialog



Material List

Select, add, delete, rename and reorder defined materials.

Material Type
Select between Metallic, Texture and Plain materials.

Material Color
Define the material color.

Material Diffuse Reflection
Material Diffuse Magnitude Specify the amount of the material's diffuse color (range 0-255). Larger values result in brighter materials.

Material Specular Magnitude
Specify the amount of the material's specular reflectance (range 0-255). Larger values result in shinier materials.

Material Roughness
Specify the material's roughness (range 0-255). Larger values result in duller materials.

Material Reflectivity
Emphasizes the reflective character of the material (range 0-255). Larger numbers reflect more of their environment.

Texture Properties

Image
Set Current Texture Image. Brings up the Texture Manager.

Space
Texture coordinate spatial mapping. Selects the spatial relationship between the associated solid and the texture.
Object- assumes textures are effectively the same size as the associated solid.

- Bounds- scales the texture to match the bounds of the solid.
- Min Bound- scales the texture to match the smallest dimension of the solid bounds.
- Max Bound- scales the texture to match the largest dimension of the solid bounds.

Texture Tiling scale
Multiplies the texture size to create more or fewer tiles on the solid.

Texture Tiling origin
Shifts the texture on the solid.

Texture coordinate axis mapping

Linear- Use **S and T axes** to define location/orientation of the texture on the solid.
Auto Axes- Use the solid's surface normal to define the Location/orientation of the texture on the solid.

Texture Height Axis specification

Texture Width Axis specification

Texture Tiling behavior

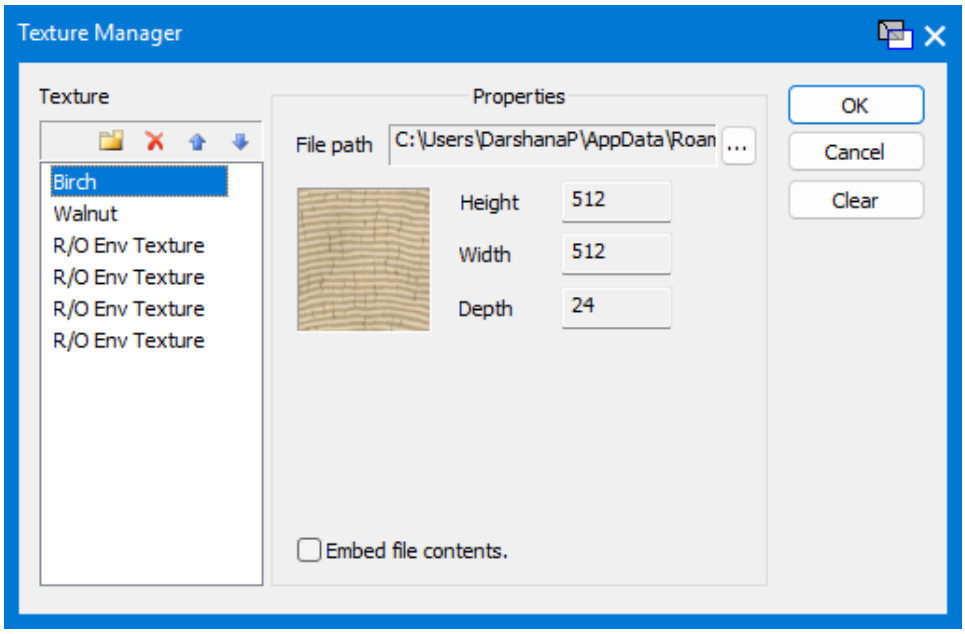
- Repeat repeats the texture across the surface
- Clamp Out of range texture assignments are set the texture boundary values.
- Border Out of range texture assignments are set the material color.

Color Blend
Texture material color blend value specifies how much material color to allow in the texture display.

Texture Blend
Texture color blend value specifies how much texture color to allow in the texture display.

Protect
Disables modifications of material definition. Protected materials can't be renamed, deleted or have their definitions modified.

Texture Manager



Texture image list
Select, add, delete, rename and reorder defined texture images.

File Path
Texture image file selection: Pressing the browse button brings up a file selection dialog to select an image file to associated with the current texture. Support file formats are BMP, JPG, and PNG.

Embed file contents
Embeds the texture image file contents in rendering table definition file. This allows the texture definition to be file location independent.

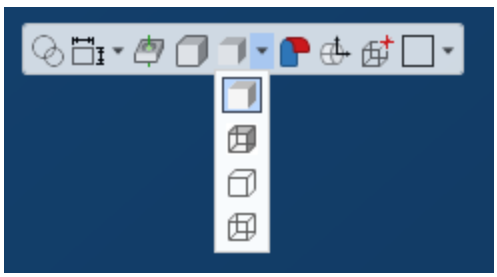
Ok/Cancel
Apply current texture to current material in the Material Manager.

Clear
Removes the defined texture from current material.

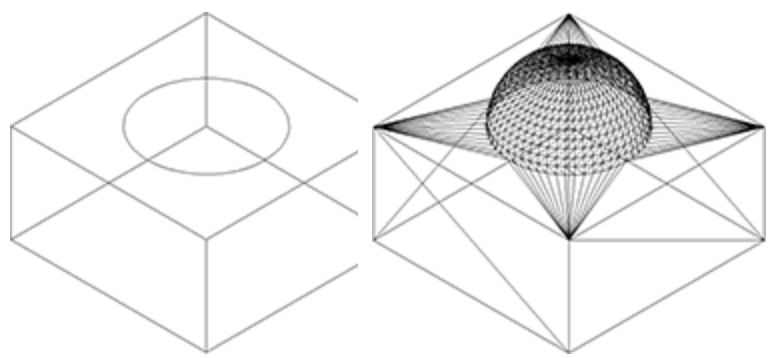
Printing
Geometry, bodies, toolpath and rendered images can all be printed by selecting the Drawing item in the Print submenu. The Printing Preferences section of the **Display** preferences lets you specify how the system will handle the background color and the colors of the image. Separate options are available for printing the geometry and toolpath and for the rendered part. Selecting Screen will print the drawing exactly as it appears on the screen, with a black background and the colors as they appear on the screen. Selecting Screen on White will print the image in color, but on a white background. Selecting Black on White will convert all the colored items to black and print them on a white background. If the printer being used is a black and white printer, the Black on White option should be selected to ensure that all portions of geometry, including those that are of a light color, can be seen in the print. If a color printer is being used, any option can be used.

Bodies

The Bodies section of this preference contains settings that affect the graphic display of bodies and sheets.



The Render Faces button (found in the Floating Taskbar) determines whether bodies and sheets will be rendered as objects or wireframe drawings. The wireframe drawings of the solids or sheets will be displayed as either Edges or Facets depending on the selection made for Wire Drawing.



Edges and Facets in Wire Frame mode

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. This setting determines the faceting resolution when bodies and sheets are rendered. The smaller the chord height, the closer the facet will be to the arc or circle, resulting in a better rendered image. When the setting is very tight, the on screen drawing of bodies and Boolean operations can become slow. A setting of 0.01 inches or 0.25mm is recommended for most systems.

Grid Brightness

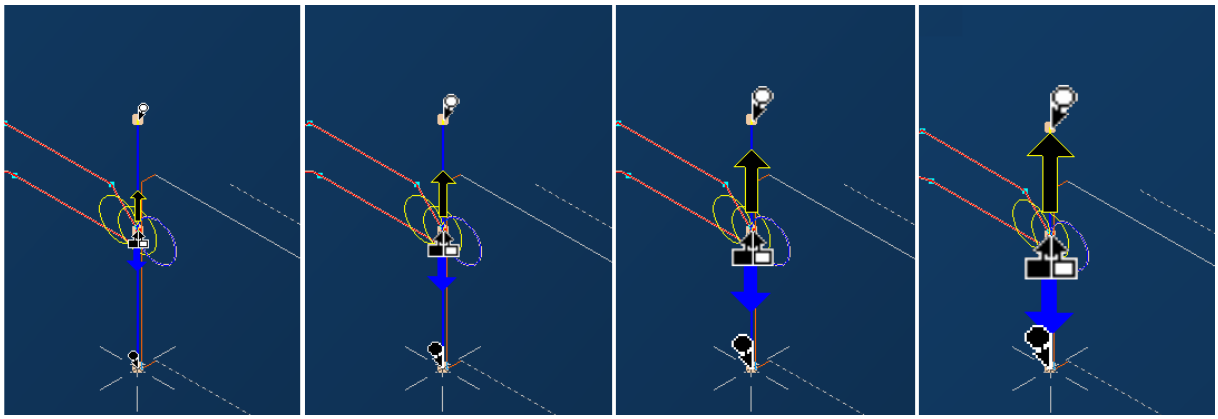
The Grid Brightness option is for the Level 2 interface only. This slider controls the brightness of the coordinate system grid in the stock diagram. The brightness can be adjusted by moving the slider.

Other Items

Marker size

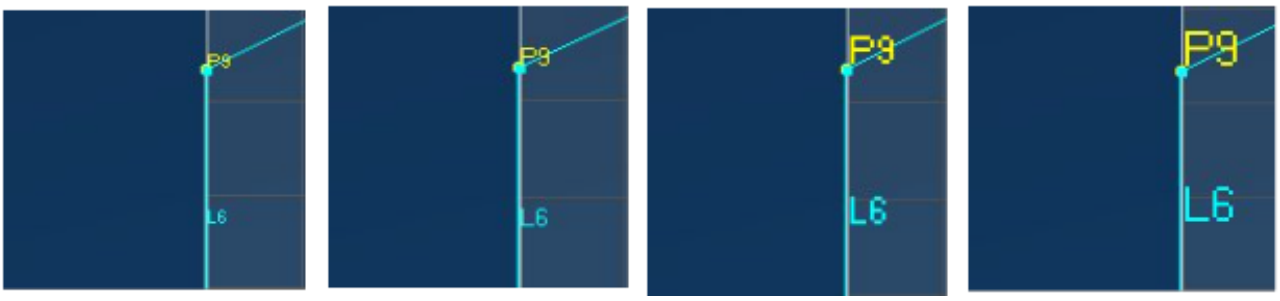
You have the option of displaying Small, Medium, Large or Extra Large Markers.

You can use Undo and Redo to revert or restore changes made to marker positions. If markers disappear after you click outside the contour, you can also use Undo or Redo to restore the markers and their previous positions.



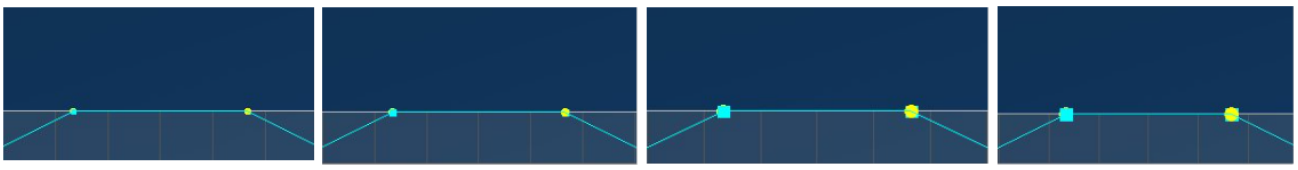
Geo Label size

You have the option of displaying Small, Medium, Large or Extra Large Geo Labels.



Point size

You have the option of displaying Small, Medium, Large or Extra Large Points.

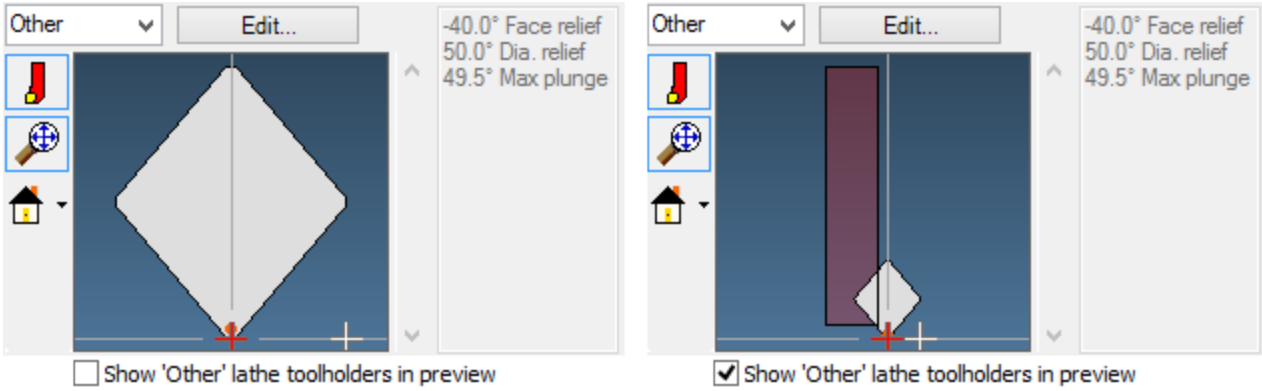


CS Label size

You have the option of displaying Small, Medium, Large or Extra Large CS Labels.

Show 'Other' turning toolholders in preview

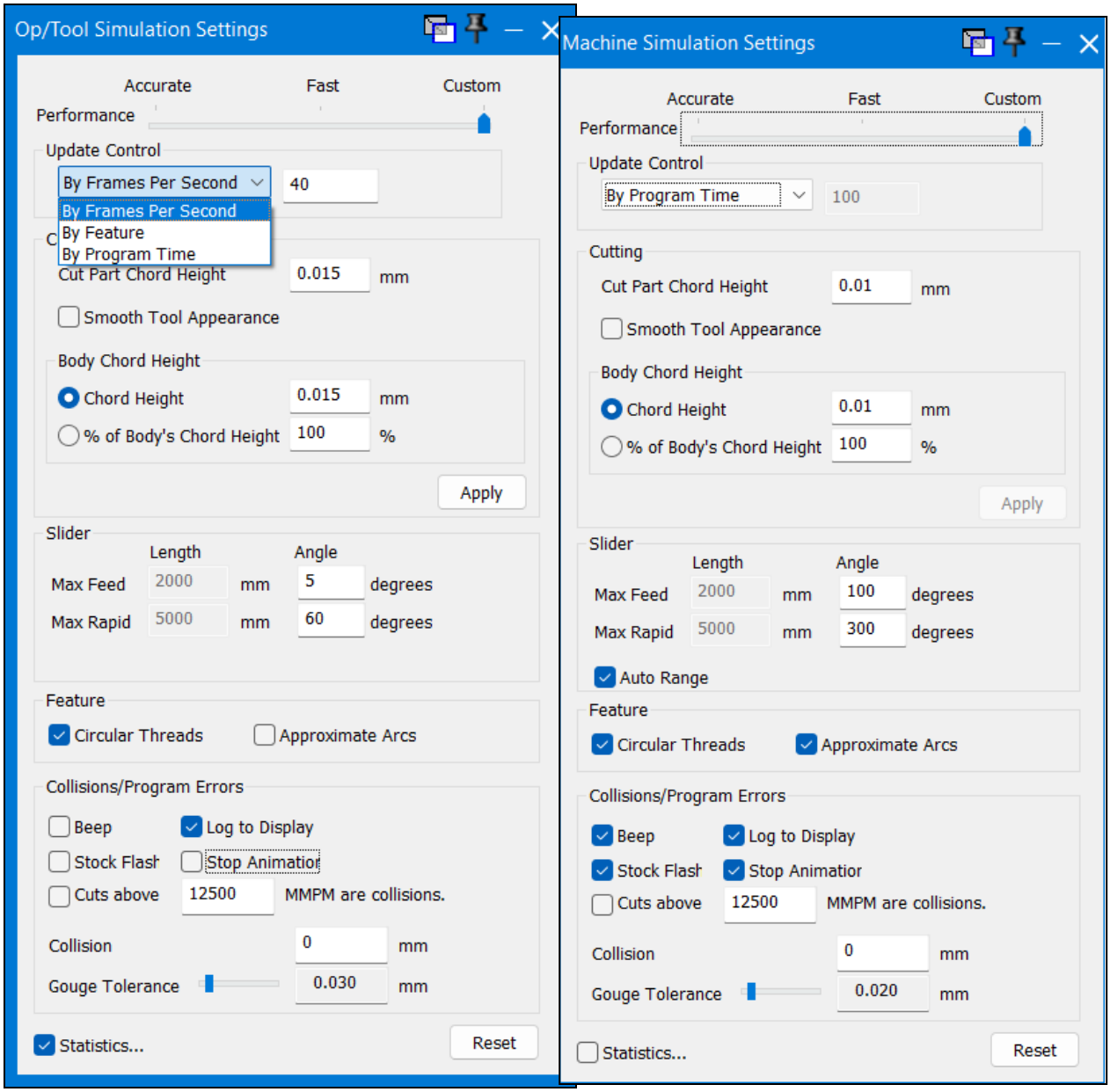
For Turning inserts when no toolholder has been specified, select this checkbox to display a "pseudoholder" (rectangle drawing) in the tool preview window.



Lines drawn for high degree curves

This setting is used when higher-order splines are imported into the system. The value specifies the number of line segments that will be drawn to create the spline. For a smoother spline drawing, enter a larger number. For the spline to be drawn faster, enter a smaller number. This value only affects the drawing resolution of the spline, not the machining tolerance.

Settings for Op Sim, Tool Sim, and Machine Sim



There are separate settings dialogs for Op/Tool Simulation and for Machine Simulation. Both have virtually the same options, but they save separate data files. They are opened by using the Display preferences tab (File > Preferences > Display).

These dialogs can also be accessed by right-clicking the corresponding rendering dialog and, from the context menu, choosing the Settings... menu option.

Performance

Drag the performance slider to choose predefined settings to set greater accuracy or greater speed, or to choose custom settings.

Update Control

From the pulldown menu, choose one of these options:

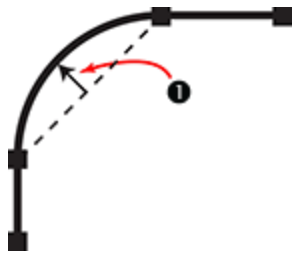
- By Frames Per Second: Simuldation will play at a uniform speed using the value you enter.
- By Feature: Simulation will pause after completing each feature.
- By Program Time: Simulation will be proportional to the program time, approximating the actual machining time.

Cutting

Frames per Second

A greater Number of frames equals better quality, but less speed.

Cut Part Chord Height



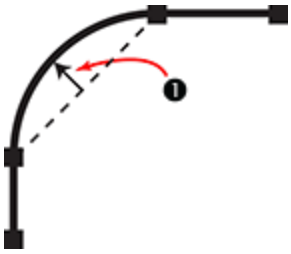
1. Chord Height

This setting is the resolution for the cut part displayed. The smaller the value, the higher quality of the display and the more resources needed by the system, resulting in a slower rendering, depending on your system's capabilities.

Body Chord Height

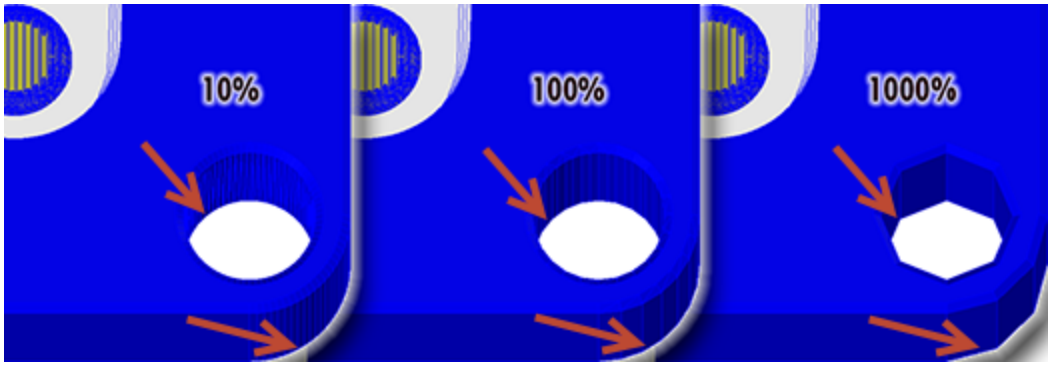
Determines the resolution of bodies (part, stock, and fixtures) in rendering.

Chord Height



You can set the Chord Height to a specific value, or a percentage of the Body's Chord Height:

The % of Body's Chord Height option uses the value set in the Properties dialog. A setting of 100% uses the body Chord Height while a setting of 10% is 1/10th of the body Chord Height. Any percentage between 1 and 100,000 is acceptable. As the percentage is set higher, the body displays faster but appears rougher.



Slider

This item affects the responsiveness and quality of the rendering. The Angle values can have a very large impact on the rendering speed versus quality, especially on rotary operations. A low number will create very small angles in rotations, resulting in a smooth image. A high number can create a rendered part that is not smooth but is very fast. Please note that this does not affect toolpath, only the rendered part.

For Machine Simulation, the Auto Range checkbox enables the use of the stock size to define minimum and maximum feed and rapid lengths.

Feature

Collisions/Program Errors

Collision and Program Error checking are available for Op Sim and Machine Sim. You can choose any or all of the various feedback methods that alert you to a collision: The Beep option provides an audible alert; Log To Display outputs errors to the **Clash Console** log window; and Stock Flash provides a visual alert to the error by flashing the rendered stock. Stop Animation will cause the rendering to stop when a collision is detected.

To guard against high-feedrate cuts, check the box for Cuts above ___ are collisions and supply a value for maximum feedrate. This can help you foresee and prevent broken tools and damaged parts even when rapid is not being used.

Collision Tolerance

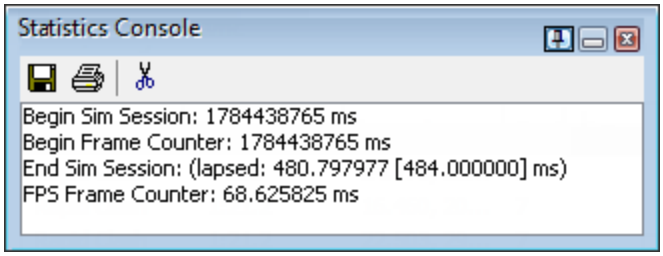
The Collision Tolerance setting allows a different value for parts in metric and inches. Any collision within the specified tolerance generates a collision alert.

Gouge Tolerance

Gouge Tolerance allows you to specify how much removed stock material can be ignored without considering it to be a gouge. Drag the slider to the left to decrease (tighter tolerance), or to the right to increase (looser tolerance).

Statistics

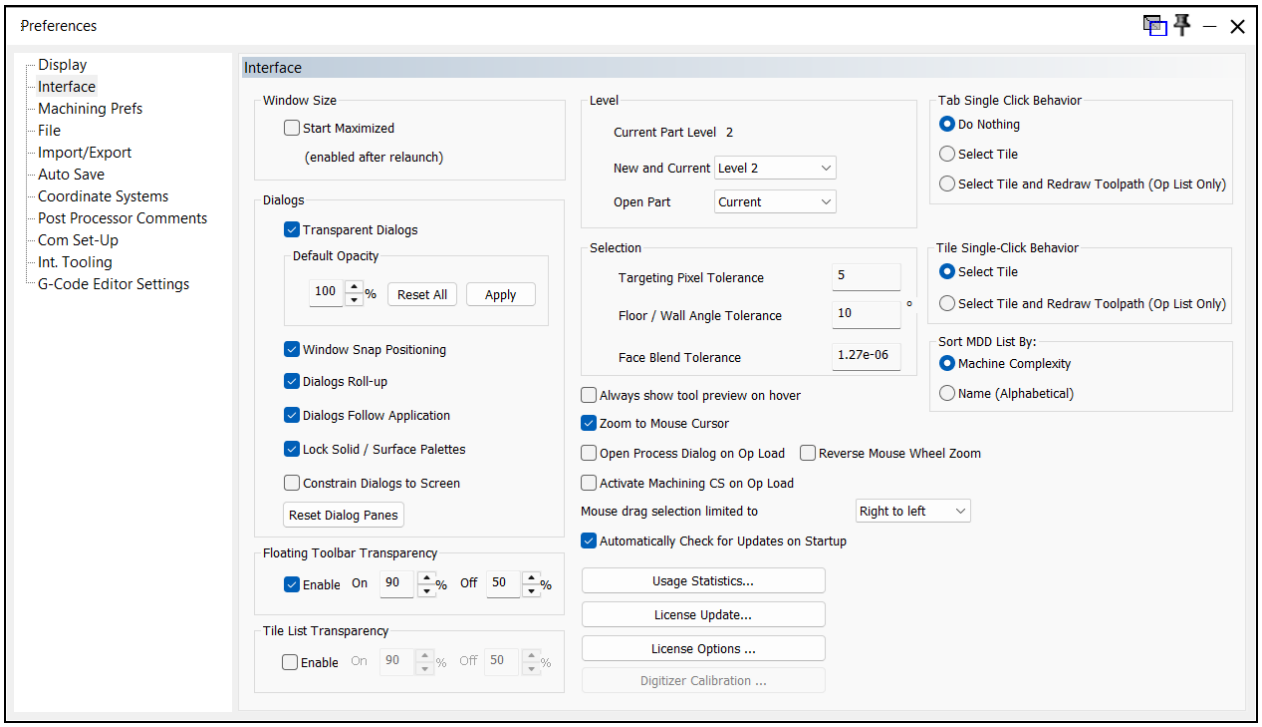
When the Statistics option is enabled, a window will open when you activate Simulation. The window logs the current frame-rate for your machine as well as any logged errors for the simulation session.



Reset

Resets all values to the default .

Interface Preferences



The items found on this tab affect the basic interactions of GibbsCAM, including its “look-and-feel”. The items on this tab let you set base size of GibbsCAM, the interface you will use, how dialogs behave, selection behavior, and many other things.

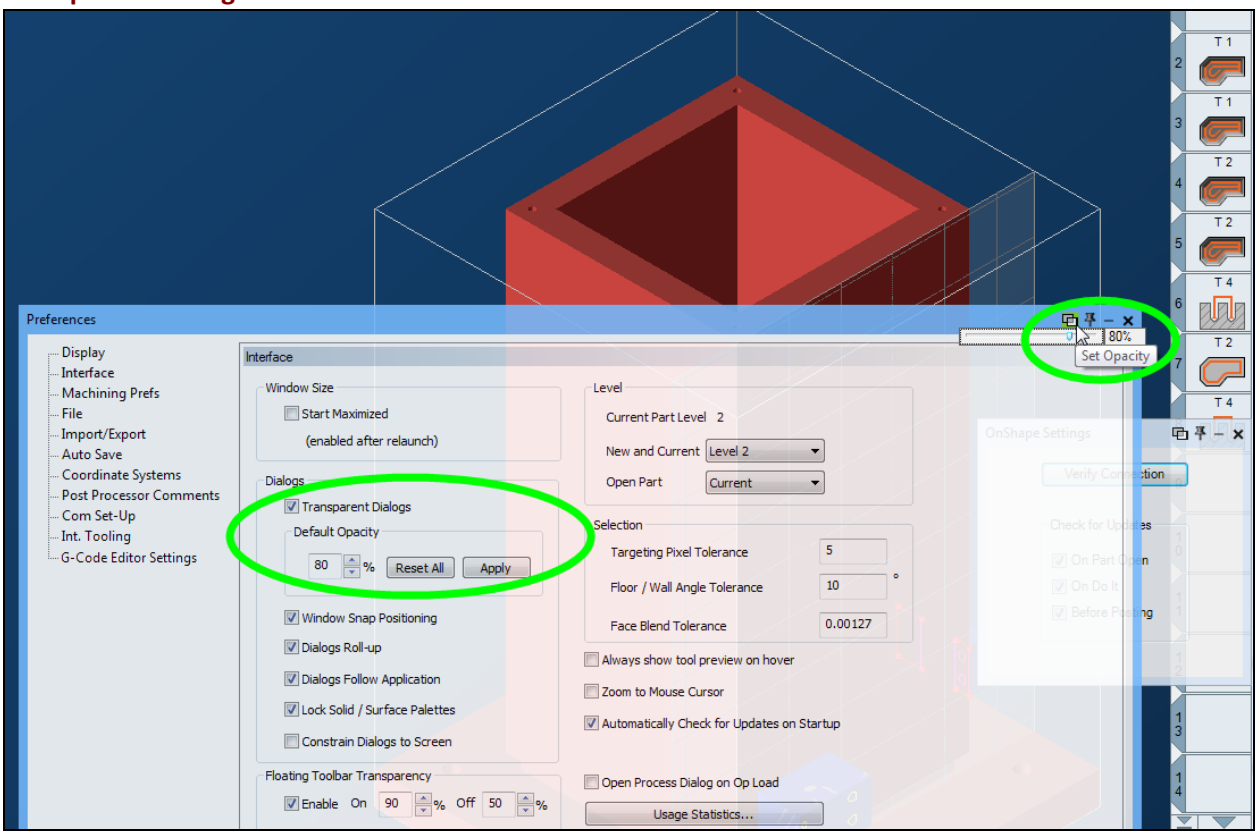
Window Size

This option determines whether the application will occupy all of your monitor (Start Maximized) or will be in windowed mode when you launch the application. Selecting the Enable Caption will leave the blue bar visible (the caption bar shows the name of the application and has a Close box). Deselecting this checkbox will hide the bar so that the menu bar is at the top of your screen.

When the application is not maximized, you can resize the window by dragging an edge or corner.

Dialogs

Transparent Dialogs



Selecting this option allows each dialog to have a custom transparency setting. You can also set a default transparency setting that all applies to all dialogs until they are customized.

When Transparent Dialogs is in effect, the title bar of each dialog includes an icon ( ).

Click the icon to bring up a slider that lets you set a custom transparency for the dialog.

If Transparent Dialogs is not selected, the icon is hidden and all dialogs are opaque.

Window Snap Positioning

Enabling this option will cause windows and dialogs to “snap” to nearby windows. This allows for easy and neat arrangement of windows.

Dialogs Roll-up

Selecting this option will cause dialogs to automatically “roll up” to show only their title bar when the mouse leaves the dialog. When the mouse enters the title bar, the dialog unrolls to show its contents. Enabling this option adds an icon to the title bars. When the “pushpin” is down ( ), the dialog does not roll up. When the pushpin is up ( ), the dialog can roll up.

Dialogs Follow Application

Selecting this option will cause all windows and dialogs to follow the application window if it is moved.

Lock Solid/Surface Palettes

This option affects the Solid Modeling and Surface Modeling palettes, the sub-palettes and their dialogs. Selecting this option will cause the sub-palettes and dialogs to follow the Solid Modeling and Surface Modeling palette. When the parent palette is moved, all palettes and dialogs accessed from this palette move along with it.

Floating Toolbar Transparency

This allows you to specify the degree of transparency for the Taskbar and custom floating toolbars. Select Enable to change the default values. The value for On specifies toolbar opacity when the cursor hovers over a floating toolbar item. The value for Off specifies toolbar opacity when the cursor is away from the floating toolbar. The default values make floating toolbars almost opaque (80% On) on mouseover, and almost transparent (20% Off) the rest of the time.

Level

The Interface preferences allows you to toggle between the Level 1 and 2 interfaces. Although you should use the interface that best suits your needs and knowledge, using the Level 2 interface is recommended. This is because the Level 2 interface is a more refined and optimized interface, providing a richer environment. New and Current Part lets you specify the default level interface the system should use. Open Part lets you designate whether the system will open an existing part with the interface level that the part was originally created in, or with the interface level that the system is currently running.

Level 1

This interface provides full functionality to the production Mill and Turning modules, using the Command Toolbar. The full functionality of Level 1 does not, however, encompass Advanced CS, Tombstone Management System (TMS), Multi-Task Machining (MTM), Solids Import, 2.5D Solids, SolidSurfacer, or Machine Sim capabilities.

Level 2

The Level 2 interface provides the most advanced set of features, including the Command Toolbar and floating Taskbar. The Level 2 interface is necessary for using product options Mill/Turn, Polar & Cylindrical Milling, MTM, Advanced CS, Solids Import, SolidSurfacer, Machine Sim, or TMS, and is available to all users.

Open Part

This defines the interface level an existing part will be opened in. Part opens a vnc file and sets the interface level to match the level used to create the file. Current opens a vnc in the interface level currently in effect.

Selection

Targeting Pixel Tolerance

This setting specifies the accuracy you need to have when you select an item in the workspace. The default setting, 5 pixels, means that the workspace item is selected if it lies within 5 pixels of the cursor tip when you click the mouse, and not selected if it lies 6 or pixels from the cursor tip. If more than one element is within range, the system selects the closest geometry.

Floor/Wall Angle Tolerance

This item is used with the face selection options found when right-clicking on a solid. Any face that falls within the tolerance specified in this dialog will be selected. For example, when selecting wall faces with the default setting of 10°, any face at an angle from 80° to 100° of the selected face will become selected.

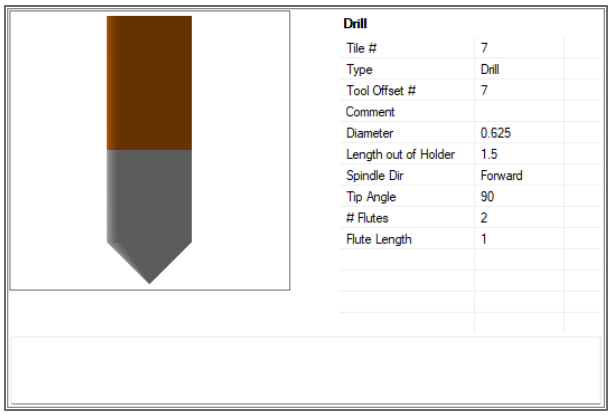
Face Blend Tolerance

This item is used with face selection. When selecting fillet faces, Face Blend Tolerance is used to detect rolling ball fillets.

Other Items

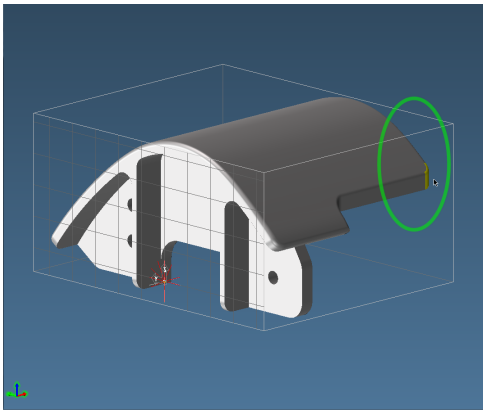
Always show tool preview on hover

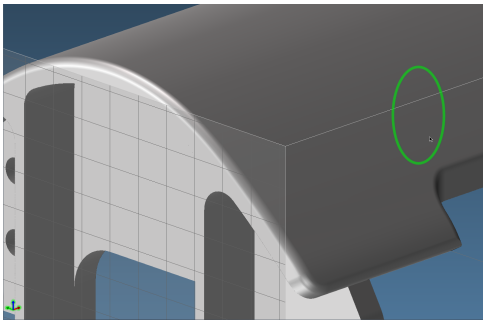
Specifies whether a preview of the tool is shown when you hover the cursor over a tool tile.



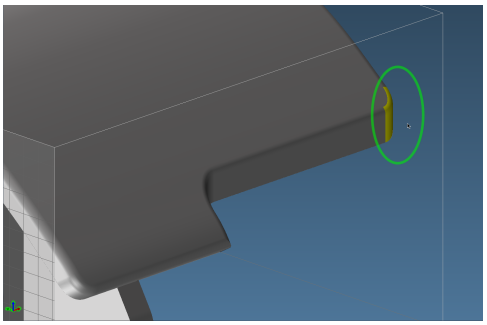
Zoom to Mouse Cursor

For *scrollwheel* zoom actions: If the Zoom to Mouse Cursor checkbox is selected, then the zoom center is the cursor; otherwise, the zoom center is the window center.





Zoom to Mouse Cursor ***not*** selected (top): Window-centered zoom



Zoom to Mouse Cursor ***is*** selected (bottom): Cursor-centered zoom

Reverse Mouse Wheel Zoom

When this checkbox is selected, the effect of rolling the mouse wheel is reversed (from zoom-out or zoom-in or vice versa).

Open Process Dialog on Op Load

When this checkbox is selected, double-clicking an operation tile will cause the associated process dialog to open.

Activate Machining CS on Op Load

When this checkbox is selected, double-clicking an operation tile will cause the machining coordinate system (CS) of the associated process or process group to be activated.

Mouse drag selection limited to

Your choice here specifies the system behavior when you click-and-drag your mouse across the workspace to select items that lie either totally or partly within the mouse-drag rectangle.

- When Right to left is in effect, a right-to-left mouse-drag includes only elements that lie completely within the rectangle, whereas a left-to-right mouse-drag includes elements that lie at least partly within the rectangle.
- When Left to right is in effect, a left-to-right mouse-drag includes only elements that lie completely within the rectangle, whereas a right-to-left mouse-drag mouse-drag includes elements that lie at least partly within the rectangle.

Automatically Check for Updates on Startup

When this checkbox is selected, the system will check if a newer version of the software is available for download. To disable this function, uncheck the checkbox.

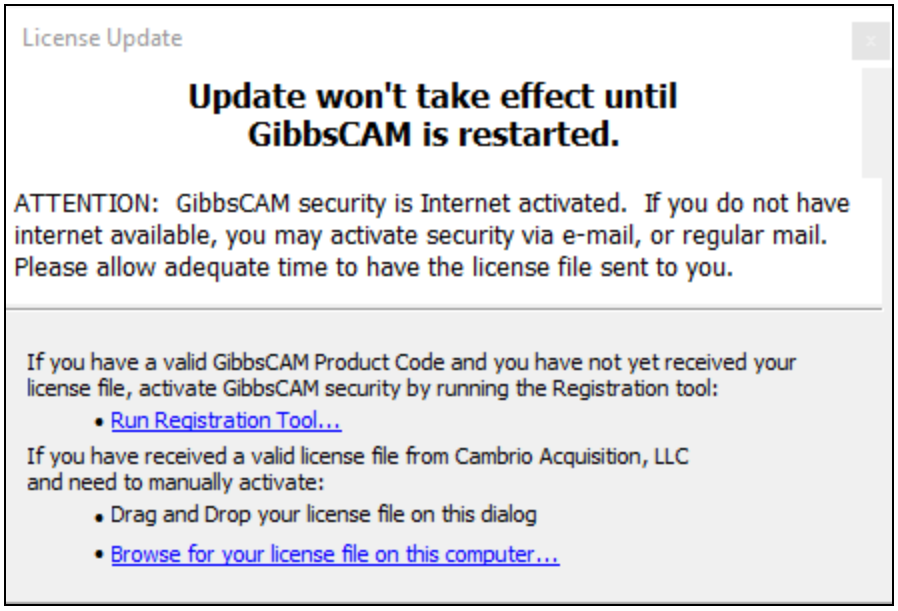
Usage Statistics

When GibbsCAM is installed, a dialog asks permission regarding the collection of usage statistics. These consist of general technical data (such as OS platform and graphics subsystem) as well as software module statistics that are specific to GibbsCAM, such as frequency of library/module calls and GibbsCAM software performance. No personally identifiable information is ever collected (such as company name or user name, computer or network ID, or the like), and no information on part design or part names is ever collected. Information is never collected without permission, and permission can be withheld either by the system administrator or (if the administrator allows it) by GibbsCAM.com users.

At installation time — or at any other time by running `Bin\GCStatsAdmin.exe` — the administrator chooses whether to opt in or out of usage data collection, and also specifies whether or not to permit individual users to override the administrator's choice.

If users have this permission, they can exercise it as a preference: File > Preferences, **Interface** tab, Usage Statistics button. Any changed setting takes effect immediately upon clicking the OK button.

License Update



The License Installation or License Update dialog lets you drag an RLM license file (*.lic) onto the dialog for application installation, browse the computer for a license file (*.lic), or run the Registration Tool to obtain a license file (*.lic) through the Gibbs Activation Service.

Note that this release uses Reprise License Manager (RLM), not Cimatron License Server (CimLM). If you try to use a CimLM *.cml file instead of an RLM *.lic file, it will not work.

> **Reseller Activation** - If your Reseller will be assisting you with your installation, have your registration files (and hardware keys, if any) ready in advance.

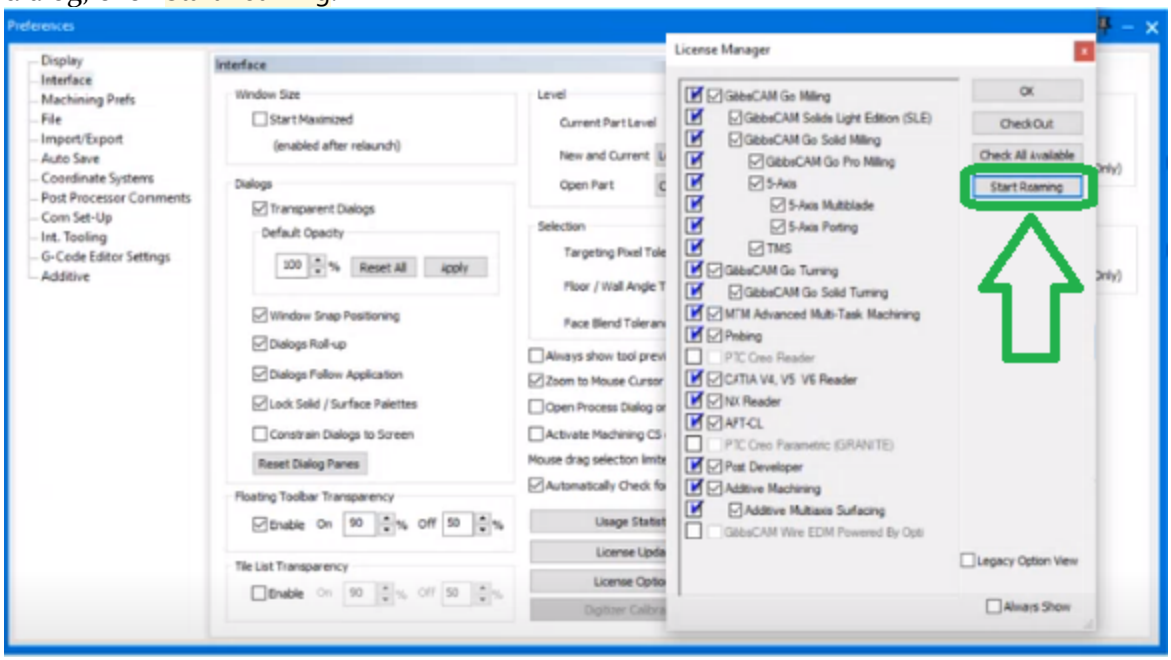
License Options

This preference is only available when the Network Licensing Option (NLO) is in effect. This preference lets you specify the licensing options to be checked out from the license server. Select the options you wish to use and click Check Out. If the options you want to use are available, you can then click OK; otherwise, you will need to select other product options.

This is useful when a full compliment of product options is unavailable or unnecessary. For example: Suppose your company has five Mill seats, five Advanced CS seats, and three SolidSurfacer seats. Even if three other users are currently using SolidSurfacer, you can still check out a Mill and Advanced CS seat, because there are two open seats for each option.

NLO License borrowing with checkout/return/expiry parameters

Users can now check out an NLO license for a specified time period, then disconnect from the network. This allows users to borrow licenses, disconnect, move their laptop, and keep using the GibbsCAM products even when the license server is stopped. Borrowed licenses can be returned manually, or they can be set to expire after a set time. *Where users can find this option*: Under **File** > **Preferences** > Interface, click License Options; then, in the **License Manager** dialog, click Start Roaming.



Digitizer Calibration

This preference allows you to set the functionality of a tablet. The default setting is Tablet as a mouse.

Tablet as a digitizer in three places: 1) Calibration 2) Mouse Point, 3) Mouse Line. When the Tablet as a digitizer setting is selected, calibration points must be entered. Any three points on a part print can be used for this purpose. Place the cursor in a text box, select a point on the print with the digitizer puck, and enter the X and Y values. Repeat this process for two more points.

Calibration must be performed for each part you wish to digitize, and the calibration does not persist across instances of GibbsCAM. The calibration can be performed whether a part is open or not.

Selection

Targeting Pixel Tolerance

This setting specifies the accuracy you need to have when you select an item in the workspace. The default setting, 5 pixels, means that the workspace item is selected if it lies within 5 pixels of the cursor tip when you click the mouse, and not selected if it lies 6 or pixels from the cursor tip. If more than one element is within range, the system selects the closest geometry.

Tab Single Click Behavior

This preference lets you specify the system behavior when you single-click the tab of a tool, process, or operation tile.

Do Nothing

Selecting the tab has no effect. This is the default setting.

Select Tile

Selecting the tab causes the tile to be selected.

Select Tile and Redraw Toolpath (Op List Only)

Selecting the tab of a tool or process tile causes the tile to be selected. Selecting the tab of an op tile causes the tile to be selected and the toolpath to be redrawn.

Tile Single Click Behavior

This preference lets you specify the system behavior when you single-click a tool, process, or operation tile.

Select Tile

The tile is selected. This is the default behavior.

Select Tile and Redraw Toolpath (Op List Only)

Selecting a tool or process tile causes the tile to be selected. Selecting an op tile causes the tile to be selected and the toolpath to be redrawn.

Sort MDD List By:

This preference lets you specify how the system displays lists of MDDs, as in the DCD > **General** tab, Machine pulldown.

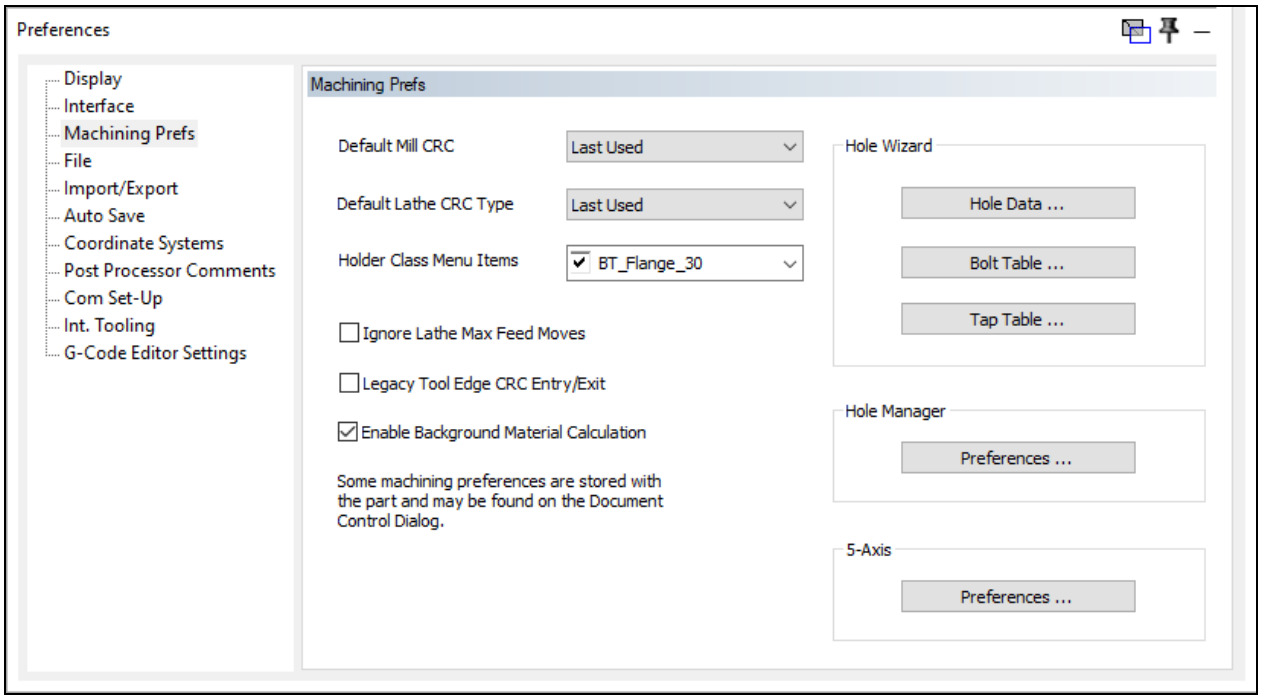
Machine Complexity

MDDs are listed in order of increasing machine complexity. This is the default behavior.

Name (Alphabetical)

MDDs are listed alphabetically by machine name, case-insensitive, with numerals ranking before letters.

Machining Preferences



The Machining Preferences tab contains several settings for the system's behavior when generating toolpath.

Default Mill/Turning CRC Type

This option allows you to set the default behavior of Cutter Radius Compensation in operations. This is a default; for any particular part, you can override this default in the DCD (Document Control dialog), Machining Preferences tab. The Default CRC Type options include Tool Center, Tool Edge, Finish Profile and Last Used. The toolpath displayed on-screen will not change with each selection, but the posted output will vary according to your selected preference. Additionally, if you convert an operation's toolpath to geometry, the resulting geometry will be reflective of the CRC Type used.

When using custom form tools for turning-type parts, do not use the Tool Edge setting, because the entirety of the form tool is used in the calculation. Tool Center is the recommended option because that is the method used by the system to display the toolpath (orange lines) and cut part rendered images. Regardless of the setting you choose, all toolpath drawing and cut part rendering display as tool center.

Tool Center

The toolpath is rendered on-screen this way regardless of which CRC type option is selected. With this option, the toolpath is calculated to the center of the tool including any stock settings. When using Tool Center, the offset in the CRC register at the control should be the difference between the tip radius of the actual tool used and the tip radius of the tool

programmed in the system. If the tools are identical, the CRC offset number should be zero. If the actual tool is smaller, you can use a negative value.

Tool Edge

With this option the toolpath is calculated to the edge of the tool including any stock settings. CRC offset settings on the control should be set to the tool size.

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

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

> You cannot enter negative line lengths in Tool Edge CRC approach/exits. This means there is no way to have an arc only approach. With tool center you simply set the line length to "0". With tool edge you have to enter line length equal to -tool radius.

Finish Profile

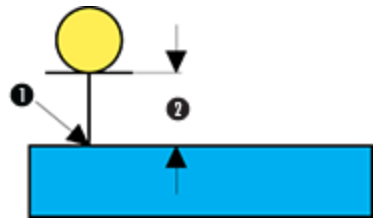
With this option, the toolpath is calculated to the finish geometry of the part and does not include stock. The G-code output is essentially a blueprint of your part. CRC offset settings on the control should be the tool size, plus any stock allowance. This option supports Rough Bore, Finish Bore, Contouring and Offset Pocketing operations. For pocketing and rough bore operations the Cut Width must be greater than the tool diameter otherwise tool center is output in the code. This option is meant to be used as a finish pass.

Last Used

This option keeps the CRC offset setting currently in use.

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

What exactly is meant by arc radius and line length in CRC? The arc radius is to tool centerline for all CRC types. Line length is the distance you want the tool edge to start away from the geometry start point.

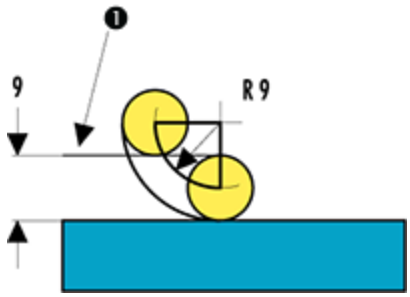


1. Machining Marker
2. Process dialog Line value

90 degree Line Approach

With Tool Center the line length and arc radius are tool centerline distances, meaning that a 90°, 9mm radius tool centerline arc will bring the tool edge down 9mm from the finish wall.

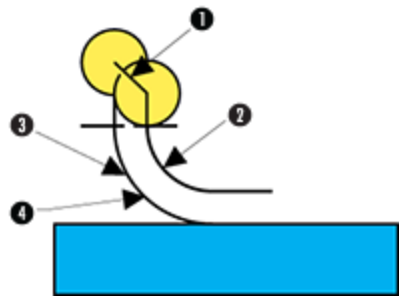
Tool Edge CRC defines the line length and arc radius in terms of edge position. If you specify "0" for the line length, the tool edge starts on the geometry, and a tool radius-long line will be output in the G-code for CRC to turn on. This results in "no move." If a line-arc entry is used, you have to visualize the equivalent finish geometry approach radius that will result from the arc radius specified. The line length is measured from this finish geometry arc start point.



1. Tool Edge to part = centerline radius

Tool Edge Entry

Please note that the toolpath for "0" length approach/exit lines are drawn incorrectly. The output toolpath appears to be correct. You can check the output toolpath by converting an operation to geometry. The orange toolpath line drawn is not the output toolpath, but rather a prediction of actual tool motion from turning CRC on/off. If you set the preferences to Tangent Entry Line to Arc, and Tool Edge, it is easy to see how these values work. The very first move is an angled line. The tool positions at a finish geometry equivalent position at the start point of the entry line. We assume G41 will be applied on the first move, and the orange line angles over to the start point of the centerline toolpath. This stays centerline until CRC off on the last line.



1. Turn on CRC move
2. Entry radius
3. Resulting Tool edge arc
4. Finish geometry to G-code

Tool Edge G41 start

Holder Class Menu Items
This item allows you to select which mill holders show up in the Document Control dialog. Most users will not have all classes of back ends available, so this item helps you minimize an otherwise long list of items.

Ignore Lathe Max Feed Moves
When this checkbox is selected, no turning moves will use maximum feedrate. Instead, any moves programmed for Max feedrate will continue at the feedrate immediately preceding. This is normally Contour Feed, unless the preceding utility marker was changed — to User, for example. Please note that if your Post Processor was written before this option was available, you will need to get it updated.

Legacy Tool Edge CRC Entry/Exit
Check this option to enable output of values along the Entry and Exit Angle in G-code, but the motion will deviate by a tool radius. The default is unchecked, where the Entry/Exit motion displayed will match the Entry/Exit angle, but the G-code will show values that differ by a tool radius.

Enable Background Material Calculation
Check this box to run 3D Material-Only calculations in the background, which is faster but takes more memory.

Uncheck it to run these calculations in the foreground, which is slower but provides more control.

With 3D Material Only, material can be shared across different operation types (such as Milling and Turning operations) or different CS's, and 3D milling can use rest material. It also provides Machine Sim with a more accurate picture. When a process dialog's **Solids** tab is bolded, it can use 3D Material Only. The DCD lets you set a global tolerancing preference for 3D Material-Only: Tighter tolerances mean better accuracy, but they incur larger model sizes.

Solid to Facet Body Conversion Tolerance
This value (default: `0.1` mm, `0.005` inch) controls the tolerance for automatic conversions from geometric solids (b-rep bodies) to facet bodies. This value should be used only for CAD operations like those in the Facet Body Solid Modeling Palette, not for machining or display (which have their own separate faceting tolerances).

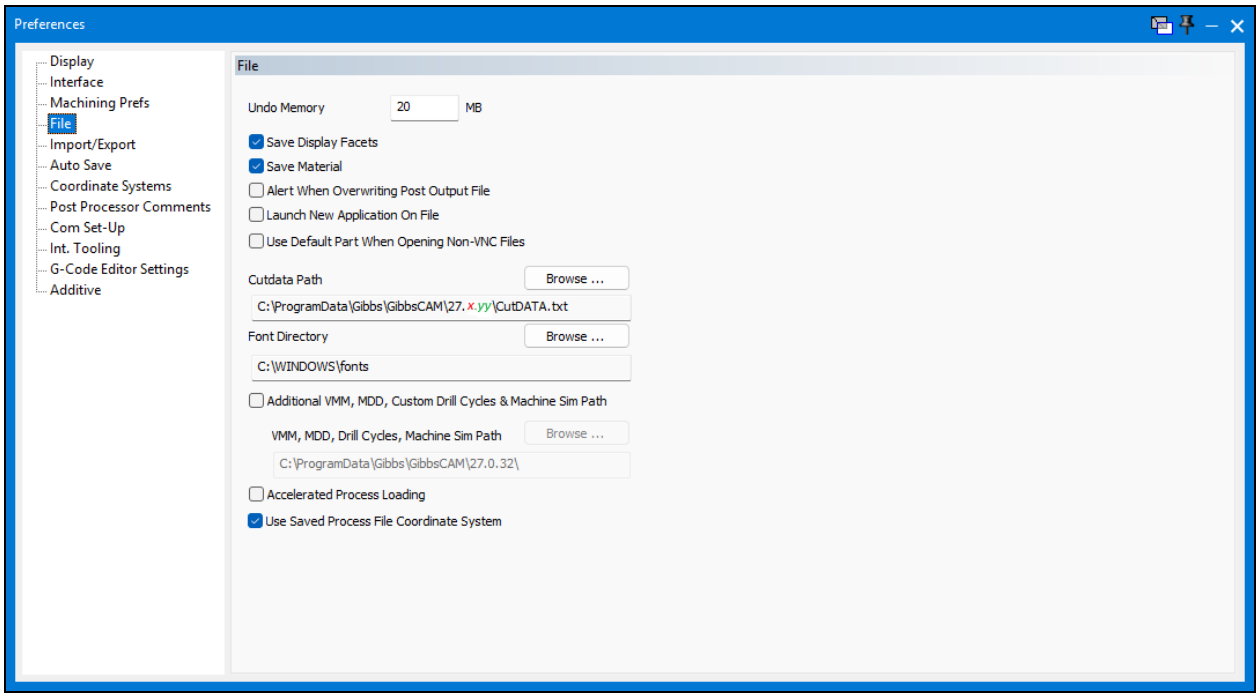
Hole Wizard
The Hole Wizard section contains buttons to access the preferences for Hole Data. These preference settings control the behavior of the Hole Wizard. In addition a predefined list of bolts and tap definitions can be accessed here. The Bolt Table and Tap Table dialogs are spreadsheets that contains standard data and user entered definitions of bolts and tap holes. These items are fully described in the *Features* guide.

Hole Manager
Opens a dialog which enables setting of preferences for Hole Manager. This is fully described in the *Features* guide.

5-Axis Preferences
Opens a dialog which allows you to set caching parameters for 5-Axis toolpath calculation. Caching uses additional space to speed up Redo performance during a session.

For example, if 5-Axis toolpath has been created during a session, then enabling caching means that a Redo of that toolpath will recalculate only the changed data instead of recalculating all data. For complicated toolpath and a location with fast I/O, such as an SSD (solid-state drive) or EFD (enterprise flash drive), this can significantly improve performance.

File Preferences



The items found in the File tab determine how GibbsCAM reads and writes items in files.

Undo Memory

This value determines the amount of memory space in megabytes to be reserved for performing Undo functionality.

Save Display Facets

Check this box to save facet information along with the part file. Saving without facets creates a smaller file that requires more time to regenerate display facets when opened

With 3D Material Only, material can be shared across different operation types (such as Milling and Turning operations) or different CS's, and 3D milling can use rest material. It also provides Machine Sim with a more accurate picture. When a process dialog's **Solids** tab is bolded, it can use 3D Material Only. The DCD lets you set a global tolerancing preference for 3D Material-Only: Tighter tolerances mean better accuracy, but they incur larger model sizes. .

Save Size

This preference lets you control the size of the files that have solids. The files can be saved as a Large or Minimal (without facets) file. Saving without facets creates a smaller file, but will require more time to regenerate the facets when the file is opened. On slow systems, this regeneration may be quite long, depending on the size and number of solids.

Alert When Overwriting Post Output File

This option will present an alert if you overwrite an existing NCF file when generating a post. This option will also open a Save dialog, allowing you to select a different name for the new post, protecting the older file.

Launch New Application on File

This option dictates how the system will respond when a VNC file is opened. If Launch New Application on File is selected, a new instance of GibbsCAM will open with the selected file. If this option is not selected, the file will be opened in the current instance of GibbsCAM. If a file is already open, you will be prompted to save the open file if necessary.

Use Default Part When Opening Non-VNC Files

Enables the MDD or Machine Type Default Part to be loaded before importing files with the Open command. If unchecked, Open imports the file into an empty part.

Cutdata Path

The system looks for the Materials Database file in the path and filename specified here. The default path is `C:\ProgramData\Gibbs\GibbsCAM\<version>\`.

- If you have **not** purchased CutDATA, the Materials Database file has the default name `Material.txt` if empty, or else, if it contains custom data, `MATERIAL.txt`.
- If you **have** purchased CutDATA, the Materials Database file has the default name `CutDATA.txt`.

In either case, you might want to name the file something else *(not recommended)*, or you might want to keep the file in a more central or convenient location such as the My Documents folder or the desktop. Clicking the Browse button lets you define where the system should look for your material database file.

Font Directory

This preference allows you to manually set the font directory that will be used in the Text Creation dialog. Some users may designate a different folder for engraving fonts to make selecting a font easier than sifting through a listing of all available fonts on a system. The system automatically determines the location of the operating system's Fonts folder; for most users, therefore, this preference may be ignored. However, if the fonts that will be used for engraving are not kept in the default system fonts folder (`C:\WINDOWS\Fonts`), this preference will be necessary. Simply highlight the target folder that has the engraving fonts and click on OK. Please note that only TrueType fonts can be used with the system's Text Creation function.

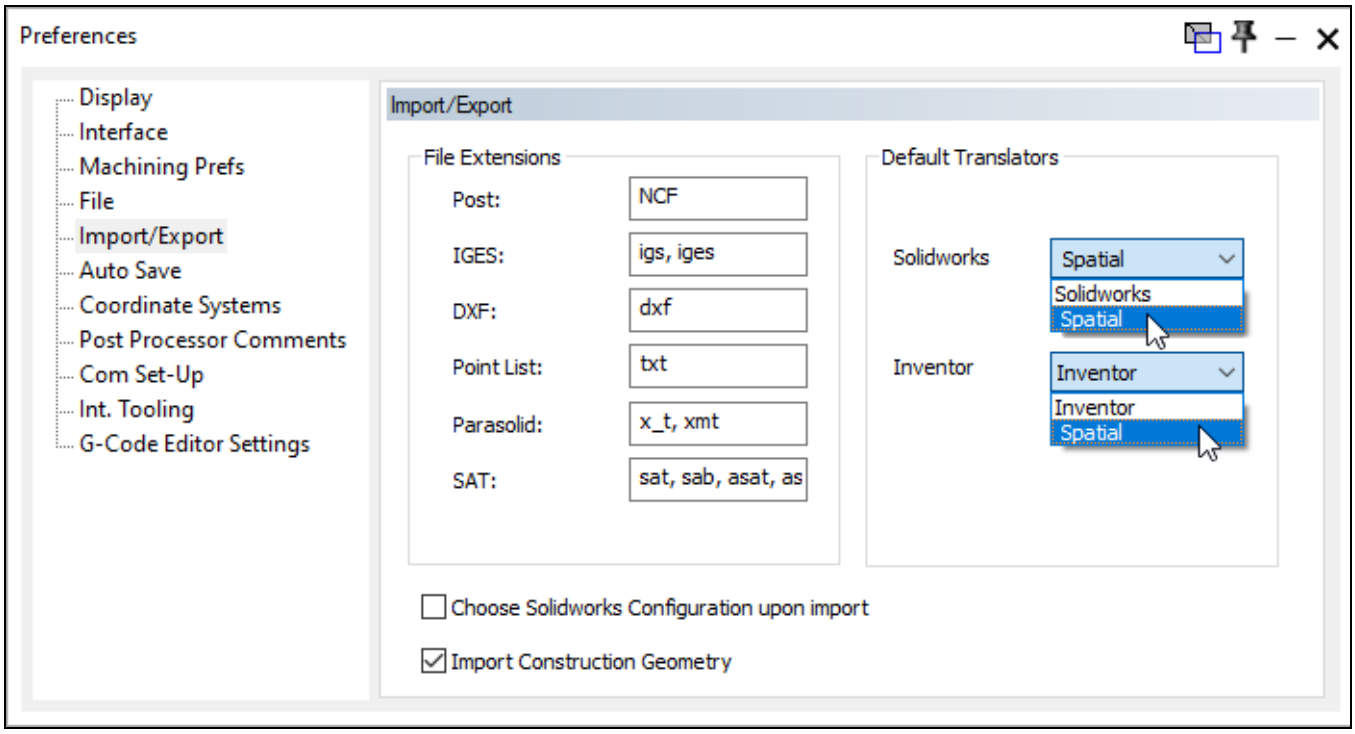
Additional VMM, MDD & MachineSim Path

This preference allows you to manually set the directory where the system looks for VMMs, MDDs, and Machine Sim models.

Accelerated Process Loading

Significantly increases system performance for loading process lists and tool lists, using additional memory.

Import/Export Preferences



The items found in the Import/Export tab affect how data is translated to and from GibbsCAM.

File Extensions

These preferences set the extensions that will be added to different types of files that are created by the system:

- The extension associated with Post is added to the text file generated by the system when a part file is post processed. The default extension is NCF.
- The extensions associated with IGES, DXF, and Point List are used when importing and exporting geometry.
- The extensions associated with Parasolid, SAT, CATIA V4, and VDA are used for importing and exporting solids files.

Each entry in the dialog may contain more than one extension for a type of file. For example, Parasolid files might have the extension x_t or xmt, depending on the application that saved the file. Multiple entries must be separated by a comma and a space.

Default Translators

This section allows you to specify which import/export option will be used by default on operations like File > Open or drag-and-drop, instructing the system which translator to use for SolidWorks (choices are SolidWorks and Spatial) and Inventor (choices are Inventor and Spatial). For more information, see the *Data Exchange* guide.

Choose Solidworks Configuration upon import

A SolidWorks "configuration" allows the creation of multiple variations of a part or assembly within a SolidWorks file. If this box checked, a dialog will appear during import to enable selection of the configuration.

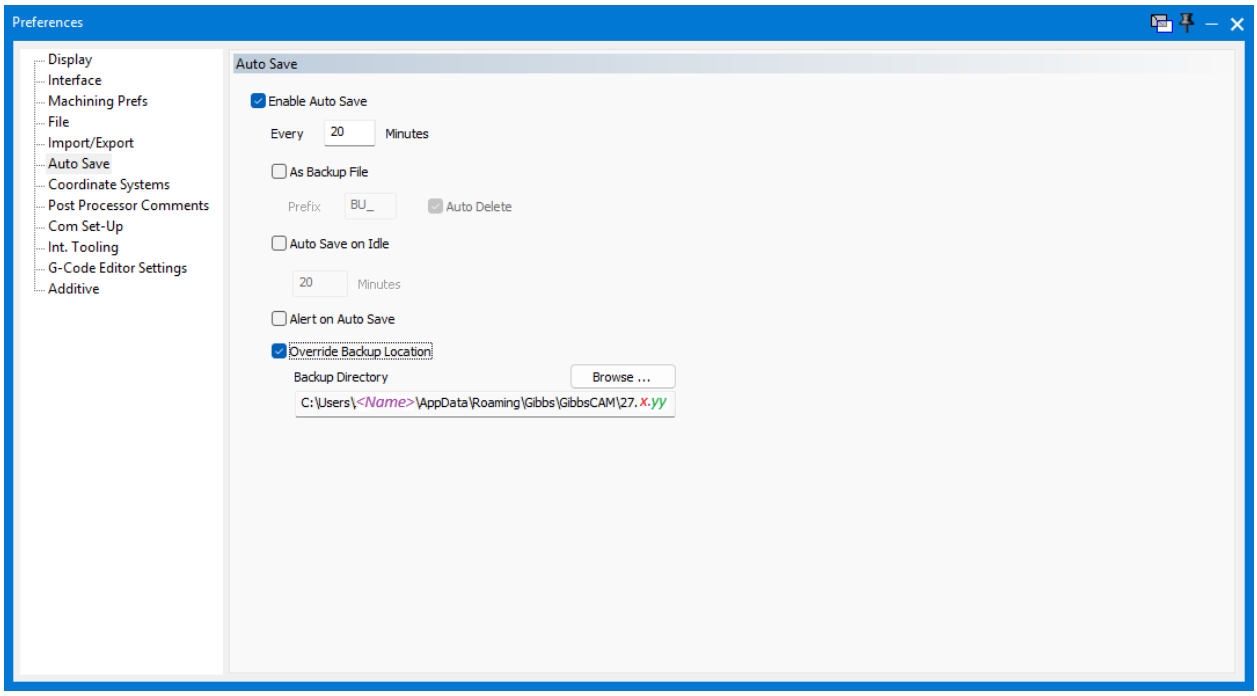
If this box is unchecked, the active configuration will be used.

Import Construction Geomeetry

If this box checked, construction geometry will be imported along with the model. If this box is unchecked, no construction geometry is imported.

Auto Save Preferences

Some items might or might not appear depending on the current license, installed options, and machine type. For example, GibbsCAM Viewer offers a subset of capabilities provided by GibbsCAM Industrial Edition.

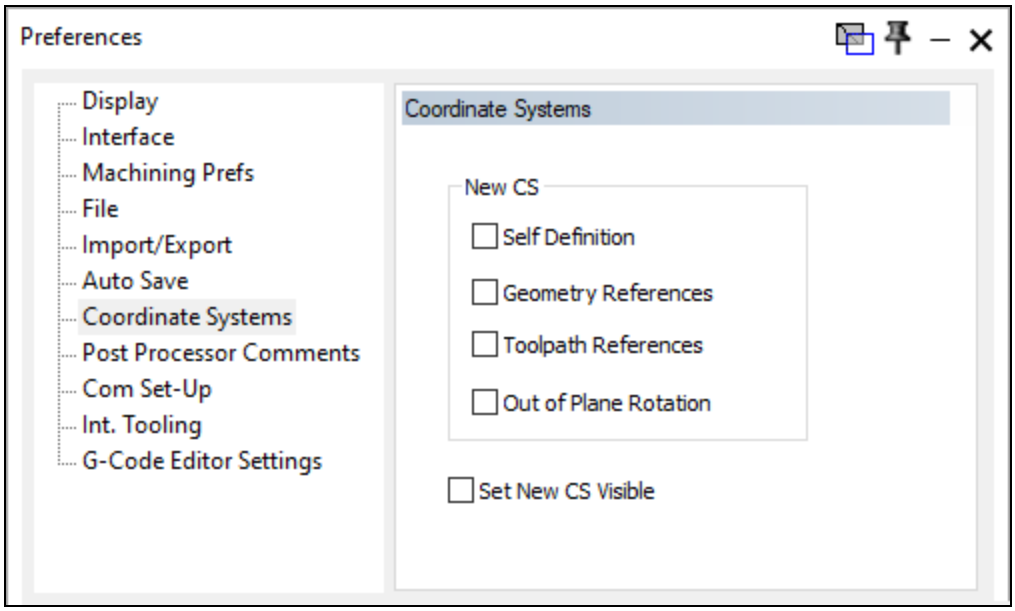


GibbsCAM has the option of automatically saving the file you are working on. You should be aware that saving will purge any available Undos and Redos.

By default, auto-saved part files are saved in the same folder where they were opened. You can override this default by selecting Override Backup Location and specifying a backup directory.

You have the option of saving As Backup file. Selecting this option has the advantage that you can continue working on your current file as normal, with redo and undo available. The backup file created is a temporary file that can be automatically deleted when you close GibbsCAM (although you can override the automatic deletion if you want). The backup file is used only if the software crashes before you close GibbsCAM.

Coordinate Systems Preferences



The Coordinate Systems tab allows users to set the guidelines for the automatic creation of new coordinate systems. It is recommended that beginning users check all of the items in the New CS section. This will result in more coordinate systems being created but geometry will not be modified.

These preferences affect the associativity between coordinate systems, geometry and toolpaths. When these items are all unchecked, geometry and toolpaths may change their position to reflect modifications made to the coordinate system on which they are based. In some cases, this may not be a desired effect, so these preferences have been included to give the user control over CS associativity.

Self Definition

This has an effect when modifying a CS and geometry is selected.

- When this item is checked the system will automatically create a new coordinate system rather than updating the current CS. This will only occur if geometry was selected to modify the CS and it is assigned to the CS being modified. The result is that the geometry selected for the modification will not change its location. The geometry will still be based on the coordinate system in which it was created rather than having its location in 3D space moved to reflect the new CS modification.
- If this item is unchecked, any modifications to a CS will result in geometry in the CS moving with the CS.

Geometry References

This has an effect when modifying a CS that contains geometry.

- When this item is checked, if a modification is made to a CS that was used for existing geometry, the system will create a new coordinate system rather than modify the current CS, which would result in moving the geometry.

- If this item is unchecked, any modifications to a CS will result in geometry in the CS moving with the CS.

Toolpath References

This has an effect when modifying a CS that was used for machining.

- When this item is checked, if a modification is made to a CS that was used as the Machining CS for an operation, the system will create a new coordinate system rather than adjust the current coordinate system. This prevents the changing of the position of toolpath.
- If this item is unchecked, any modifications to a CS that was used for existing toolpath will result in modified toolpath.

Out of Plane Rotation

This has an effect when modifying a CS that was used for machining.

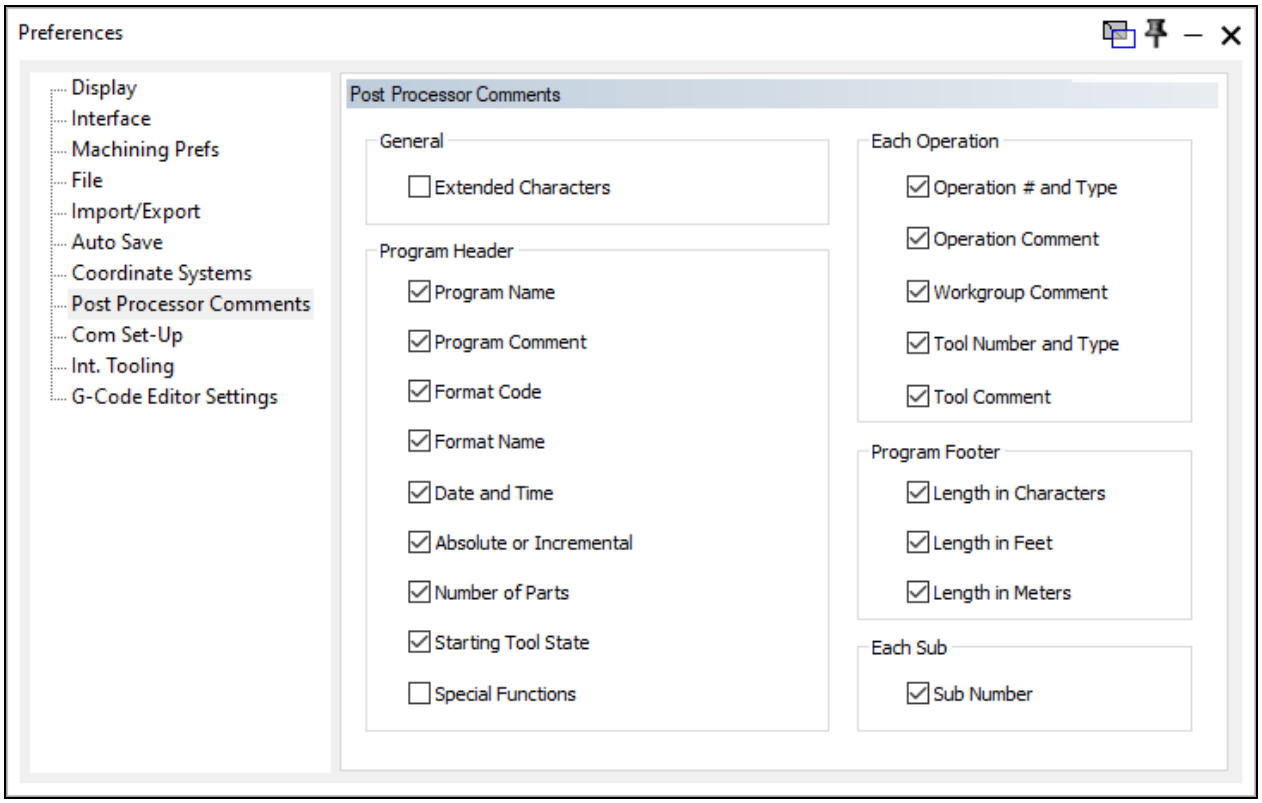
- When this item is checked, the system will automatically create a new coordinate system anytime geometry is rotated out of the plane in which it was based. Rotating out of plane would be rotating geometry along its horizontal or vertical axis, as opposed to about its depth axis.
- If this item is unchecked, geometry may be rotated and the CS will not be modified.

Set New CS Visible

When this is checked, the CS Frame Indicator will be displayed on the screen for any CS that is automatically created by the system based on these preferences.

Post Processor Comments

Some items might or might not appear depending on the current license, installed options, and machine type. For example, GibbsCAM Viewer offers a subset of capabilities provided by GibbsCAM Industrial Edition.



The Post Processor Comments preferences allow the user to specify comment data that is output in the NCF file generated by the system. If an item is checked, that data will be output. At the end of this section is a simple program that shows most of these options in use.

Extended Characters
Selecting this option will allow characters beyond the ASCII 1-128 character set to be output in the post. Your machine should support non-English character sets to use this option.

Program Name
This item will output the name of the part file at the beginning of the program.

Program Comment
This item will output any comment entered in the Document Control dialog “Comment” section. The comment will be placed in the program header.

Format Code
This option will output the ID number of the post processor that was used to generate the program. The ID number will be placed in the program header.

Format Name
This option will output the exact name of the post processor that was used to generate the program. The name will be placed in the program header.

Date and Time
This option will place the date and time the program was created in the header.

Absolute or Incremental
The program will include in the header whether the output is absolute or incremental.

Number of Parts
This outputs the number of parts being posted in the program header.

Starting Tool State
This option outputs whether the post processor expects the first tool to be in the spindle or tool changer.

Special Functions
By default this option is off. When active, the post outputs an explanation of Utility Data commands that are available with this post. This is potentially a significant amount of text.

Operation Number and Type
With each new operation in the program the operation number and type will be output.

Operation Comment
This option will output any comment associated with the operation at its start in the program.

Workgroup Comment
This option outputs the workgroup name at the start of each operation.

Tool Number and Type
This option outputs the tool number and type at the start of each operation.

Tool Comment
If there is a comment associated with a tool, it will be output at the start of the operation.

Length in Characters
This option will output the program length in characters in the program footer.

Length in Feet
This option will output the program length in feet in the program footer.

Length in Meters
This option will output the program length in meters in the program footer.

Sub Number
In operations that have sub-routines, the sub-routine number will be output as a comment.

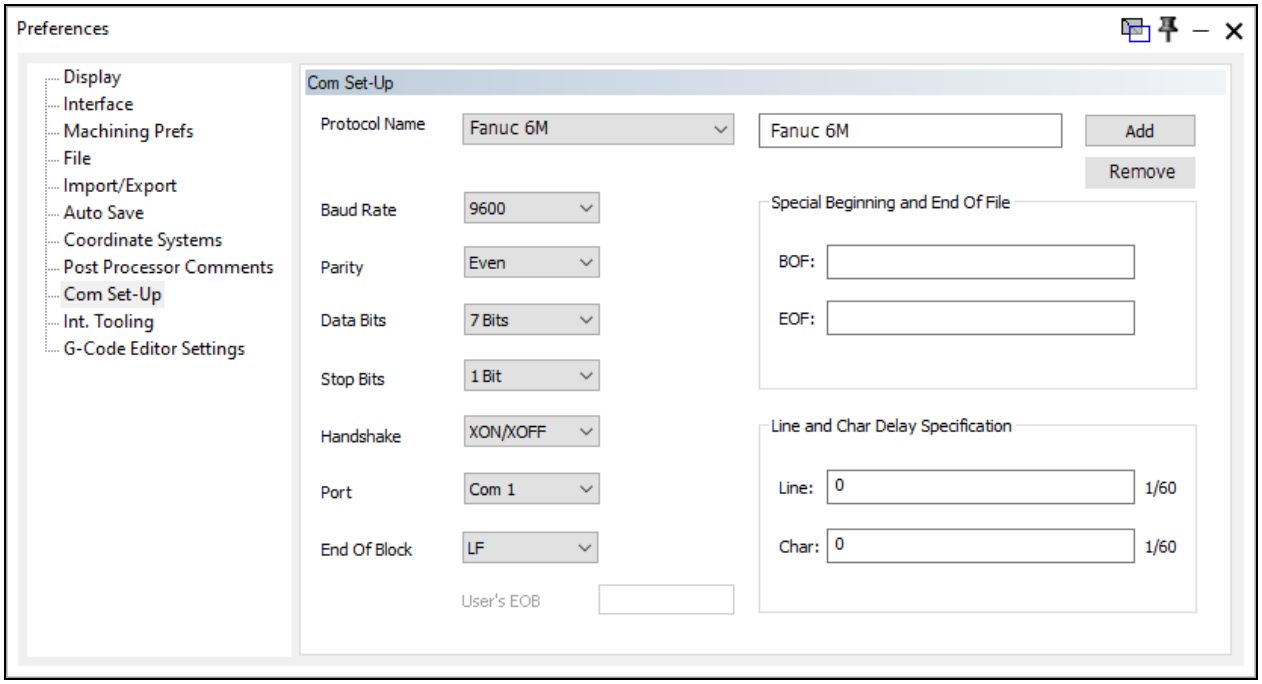
```
%
O1( PROGRAM: POSTEXAMPLE.NCF )                    Program Name
( THIS IS FROM THE DOCUMENT CONTROL DIALOG. )     Program Comment
( FORMAT: FANUC 6M [VG] M001.19M.PST )            Format Name
( 3/21/06 AT 3:57 PM )                            Date and Time
( OUTPUT IN ABSOLUTE MILLIMETERS )                Absolute or Incremental
( PARTS PROGRAMMED: 1 )                           Number of Parts
( FIRST TOOL NOT IN SPINDLE )                     Starting Tool State
N1G17G40G80
```

```
N2T1
N3M6
( OPERATION 1: HOLES )                    Operation Number and Type
( DRILL A SINGLE 13MM HOLE )              Operation Comment
( DRILLING WORKGROUP )                    Workgroup Comment
( TOOL 1: 13. DRILL )                     Tool Number and Type
( 13MM DRILL )                            Tool Comment
N4G54
N5S3000M3
N6G90G0X20.Y20.
N7G43Z50.H1
N8M8
N9Z2.5
N10G81G99X20.Y20.Z-53.906R2.5F250.
N11G80G0Z2.5
N12M9
N13G91G28Z0.
N14M5
N15M30
%
( FILE LENGTH: 502 CHARACTERS )           Length in Characters
( FILE LENGTH: 4.46 FEET )                Length in Feet
( FILE LENGTH: 1.43 METERS )              Length in Meters
```

A simple program showing most of the available post options in use

Communication Setup Preferences

Some items might or might not appear depending on the current license, installed options, and machine type. For example, GibbsCAM Viewer offers a subset of capabilities provided by GibbsCAM Industrial Edition.

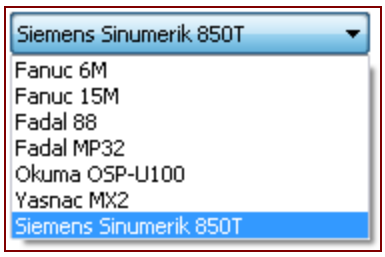


To send a post file to a CNC machine, the parameters for data transfer must be set up in the File > Preferences Com Set-up dialog. This dialog is used to set up two-way communications for transmitting data between a control and workstation. Different controls have different protocols or parameters. Refer to the machine control's manual for the protocol specifications.

Adding a Protocol

To add a new communication protocol, enter the name of the new protocol, change the settings to match those of the CNC machine, and click the Add button. The new protocol will appear in the list.

Changing a Protocol:

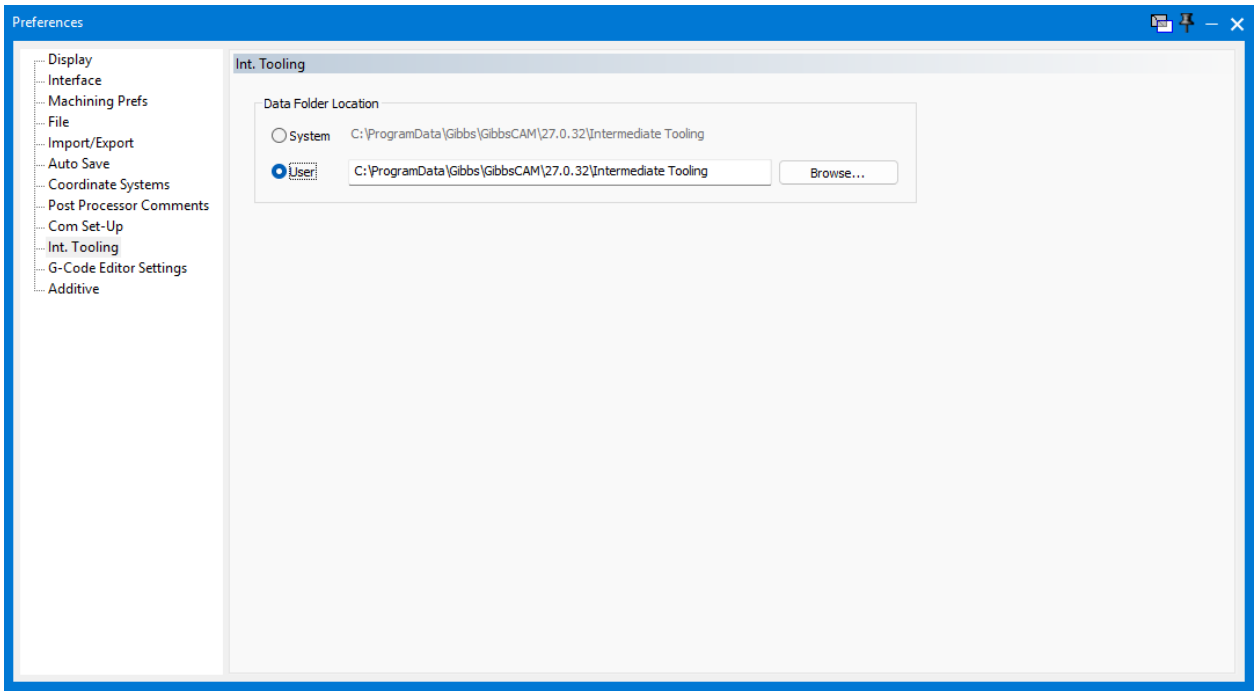


To change protocol information, select the protocol from the list and change the information. The changes are automatically saved when the dialog is closed or when a different protocol is selected in the list.

Removing a Protocol:

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

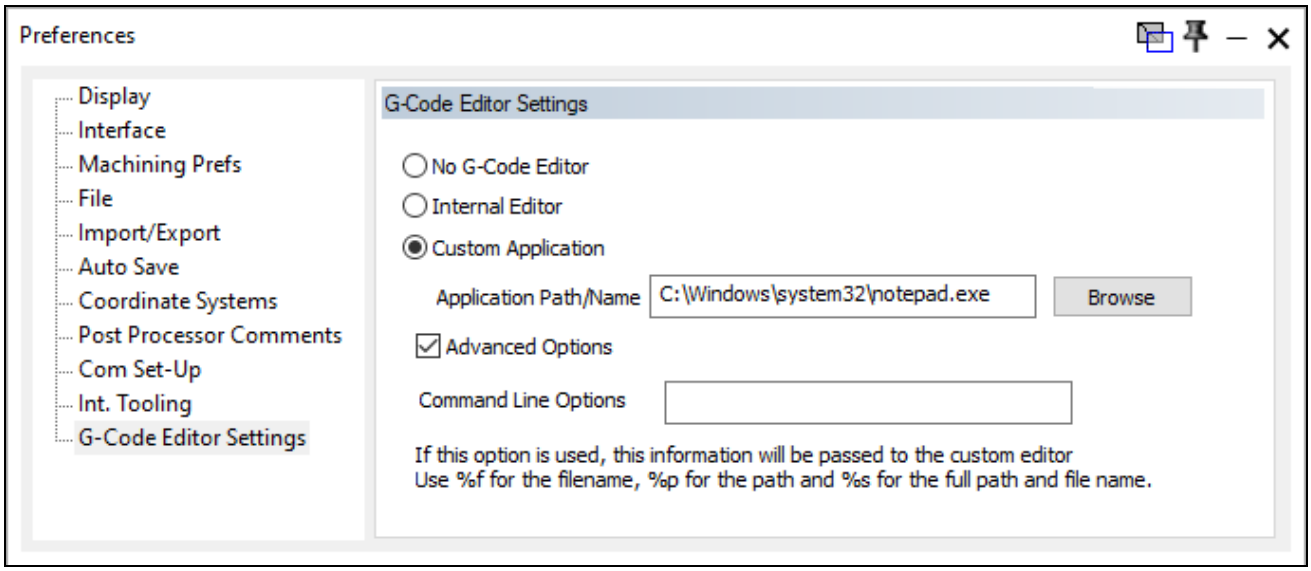
Intermediate Tooling Preferences



This lets you set the default folder used for Intermediate Tooling.
You can set the default view to either Quick View (static image) or Dynamic View:

- Quick View is suitable for slow systems with underpowered graphics.
- Dynamic View is recommended for most systems, because you can pan, rotate, and zoom the preview using many of the same mouse movements and keyboard shortcuts that you use in the workspace, like CTRL-drag to pan, center-mousebutton-drag to rotate, marquee to zoom in, CTRL+U to unzoom, and so forth.

G-Code Editor Settings



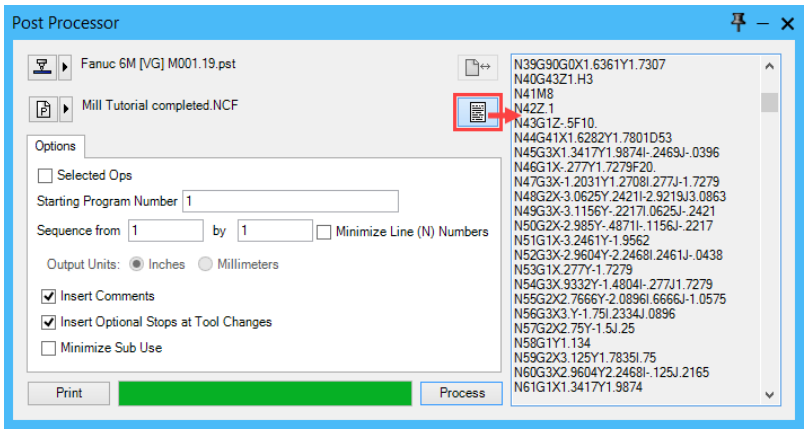
This lets you select an application for displaying and editing post files.

Editor Choices

When a post file is processed, the system will automatically launch the application of your choice and display the posted output.

No G-Code Editor

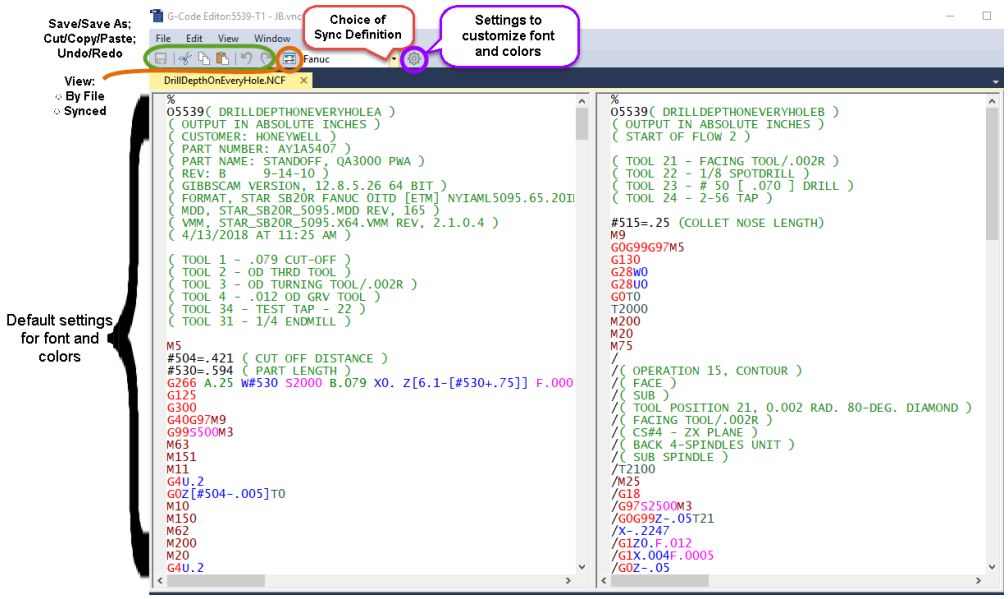
You can choose to disable the editor, in which case you will still be able to look at the output in the post Processor dialog window.



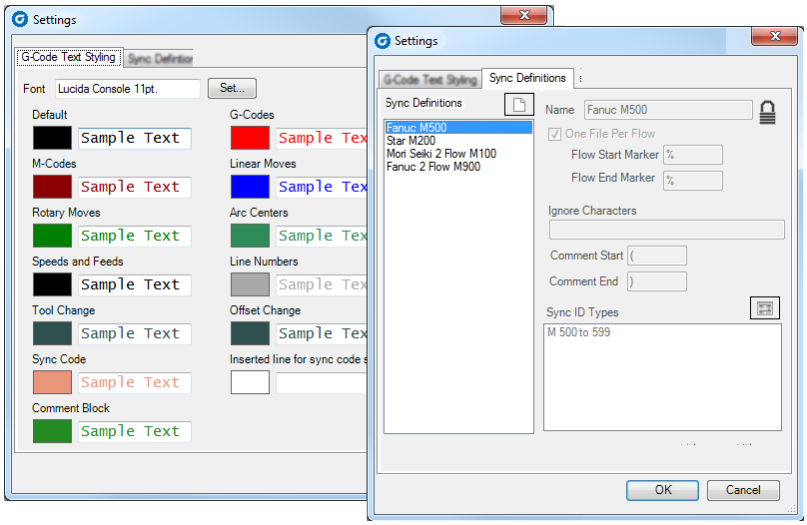
Internal Editor

The GibbsCAM built-in G-Code editor provides the following features:

- **Editing and file capabilities**, such as Cut/Copy/Paste, Undo/Redo, and Save/Save As.
- **Context colors** that distinguish at a glance items like code, code labels, and parameters within individual G-codes and M-codes.
- The ability to edit **multiple subs**. Each sub occupies its own tab in the window.
- When editing **multiple flows**, you can choose either independent scrolling, where each flow is independent of all others, or **synched scrolling**, where each G-code stream is padded as needed so that separate flows remain aligned according to their syncs.



- **Customization:** You can set colors and font however you like, and you can edit and create sync definitions.



If you require the G-Code editor to be permanently displayed, open it from the Plug-Ins menu using

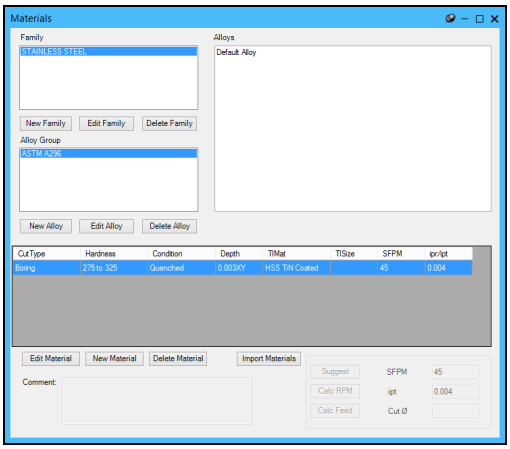
Open G-Code Editor. If you have your own editor, this can also be opened from the Plug-Ins menu using External G-Code Editor.

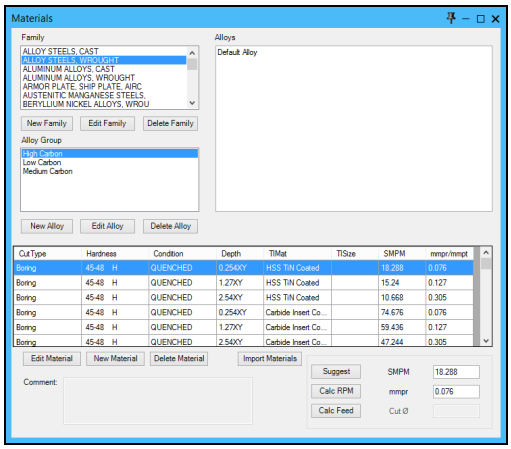
Custom Application

Any application that can open and read ASCII text files such as Notepad or Word can be used as your custom editor. Click the Browse button to locate the custom editor you want to choose.

You can select the Advanced Options checkbox if you want to use the ability to pass variables to the custom editor. When this option is enabled, `%f` passes the filename; `%p` passes the path down to the folder that contains the file; and `%s` passes the path and filename.

Materials



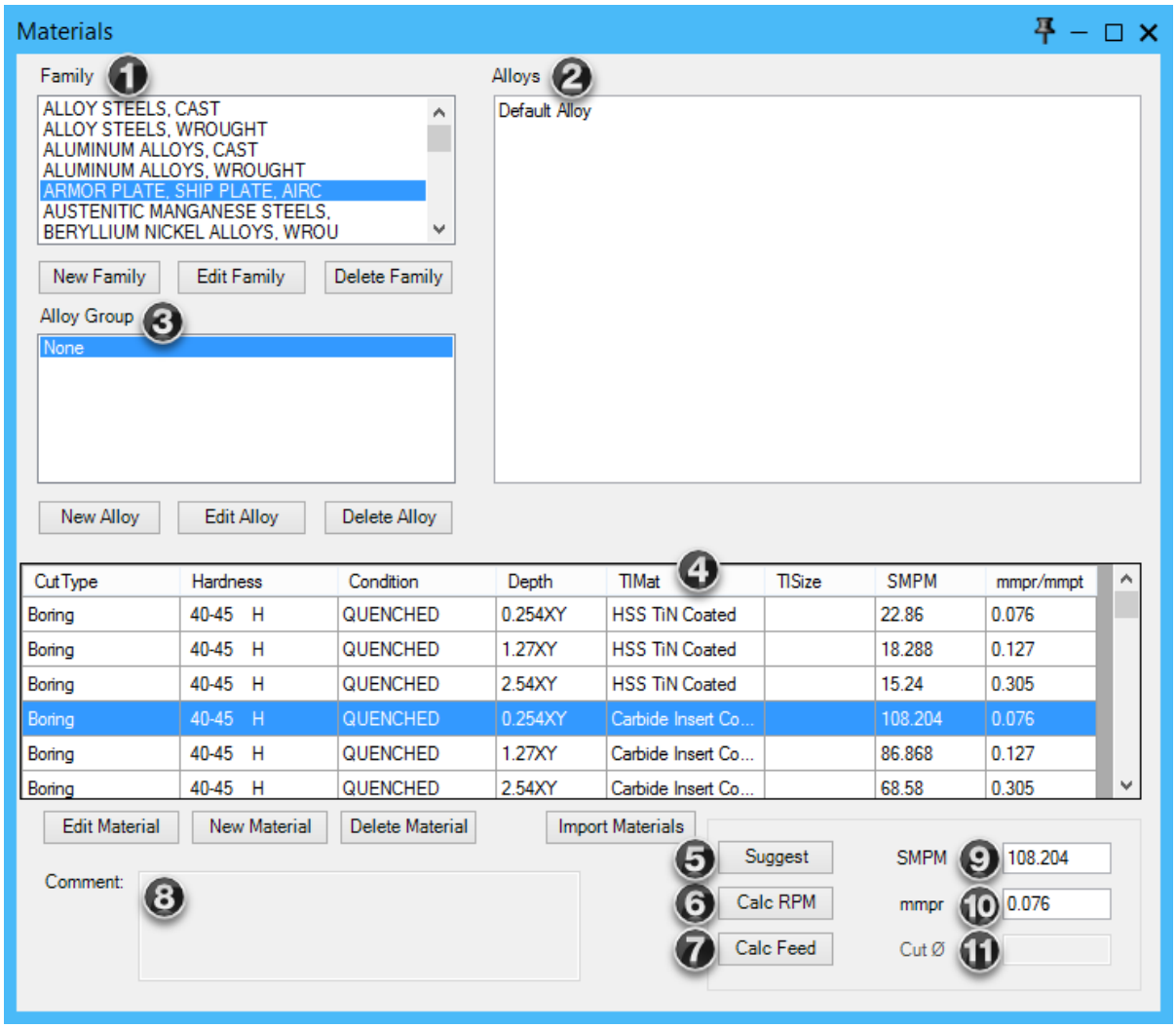


The Material Database with and without CutDATA

Materials database is available from the File menu Materials. The Materials Database is used for storing and quickly retrieving feeds and speeds recommendations for various types of materials. All Materials Database information must be entered by the user unless the CutDATA™ material library has been purchased. To access the database, you can also click on the Material button in any of the process dialogs.

If you have **not** purchased CutDATA, the Materials Database file has the default name `Material.txt` if empty, or else, if it contains custom data, `MATERIAL.txt`. If you **have** purchased CutDATA, the Materials Database file has the default name `CutDATA.txt`.

The database is set up in a hierarchical format. A Family can contain more than one Alloy group and is at the top of the hierarchy. Within the Alloy group(s) you can then specify materials. Materials are sorted into hardness ranges and cut types. When you select an alloy group the Alloys text field becomes active, where all of alloys that fall into the alloy group can be listed.



Materials Dialog	
1. Family 2. Alloys textbox 3. Alloy Group(s) 4. Material data list 5. Suggest button	6. Calculate RPM button 7. Calculate Feed Button 8. Comment 9. SFPM (SMPM) 10. ipr (mmpr) 11. Cut diameter (Turning)

Family

The dialog above shows some of the different families that come with the CutDATA™ database. When a family is selected, the Alloy Group list will fill up with the groups that are part of this family.

Alloy Group:

An alloy group contains a list of all the alloys that fall into a material family and can be machined with the same parameters. When an existing alloy group is selected, the material list will fill up with materials.

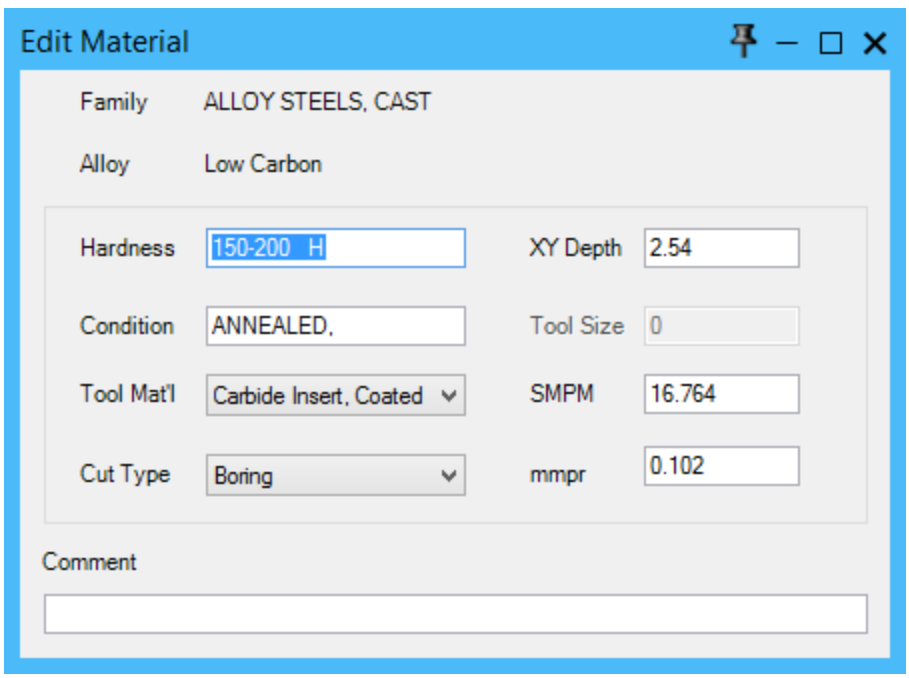
Alloys
This is a large text edit box which if using CutDATA, already contains a list all of the alloys that are in the alloy group. The information in this box is available to edit when an alloy is selected.

Material Data List
The materials in this list are grouped together by cut type, hardness, and tool material.

Comment
If a comment has been entered, it will be displayed here.

Edit Material
To edit material information:

1. Select a Family, Alloy Group and Cut Type.
2. Click the material you wish to edit.
3. Type the information required.
4. Close the dialog. The information is updated in the Materials list.



The following settings are only available from the Edit Material or New Material dialogs.

Hardness
The text in this field is used to set up different hardness ranges that can all be machined with the same settings for a specific material.

Condition
This is the condition of the material. It is related to the hardness value. It is only used for reference and will not affect any calculations.

XY Depth
This box is used to enter the depth that the tool will be buried into the material. The axes change depending on the cut type selected.

Tool Size
This is the diameter of the tool.

Tool Material
This dropdown menu is a list of the available materials for tools.

Cut Type
This dropdown menu is a list of the different cut types available in the database.

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

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

Comment
This box allows an additional comment to be displayed when the material is selected in the Materials dialog.

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

Import Material
Click this button to open a dialog where you can navigate to and select a tab-delimited `*.txt` file that defines a material.

Importing a Material into the Database

The Import Material lets you import custom material data can be imported into the Material database. This option can be used whether you have purchased the CutDATA Materials database or not. This function lets you define new material types and cutting parameters in an external file and perform a bulk import into the database, either adding to the existing data or creating an entirely new database of your own. The data may be created in any application that can save or export a tab-delimited text file, such as Microsoft Excel or Notepad. A spreadsheet application is recommended for its ability to clearly view your data.

Click the Import Material button, navigate to the correct folder, and open the file that contains your custom data.

By default the Materials Database is stored in the GibbsCAM global data folder:
`C:\ProgramData\Gibbs\GibbsCAM\<version>`

The default name of a CutDATA file is `CutDATA.txt`; the default name of an empty Materials Database file is Material.txt; the default name of a customized Materials Database file is MATERIAL.txt; the all-uppercase letters indicate that custom data is present. If the file does not exist in this folder or a location you specify in the File > Preferences > File I/O, then an empty database file is automatically created when launching the application. Regardless, any imported material will be added to the currently specified file.

The import process only adds data, it will not overwrite any existing entries. It is therefore possible that you may have duplicate entries. The duplicates can be deleted using the Delete Material button.

Fourteen data categories are supported. Each new material entry must be on a single line and there must be a tab between each entry. All categories must be entered; if a material you are defining does not have an entry (typically a comment), then the category must be entered as "NULL" (without quotes). Each category has a particular type of data it can read. A "string" is text and a "real number" is any non-irrational number, e.g., -4, 0, 8, or 0.1215.

Category	Type of data	Example
Family	string	Alloy Steel, Cast
Alloy Group	string	Low Carbon
Alloy Comment	string	This is a low carbon steel alloy casting
Hardness	string	Over 50 HRC
Condition	string	Carburized &/or Quenched and Tempered
Tool Material	string †	Carbide Solid
Cut Type	string ‡	End Mill, Peripheral
Cut Depth	real number	75
Tool Size	real number	6
Surface Feed	real number	45
Inches per revolution or Millimeters per thread	real number	0.102
Use Comment	0 or 1	`0` for no comment or `1` for using a comment
Comment	string	`NULL` (if there is no comment)
Metric	0 or 1	`0` for inch or `1` for metric

† These strings must match strings in the current Material Database dialog. That means the Tool Material must be

HSS	HSS TiN Coated
Carbide Insert	Carbide Solid
Diamond	Other.

‡ These strings must match strings in the current Material Database dialog. That means the Cut Type must be

Boring	C. Bore	Drilling
End Mill, Peripheral	End Mill, Slot	FaceEdit Mill
Reaming	Spotface	Tapping
Turning	Cutoff	Thread.

Following is an example of a material entry as seen in Microsoft Word. The arrows represent tabs and the pilcrow (¶) is a return.

```
Alloy Steel, Cast→Low Carbon→ This is a low-carbon steel alloy
casting→Over 50 HRC →Carburized &/or Quenched and Tempered→Carbide
Solid→End Mill, Peripheral→75→6→45→0.102→0→NULL→1¶
```

- The data is not case-sensitive, but it is language-sensitive. In other words, if you are running a French or Spanish version of GibbsCAM, then the Tool Material and Cut Type must be localized.
- If an entry is not complete, it will be skipped; that is to say that if an entry has only 13 tabs, the entire entry will be skipped or if it has an empty field (it does not have "NULL") it will be skipped. If an entry is skipped, then the import process will move to the next valid entry and will continue from there.
- The importation process is immediate and cannot be interrupted.

Selecting Material from within a process dialog

Clicking the Material button from within a process opens the Materials Database. Select a material by clicking first the Family, then Alloy, then the material required. Once the material has been selected (it is now highlighted in blue) you can populate the Speed RPM, Entry, and Contour Feed in the process dialog from the data set up in the Materials Database, either by clicking the buttons in the process dialog or the Calc RPM and Calc Feed in the Materials dialog.

The following buttons are only active when the dialog is accessed through a process dialog.

Suggest

Clicking this button will tell the software to select the material in the list that most closely represents the cut condition of the open process dialog. First it will try to match the cut type. Then it will match hardness, depth of cut and tool material.

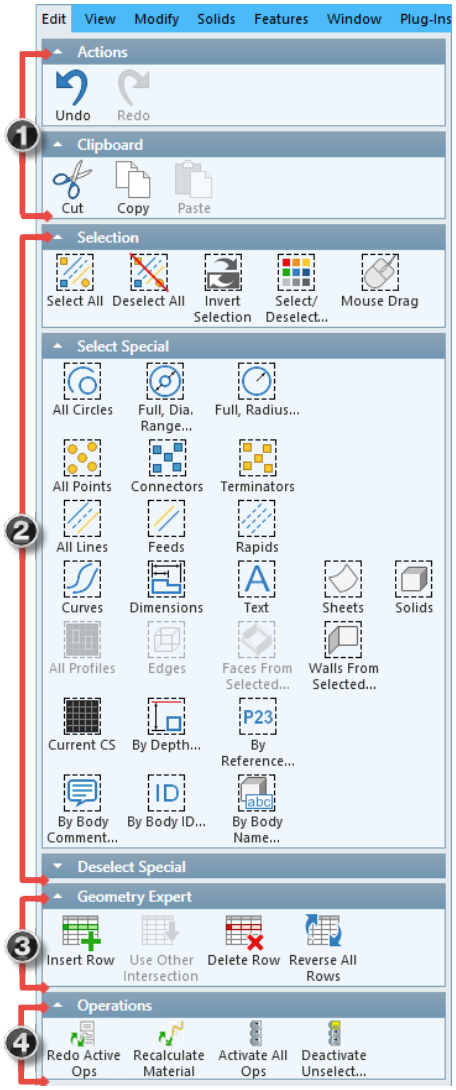
Calculate RPM

This button calculates an RPM value and places it into the process dialog.

Calculate Feed

This calculates a feed value and places it in the Feed textboxes in the process dialog.

The Edit Menu



The Edit menu contains options that enable you to adjust and select file contents.

1. The actions and clipboard sections are the standard Windows commands for removing and inserting text and features. You can set the amount of memory allocated to "undo/redo" in the File Preferences menu.
2. Select and deselect,
3. Geometry Expert. These four commands are used with Geometry Expert.
4. Edit Menu - Operations.

Clipboard

The Clipboard is used as a temporary storage space for geometry, solids, features, attributes (including PMI), text, tool tiles, and process tiles. You will find the clipboard from the Main menu > Edit > Clipboard. You can also use the standard Windows keyboard shortcuts.

Cut option (Ctrl+X) deletes whatever items are selected and places them on the Clipboard. Previous contents of the Clipboard, if any, are replaced by Cut.

Copy (Ctrl+C) is similar to Cut inasmuch as it places a copy of the item in the Clipboard, replacing the Clipboard's previous contents, but it does not delete the selected item.

Paste (Ctrl+V) is available only when the Clipboard contains something. It can do two things. If something is selected, replaces that selection with the contents of the Clipboard. If nothing is selected, it pastes the contents of the Clipboard in either the workspace (if the Clipboard contains geometry) or into the active text box (if the Clipboard contains text).

The Clipboard can only hold one selection at a time. This means that it can hold either geometry (with associated attributes, etc.), or text, but not both. If text is copied into a Clipboard holding geometry, the geometry in the Clipboard will be lost.

Cut and paste is an easy way to move geometry between Workgroups in the same part file. You can also move geometry, solids, tool tiles, and process tiles between parts. If you are Cutting and pasting items between GibbsCAM instances, they must be the same GibbsCAM version for the paste to work. Items are retained on the Clipboard even after GibbsCAM is closed and relaunched. If you copy Processes, the system will automatically recreate the tools that are associated with the Processes. Please note that you can only copy Workspace items (geometry and solids) or items in the Tool or Process list.

Select and deselect

Selection

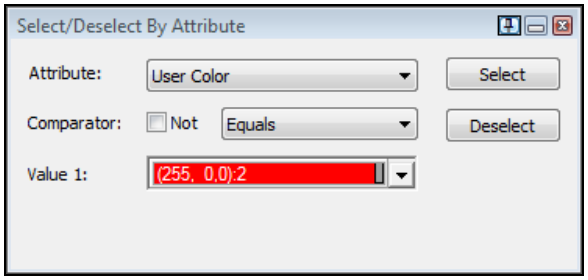
Select/ **Deselect All**

Select/deselect all available workspace items

Invert Selection

Selected items are deselected and all unselected items are now selected.

Select/Deselect by Attribute



This item opens a dialog box that lets you specify an attribute, comparator, and comparison value. When you click the Select button, all elements and bodies that match are added to the selection set; when you click the Deselect button, all elements and bodies that match are removed from the selection set. For information on attributes and comparisons, see the *Features* guide.

Mouse Drag Select and Mouse Drag Deselect

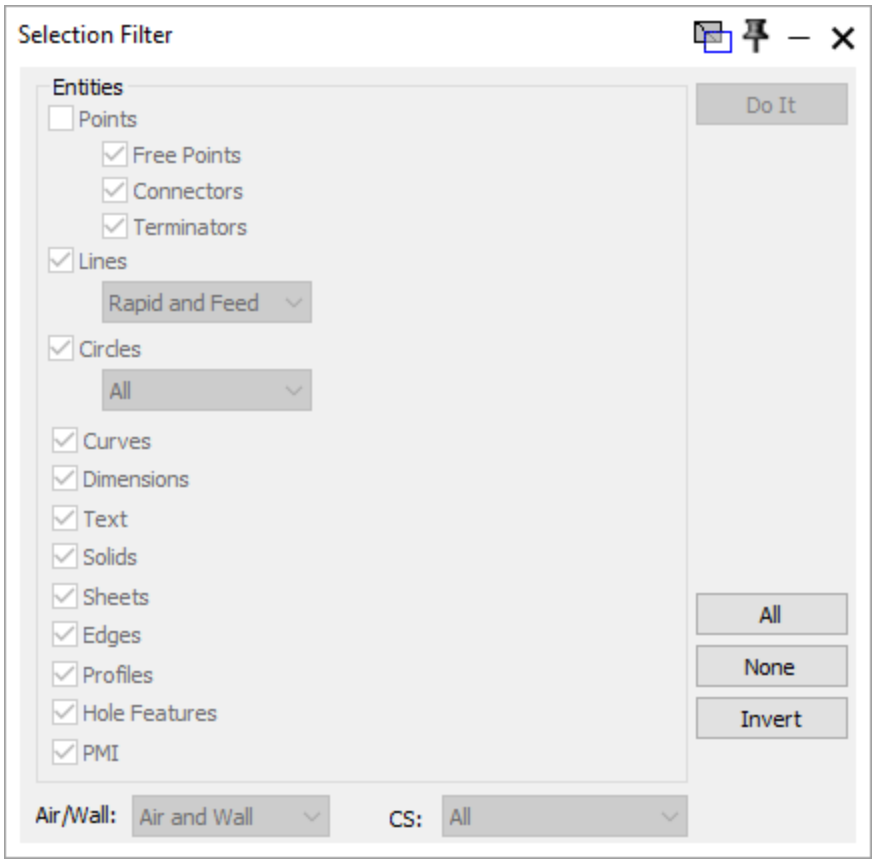
After you click , you can click-drag the cursor to form a rectangle to select all items contained in the rectangle. This is equivalent to holding down the `Shift` key when doing a click-drag.

After you click , you can click-drag the cursor to form a rectangle to deselect all selected items in the rectangle. This is equivalent to holding down the `Ctrl+Shift` keys when doing a click-drag.

Note that rectangles dragged left-to-right and right-to-left exhibit different behavior: One direction affects the subset of elements that are entirely contained within the rectangle, and the other direction affects the superset of elements that are even partly in the rectangle. You can set a preference specifying which behavior occurs for which direction: File > Preferences > Interface: Mouse drag selection limited to.

Filter Selections

Clicking opens the **Selection Filter** dialog box. This dialog lets you include and/or exclude items by entity type: Not just points, lines, etc., but also dimensions, text, solids, holes, PMI, and more. Because the results of selecting/deselecting items can be stacked, this is an extremely powerful tool, providing the finest control possible over a selection set.



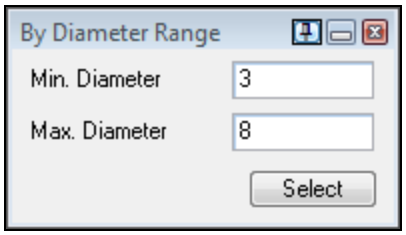
Select Special

Circles

There are three circle selection options. All Circles will select all circles in the current workgroup. Full, Radius and Full, Dia. Range will bring up the following dialog boxes.

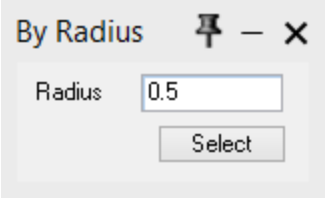
Full Diameter Range

This dialog allows you to select all full, unterminated circles in the current workgroup whose diameter is equal to or between the value range entered in the text boxes.



Full, Radius

This dialog allows you to select all full, unterminated circles in the current workgroup with a radius equal to the value entered in the text box.



Points

The Points selection options are as follows:

All Points will select all points in the current workgroup.

Connectors will select all points used to connect geometry in the current workgroup.

Terminators will select all points used to terminate open geometry in the current workgroup.

Lines

There are three options to select Lines.

All Lines will select all lines in the current workgroup.

Feeds will select all lines designated as “Feed” in the current workgroup.

Rapids will select all “Rapid” lines in the current workgroup. See the Modify menu section of this document for more information on Feed and Rapid geometry.

Curves

Selects all splines or curves in the current workgroup.

Dimensions

This will select all dimensioning annotations made in the current workgroup.

Text

Selects all textual annotations made in the current workgroup.

Sheets

Selects all sheets in the Workspace and open Body Bag.

Solids

Selects all bodies in the Workspace and Body Bag if it is open.

All Profiles

Any profile shapes will be selected.

Edges

The Edges item will select all edges of selected bodies and sheets that are in the Workspace. Any edge you do not want to select must be manually deselected.

Faces From Selected Profiles

Faces that are bound by the current profile will be selected.

Walls From Selected Edges

Any walls connected to selected edges will be selected.

Current CS

The Current CS item provides a quick way to select all geometry and bodies in the current coordinate system. Geometry or bodies defined in coordinate systems other than the currently active CS will not be selected.

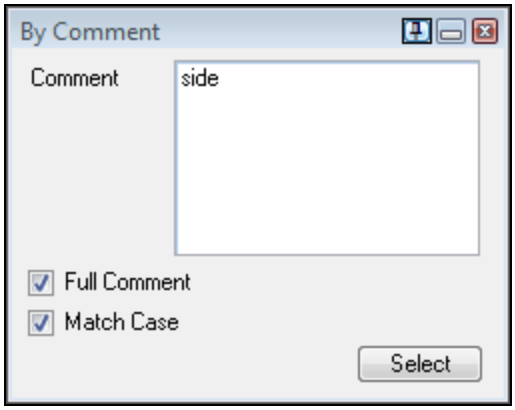
By Depth

To select elements By Depth, enter values for From and To (in other words, depth values) to define the maximum and minimum parameters for the selection. When the Select button is clicked any element that lies within the range (wholly or partially) will be selected. If a section of connected geometry falls within the selected range, only within-range elements will be selected. None of the connection points or other elements outside of the range will be selected.

By Reference

To select an element By Reference, enter the Reference number that corresponds to the desired selection. Reference numbers are assigned by the system to every geometric element created. They can be viewed by turning on Labels from the View menu. To make the selection, click the Select button.

By Body Comment



To select (or deselect) bodies by comment, simply enter the search criteria in the By Comment dialog and click on Select. This will select all bodies whose Comments field contains the search criteria.

A search can be very specific, such as a name given to the body, or the search may be very simple, such as selecting all bodies with the letter "s" in the comments. When Full Comment is selected, only bodies whose comments are exactly the same as what you entered will be selected. When Match Case is selected, capitalization of words is taken into account: even if "rounded" was entered and a solid or sheet contained "Rounded" as a comment, it would not be selected.

If the entry is left blank, all bodies with no comment will be selected.

By Body ID

To select a body using Select > By Body ID, enter an ID number that corresponds to the solid or sheet you want to select. ID numbers are assigned by the system to each body that is created or imported, and can be viewed by querying the properties of the body. For bodies in the Body Bag, if View > Show Solid ID is in effect, the body ID is appended to the body name.

By Body Name

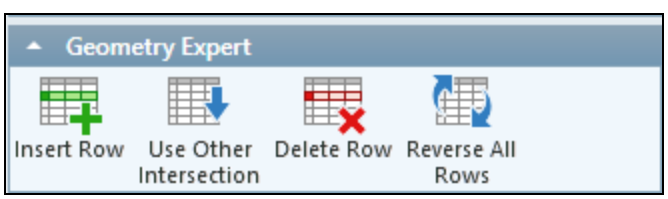
To select bodies (solids and sheets) By Body Name, you enter some data in the Name box to specify the selection. All bodies have names — either automatically generated (such as Sphere 35) or given by the user. When Full name is selected, only bodies whose names exactly match the text entry will be selected. When Match case is selected, capitalization of words is taken into account.

Example: If you were to enter the text "round" into the Name box, it would not match a body named "Rounded" if either checkbox is selected.

Deselect Special

The items contained in this sub-menu will deselect any geometry or bodies of the chosen type. The Deselect sub-menu items have the same criteria as the Select sub-menu (explained above).

Geometry Expert



Insert Row

This item will insert a blank row in the spreadsheet above the currently selected row. When an inserted row contains the necessary information and is entered, the system will recalculate the shape and attempt to incorporate the inserted feature into the existing shape, if possible.

Use Other Intersection

This item will toggle between Use Intersection #1 and Use Intersection #2 in the menu. This option enables the user to select different intersections between concurrent features in the spreadsheet. This option is available when the feature defined by the current row intersects the feature defined by the following row in two places. This item will change the point of intersection selected in a previous post targeting dialog to the other possible choice.

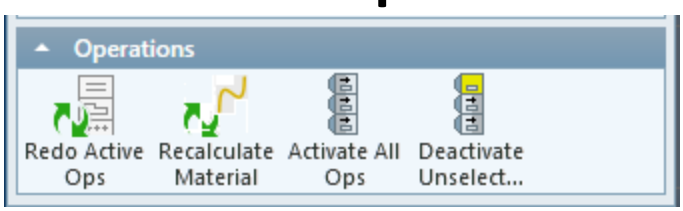
Delete Row

This item will delete the selected row from the spreadsheet. As soon as the row is deleted, the system will adjust the remaining shape to maintain the appropriate connections, if possible.

Reverse All Rows

This item is only active when the Geometry Expert spreadsheet is open. When selected, it will reverse the order of the rows in the spreadsheet which will change the direction of the shape as it is loaded in Geometry Expert. This is useful when loading a shape that was imported or created using the Free-Form CAD tools. When a shape that was not created using Geometry Expert is loaded into the spreadsheet, the system decides whether to go in a clockwise or counterclockwise direction when placing the features into the spreadsheet. Shape direction has the potential to affect the angle values of lines and the direction of arcs. There are times when the system will load the shape in the opposite direction than the one desired. When the shape has been loaded into the spreadsheet, select the Reverse all rows item to reverse the order of features. This item is only available when the spreadsheet contains feature rows.

Edit Menu - Operations



For more information on active and inactive ops, see “Operations List” on page 211.

Redo Active Ops

Recalculates the toolpath of all active operations.

Recalculate Material

Recalculate Material recalculates all data for Rest Material, including 3D Rest Material. This ensures that the model's data is up to date, but it can require a long time for a large 3D model with extensive Material-Only processes.

With 3D Material Only, material can be shared across different operation types (such as Milling and Turning operations) or different CS’s, and 3D milling can use rest material. It also provides Machine Sim with a more accurate picture. When a process dialog’s **Solids** tab is bolded, it can use 3D Material Only. The DCD lets you set a global tolerancing preference for 3D Material-Only: Tighter tolerances mean better accuracy, but they incur larger model sizes.

For more information, see Machining Preferences on page 43 and File Preferences on page 47

Activate All Ops

Changes all operations currently in a deactivated to an active state.

Deactivate Unselected Ops

If this icon is clicked, all operations that are selected (yellow highlighting) will remain active (or inactive if selected and is in an inactive). Unselected ops will change to an inactive state.

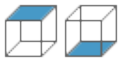
The View Menu

The View menu provides access to commands that affect the display and how things are viewed. This includes the Workspace, items within the Workspace, geometry, solids, workgroups, and coordinate systems.

- View Orientation and Control next

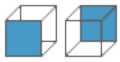
- "Explode/Coordinates" on page 77
- "Visibility" on page 78

View Orientation and Control



Top Bottom

(`Ctrl+E`) Top view is the standard, default view of the XY plane in Mill, ZX in Turning. Bottom is the opposite view.

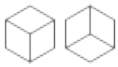


Front Rear

(`Ctrl+F`) This a view of the XZ plane, Rear is the opposite view.

Right Left

(`Ctrl+G`) This is a view of the positive YZ plane in Mill, positive ZY in Turning. Left is the opposite view.



Isometric Reverse Iso

(`Ctrl+I`) This is a "world" view. The part is drawn such that the part dimensions are all foreshortened equally. Reverse Iso is the opposite view of the isometric part.

Home Reverse Home

(`Ctrl+H`) This is an option for Mill/Turn, Multi-Task Machining, Advanced CS, SolidSurfacer, and products that build on these products. Home View is equivalent to the Top View for 2-axis mill or turning parts. For any part with multiple coordinate systems, Home View rotates the view of the part to a view normal (3D perpendicular) to the current coordinate system. The Home view always displays the part with the positive depth axis projecting out of the screen. This has the effect of showing the part rotated into the position it will be in when it is machined. Reverse Home shows a view of the negative side of the active CS.

Op Cutting Plane Reverse Op Cutting Plane

Click to highlight an operation, then select this option. The Cutting Plane of the operation is displayed.

Unzoom

(`Ctrl+U`) Also known as no zoom. This sets the scale of the stock diagram to the maximum size that displays the entire stock diagram. The actual size is based on the stock size and the current view.

Redraw

(`Ctrl+R`) This item refreshes the screen image allowing the user to get a better indication of what is currently drawn on the screen. Geometry, toolpaths, or the rendered part image can be redrawn using this item.

Previous View

(`Ctrl+K`) This item changes the current view to the last view that was worked in.

Zoom In and Zoom Out

(`Ctrl++`) Zooms in (closer view) each time you press `Ctrl++`.
(`Ctrl+-`) Zooms out (farther view) each time you press `Ctrl+-`.

Explode/Coordinates

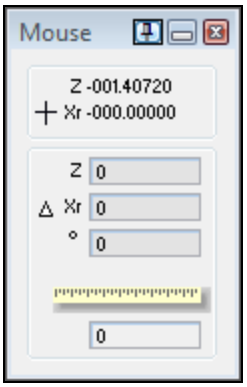
Explode View with Preview

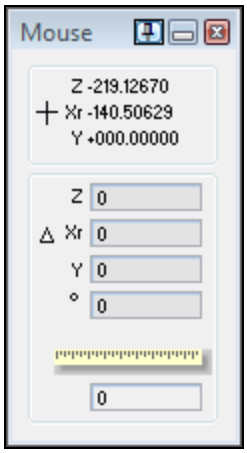
View each connected shape and solid with space separating it from the others, displayed from the cameras perspective. An insert preview window is displayed. Hovering over any area in the preview window will zoom the main view to that area.

Explode

As above but no preview window.

Mouse Position





2D location	3D location

The Mouse position dialog may also be accessed from the geometry context menu. The top section of the dialog allows the user to view the absolute two-dimensional (X and Y) or three-dimensional (X, Y and Z) location of the cursor. As the cursor is moved, the values are updated. The lower section of the window displays the incremental distance between two selected points, the relative distance between a selected point and the cursor and the angle of a straight line between two selected points or a point and the cursor.

By selecting two points with the mouse, the Mouse Position dialog will display the distance between the two chosen points as an incremental value. Each point selected will be confirmed by a red marker on the ruler within the dialog. When one point is selected, the system will display the distance (and angle) between the selected point and the cursor.

Visibility

Labels

(`Ctrl+L`) If this item is active, labels will be drawn next to every geometry feature. Circles and arcs will be labeled `C#`; Lines `L#`; Points `P#`, and Bezier splines (free-form curves) `B#`. The # is determined by the order the points and features were created in.

Points

(`Ctrl+J`) When this is active, all points (plain points, connectors, and terminators) are shown.

Virtual Points

(`Ctrl+J`) When this is active, all virtual points (line/arc/spline endpoints, circle/arc centerpoints, etc.) are shown.

Extend Lines

When this is active, all unconnected lines are extended to the boundary of the stock. If it is turned off, lines extend only to the features they were created between.

Show Stock & Origin

If this option is turned on, the gray outline of the stock and the origin marker are drawn.

Show Geometry

(`Ctrl+[`) When enabled, this redraws only points, lines, circles/arcs, and splines. It will draw geometry over the rendered image of the part.

Show Hole Features

This item toggles screen display of hole features. The top of the hole is shown as a cross. The bottom of a blind hole is shown with a line perpendicular to the hole axis.

Show Solids

When this is active, solids and sheets in the workspace are displayed.

Show Dimensions

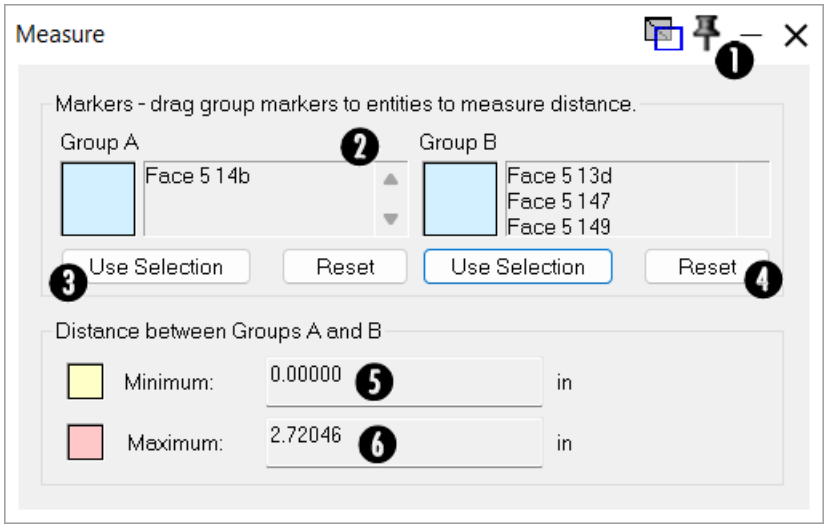
This toggles screen annotations on and off. Or, if the model contains PMI (Product and Manufacturing Information), it opens the pull-down menu for the Show Dimensions Floating Toolbar button, so that you can choose which items to show or hide.

Indicate Sheet Side

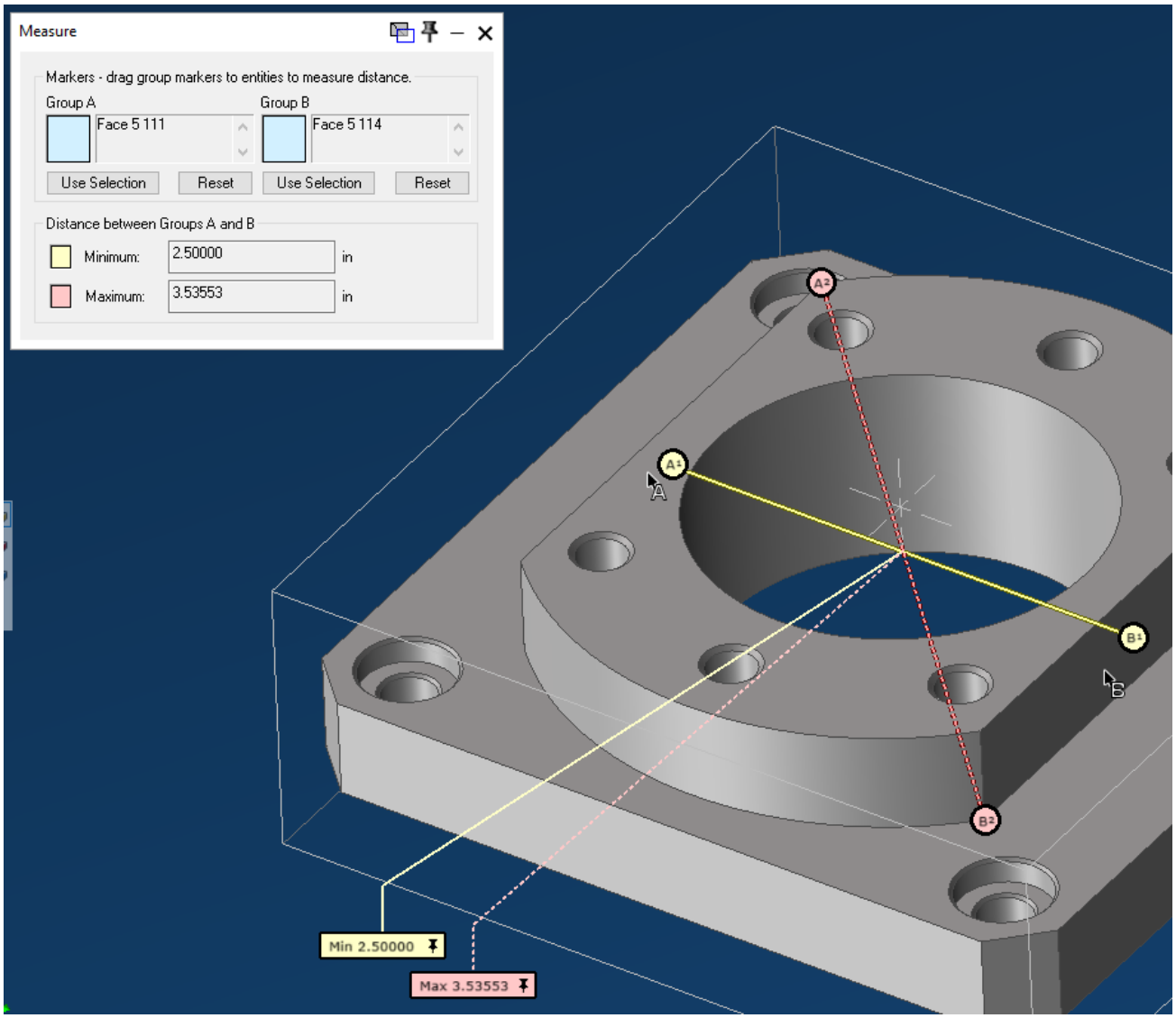
When enabled, the negative and positive sides of each sheet will be drawn in different colors.

Measure

Clicking Measure opens the **Measure** dialog box. To specify a pair of elements, you can drag and drop markers A and B onto the workspace and then reposition the marker as needed. And/or, you can select an element or a group of elements and click Use Selection. The minimum and maximum distances between A and B are displayed in the dialog box.

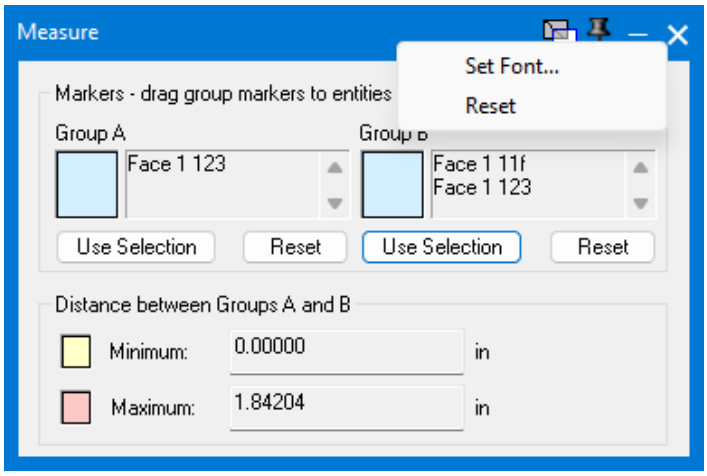


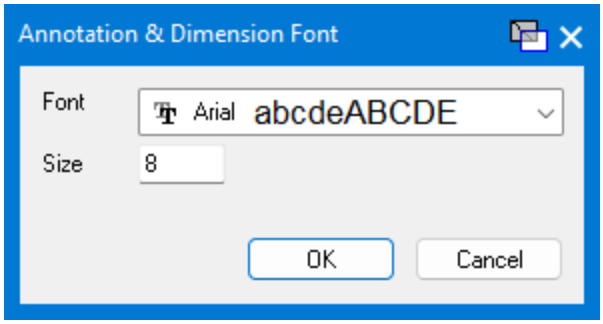
1. Group A, Group B: Each group lists the ID(s) of elements to be measured.
2. Use Selection — Replace current group items with selected element(s).
3. Reset — Remove all elements from this group. To reset both groups: Right-click, then choose Reset.
4. Minimum — Minimum distance between A and B.
5. Maximum — Maximum distance between A and B.



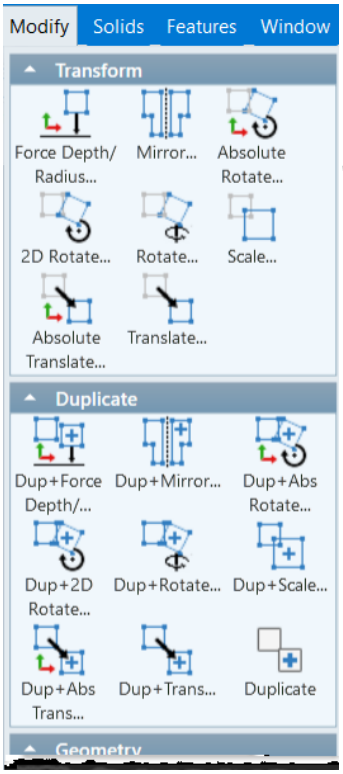
The minimum (yellow) and maximum (pink) distances are also displayed in the workspace as textboxes that can be repositioned. To lock the position of either textbox, click its pin button ( ).

Font type and size for dimension labels can be modified directly from the **Measure** dialog using the Set Font option.


Measure
Set Font...
Reset
Markers - drag group markers to entities
Group A
Face 1 123
Face 1 11f
Face 1 123
Use Selection
Reset
Use Selection
Reset
Distance between Groups A and B
Minimum:
0.00000
in
Maximum:
1.84204
in

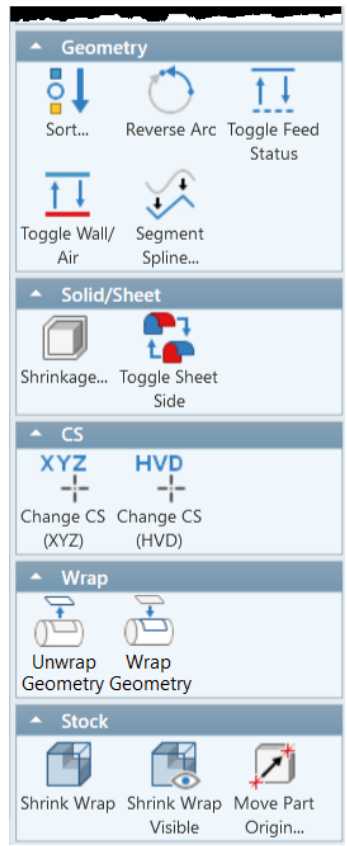

Annotation & Dimension Font
Font
Arial
abcdeABCDE
Size
8
OK
Cancel

The Modify Menu



Transform: See "Transform " on page 83

Duplicate: See "Duplicate" on page 87



Geometry: See "Geometry" on page 88.

Solid/Sheet: See "Solid/Sheet" on page 91.

CS: See "CS" on page 91.

Wrap, Stock: See "Wrap/Stock" on page 92.

Move Part Origin: See " Move Part Origin" on page 92.

The Modify menu contains commands for changing the state of geometry and bodies. All options under this menu, except Move Part Origin, require geometry or a body to be selected before their functions can be performed.

Several of the options have multiple pictures, one for the Mill module and one for the Turning module. A few options treat Mill/Turn as a separate case. The functionality of each of the options is identical for both the Mill and Turning modules; however, the dialogs differ slightly due to the different axis designations.

This checkbox option is found in several of the functions accessed from the Modify menu. The functions can be applied to entire workgroups as well as to any selected features. This function is accomplished by checking the Visible WGs checkbox and clicking Do It. Visible workgroups include all workgroups that are displayed in the Workspace, including background workgroups. The Reverse Arc, Change CS (XYZ) and Change CS (HVD) items in the Modify menu do not have dialogs associated with them. These three functions can also be applied to all items in Visible WGs by selecting the item from the WG List context menu, accessed by right-clicking the Workgroup list.

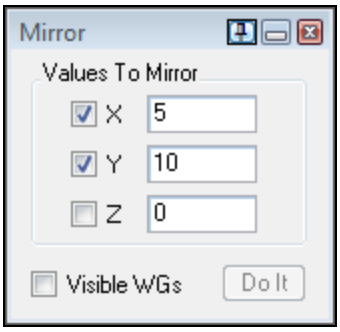
Transform

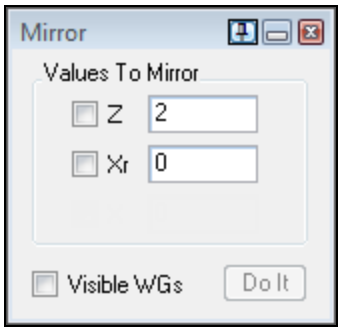
Force Depth/Radius

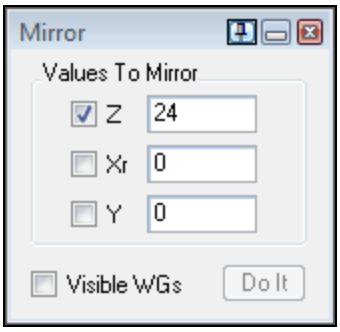
This opens the **Force Depth Radius** dialog box, allowing you to set the absolute Z value of all selected geometry to the Z value entered. when you are working in the Turning module, this option is unavailable and the menu item is grayed out.

When you are satisfied with your choices, click Do It to make the action occur. This dialog can remain onscreen for further use while other functions are performed.

Mirror





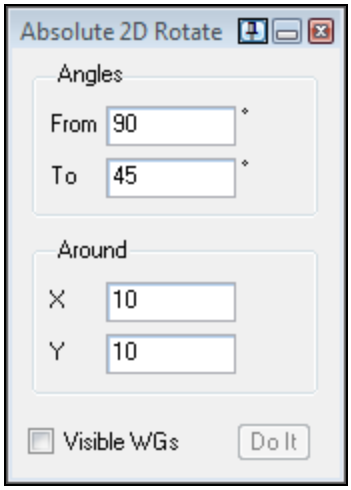


Mill	Turning	Mill/Turn

This opens the **Mirror** dialog box, allowing you to move the selected item(s) to the other side of the axis or axes specified around an axis point entered in the text boxes.

When you are satisfied with your choices, click Do It to make the action occur. This dialog can remain onscreen for further use while other functions are performed.

Absolute Rotate

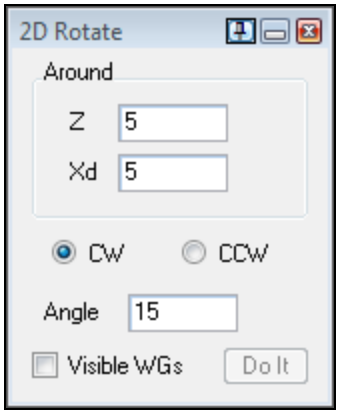


This opens the **Absolute \Rotate** dialog box, allowing you to rotate the selected item(s) at an absolute angle around a specified point. That is to say: The geometry will be rotated from an angle to another angle around the point specified in the X and Y (in Mill) or Z and Xr (in Turning) text boxes.

Holding down the `Alt` key displays **virtual points**, such as a line segment's midpoint, a circle's centerpoint, a solid's vertices, the corner and edge points of cubic stock, and many others. When you `Alt+click` a virtual point, you copy its value into the text box. This is called **interrogating** a point or virtual point. Also, the angle of a geometric element can be determined by interrogating the feature.

When you are satisfied with your choices, click Do It to make the action occur. This dialog can remain onscreen for further use while other functions are performed.

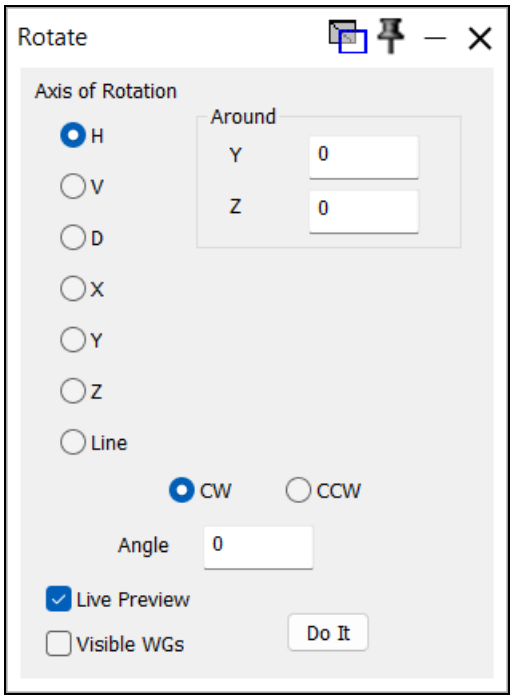
2D Rotate



This opens the **2D Rotate** dialog box, allowing you to revolve or rotate the selected geometry or bodies around the point specified in the X and Y (in Mill) or Z and Xr (in Turning) text boxes. The selected feature will be rotated by the amount specified in the Angle text box, in either a CW (clockwise) or CCW (counterclockwise) direction depending on the selection made.

When you are satisfied with your choices, click Do It to make the action occur. This dialog can remain onscreen for further use while other functions are performed.

Rotate

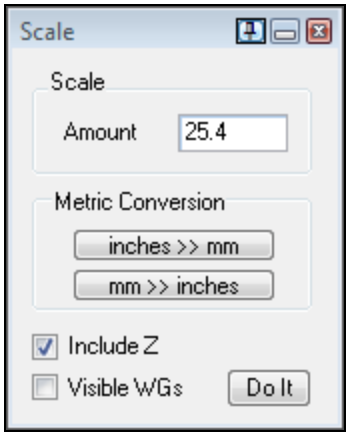


This opens the **Rotate** dialog box, allowing you to rotate the selected geometry or bodies clockwise or counterclockwise by a specified positive or negative angle around an axis that you specify. For example: Line lets you select a line, hole, or straight edge from geometry or solids. Each method lets you set the center point for rotation. Point snapping / selection is supported.

When the Live Preview checkbox is ticked, a real-time preview of the rotation direction and axis is displayed, allowing you to make changes before applying the results. A clear visual (arrow around a bar) appears in the workspace. This helps you avoid mistakes with direction or axis choice.

When you are satisfied with your choices, click Do It to make the action occur. This dialog can remain onscreen for further use while other functions are performed.

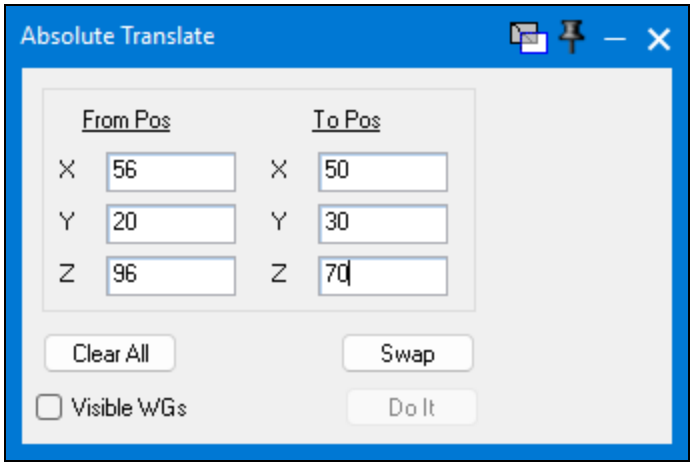
Scale



This opens the **Scale** dialog box, allowing you to change the size of the selected geometry, solids, or sheets by the amount specified. The value entered in the Amount box is a ratio of 1, and so a value of 1 is full size. The inches >> mm or mm >> inches buttons will change the units of measurement accordingly. The Mill module has an Include Z option. If the Include Z option is turned on, the Z values of the selected geometry will be included in the scaling process. Include Z will not have an effect if the selected geometry or body is at Z0.

When you are satisfied with your choices, click Do It to make the action occur. This dialog can remain onscreen for further use while other functions are performed.

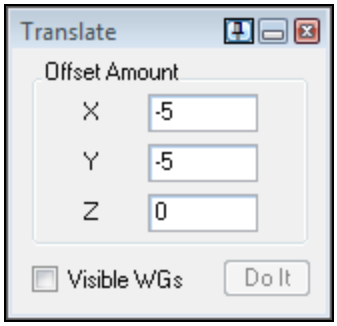
Absolute Translate



This opens the **Absolute Translate** dialog box, allowing you to move the selected item(s) to a new position. This is an absolute move by a certain value. Please note that you do not need to use a value directly associated with the geometry or body. That is to say: You can move an element from its position to a new location based on relative numbers. For example, a sphere centered at X20Y20 can be moved to X40Y40 using a FromPos value of X0Y0 and a ToPos value of X20Y20.
The Clear All button clears all input values and resets them to zero, while the Swap button exchanges the values between the FromPos and ToPos fields.

When you are satisfied with your choices, click Do It to make the action occur. This dialog can remain onscreen for further use while other functions are performed.

Translate



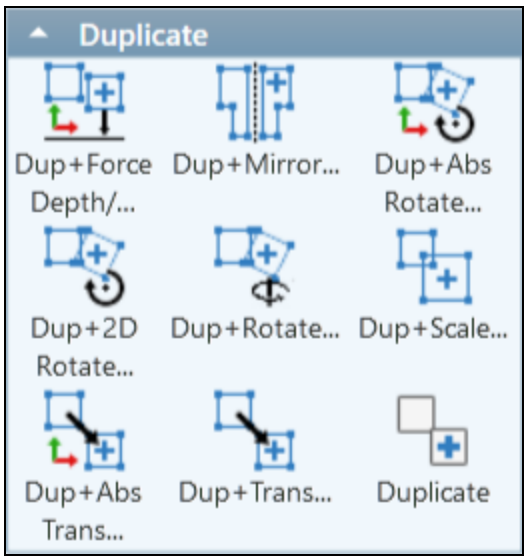
This opens the **Translate** dialog box, allowing you to move the selected geometry or bodies by the specified amount.

When you are satisfied with your choices, click Do It to make the action occur. This dialog can remain onscreen for further use while other functions are performed.

Other Modify Commands

Duplicate

Duplicate And...



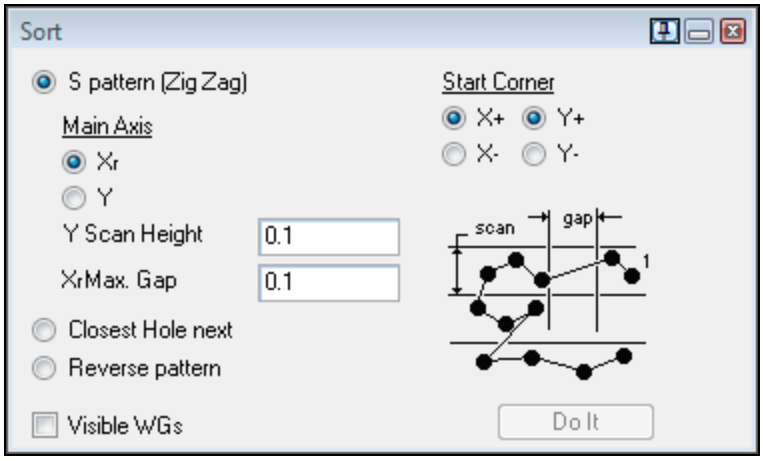
This item will Duplicate and perform the specified action. Force Depth, Mirror, 2d Rotate, Scale or Translate the selection. First, the selected geometry will be duplicated. Then, depending on the option selected, the selected geometry is moved to one or more new locations based on the number typed in the times text box.

Duplicate

(Ctrl+D) This selection copies the selected geometry or body to the same location. After this is done, the next step is to move the duplicated item to another location, possibly using another item under this menu. Additionally, Duplicate is useful for creating an unconnected feature in the same location as a currently connected feature.

Geometry

Sort



This selection is only available in the Mill module. It provides the user with a method for sorting the selection order of a selected group of points and unconnected circles. Once the points or circles are sorted, the reference numbers (labels) are changed to match the new order. This is particularly useful when performing drilling operations, because the system drills holes in the order they are selected. There are three possible sorting options: S pattern (Zig Zag), Closest Hole next, and Reverse pattern. The different options are described below.

S pattern (Zig Zag)

This option allows the user added control over how the points or circles will be selected. When using S pattern, the system creates a rectangle around the selected group points like the picture shown in the dialog. This rectangle is broken down into definite sections (like a grid) by creating scan lines and grid squares. The grid will be created starting with the point in the farthest upper right-hand corner contained in the selected rectangle.

The user designates what corner of the rectangle to select the first point by using the Start Corner radio buttons. For example, selecting X+ Y+ will start the selection order in the upper right-hand corner; X- Y+ in the upper left-hand corner, and so on. Selecting a Main Axis (X or Y) and entering a Scan Height value will create scan lines, either horizontal or vertical, depending on the main axis selection. The Max. Gap value breaks up the scan lines into grid squares that can be scanned for points. The Scan Height and Max. Gap values create the scan lines and squares which break the area up in a grid-like fashion where all the selected points are contained in a specific, defined area of the rectangle.

The system will select the point closest to the start corner (regardless of what scan line it is in) as the first point, and then proceed to the next closest point in that section of the grid. If

no other points are contained in that area, the system will move to the next closest section and select the next group of points. The system always looks at the scan line first and then looks across or down the square for the next point. This continues until all selected points in the group are found and labeled.

Closest hole next

This option is fairly self explanatory. The system starts with the first point or circle selected and then selects the point or circle that is closest. This process continues until all holes have been selected. This option is useful when creating curves.

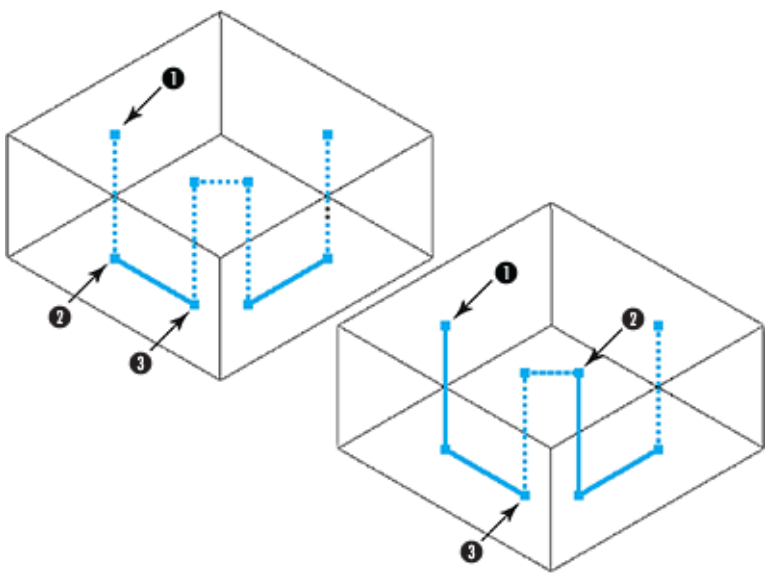
Reverse pattern

This option reverses the original selection order of the group of points or circles. The last point that was selected is now labeled as the first point and so on.

Reverse Arc

`(Ctrl+T)` A circle can be fully connected either using the less than 180° arc or with the greater than 180° arc. The system automatically selects the less than 180°arc as the fully connected feature. If the other arc is needed, select the connected arc and the Reverse Arc item. In other words, selecting Reverse Arc will tell the system to use the other side of the connected arc.

Toggle Feed Status



1. Start Point, Begin Rapid
2. Begin Feed
3. Begin Rapid

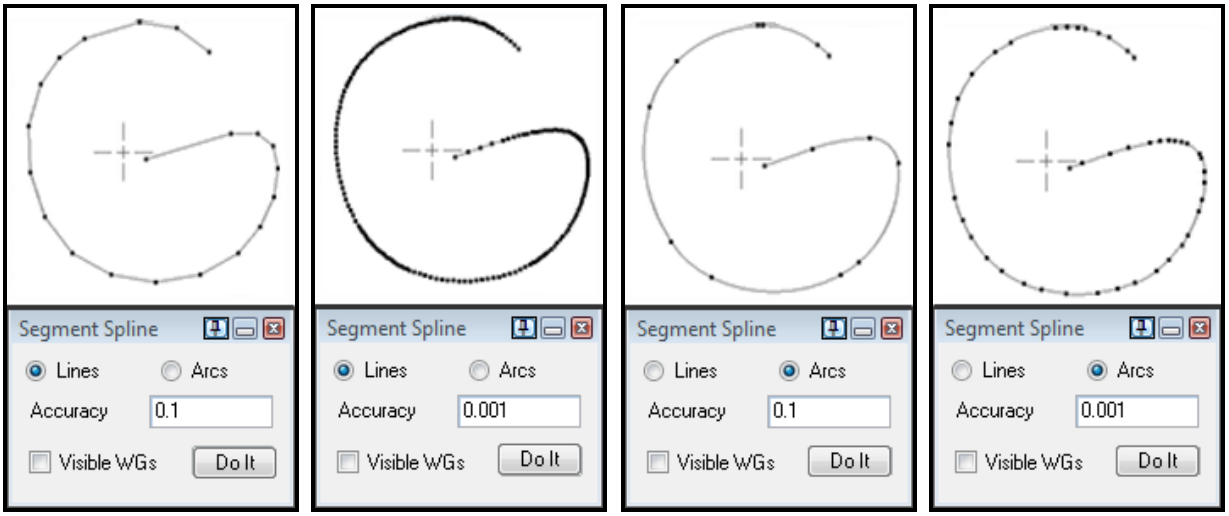
Toggle Feed Status allows the user to toggle the status of geometry from a Feed line to a Rapid line. This is particularly useful in roughing operations as illustrated in the Mill Tutorial where mouse lines are drawn, depths are changed and Feed or Rapid is designated.

This image shows a series of tool moves. The dashed lines indicate Rapid moves and solid lines indicate Feed moves. Note that we are rapiding into the part. In the second image we have toggled the lines from Rapid to Feed where the tool moves down onto the part.

Toggle Wall/Air

This option toggles geometry between wall features (light blue) and air features (red). This is used for pocketing, particularly for open sided pockets. Tools will not cross wall geometry but can cut across air features. See the Mill manual for more information.

Segment Spline



An example of various spline segmentation settings.

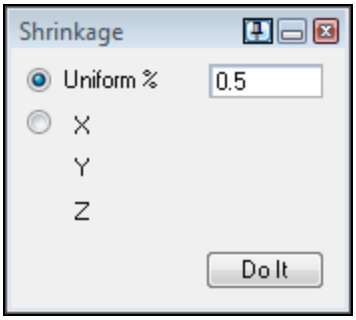
Segmenting splines means converting a smooth curve (a NURBS) into a series of line or arc segments that approximate the shape of the curve. The features, or segments, may consist of lines or arcs. When converting the spline to segments the Accuracy must be set. The Accuracy value is the chord height used to designate how closely the segments will approximate the spline. The tighter the accuracy, the more segments that will be created to approximate the spline.

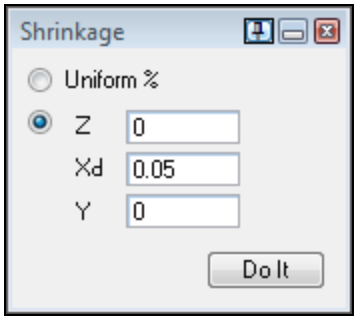
Segmenting splines into Arcs is generally preferable over segmenting into Lines. Arcs will provide smoother toolpath and less G-code output.

The series of images illustrate the differences between segmenting a spline into lines versus arcs at two different accuracy settings. As you can see, even a loose accuracy with arcs is smoother and more accurate compared to the same accuracy used with lines.

Solid/Sheet

Shrinkage





The Shrinkage function is designed to be used when creating molds to compensate for shrinkage factors encountered in the manufacturing process, compensating for the rate at which an injection substance will shrink in a mold cavity. The Shrinkage item lets you perform uniform or axial reductions or enlargements on selected bodies, including facet bodies, geometric solids, and sheets. The Shrinkage range is -10% to 10%. A selected body can be reduced or enlarged up to 10% of its original size. Repeated use of the function can yield shrinkages outside this range (not recommended).

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

Toggle Sheet Side

The Toggle Sheet Side item will flip the inside and outside of a sheet. This item is useful when solidifying sheets into bodies using the Offset solidify option. When sheets are converted into bodies 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.

CS

Change CS (XYZ)

When Change CS (XYZ) is used, all selected geometry will be assigned to the current coordinate system. The geometry will stay in its same location in 3D space. It will change color to reflect the fact that it is now in the current coordinate system. All selected arcs will be segmented (changed into line segments), if the coordinate system they are being changed into uses a different plane than their original coordinate system. If the Advanced CS module is not installed, this item will be grayed out in the menu.

Change CS (HVD)

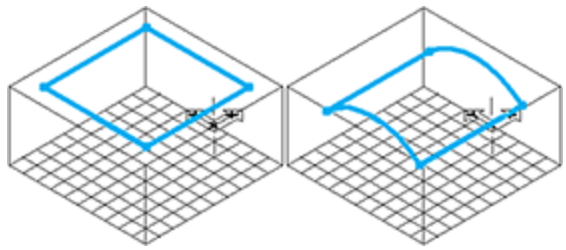
`(Ctrl+\)` When Change CS (HVD) is used, all selected geometry will be assigned to the current coordinate system, and the HVD values of the geometry will be preserved. This means that the geometry will be modified to be planar to the new CS location preserving the relative position. If the Advanced CS module is not installed, this item will be grayed out in the menu.

Wrap/Stock

Unwrap Geometry

Selecting this item returns the display of wrapped geometry to flat geometry and takes dialogs out of radial mode.

Wrap Geometry



Selecting this menu item will wrap all geometry in any Wrapped workgroup and switch Geometry dialogs to radial mode. Geometry will be "wrapped" around the part at the radius or depth at which it was defined. This item is most useful in the Level 1 interface or as an alternative to the Wrap Geometry button found in the Toolbar of the Level 2 interface.

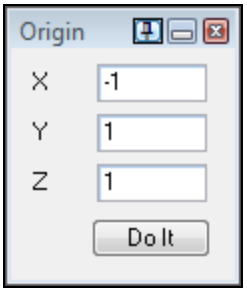
Shrink Wrap

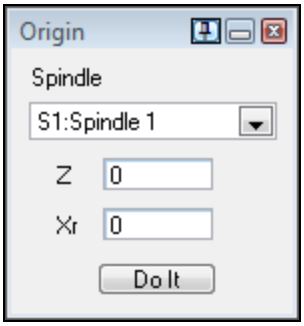
(`Ctrl+'`) When this item is selected, the window zoom size becomes just large enough to display all geometry in the part (including visible and hidden workgroups). This is especially useful when working with imported files.

Shrink Wrap Visible

(`Ctrl+]`) When this item is selected, the window zooms out or zooms in to display all *visible* workspace elements without excess blank space. This is especially useful when working with part files that include hidden (👁) workgroups or items in the Body Bag.

Move Part Origin





This function can save the trouble of recalculating the values of stock definition in the Document Control dialog if the part's origin needs to be modified. This selection will move the part origin relative to ALL the geometry in EVERY workgroup incrementally by the values typed in the appropriate axis text box. In other words, the current location of Xo, Yo, Zo or Zo, Xo will change incrementally by the values in this dialog and a new Xo, Yo, Zo will be established. When using MTM you can choose which spindle will be modified. Clicking on the Do It button will move the origin. This dialog can remain onscreen for further use while other functions are performed.

The Solids Menu

The Solids menu provides tools for verifying the validity of bodies and toolpath. Most of the items in this menu are system development tools that may be of little use to end users; however, on some occasions, they may be helpful as a method of diagnosing issues with solids.

Rebuild

This command will update any solids if changes have been made in the part's history. This command is also accessed by right-clicking on a body and is often used in conjunction with the Recreate command. This function is not undoable.

Check Self-Intersect Loops

It is recommended that this option is left on for optimal 3D toolpath generation. Turning it off will not save any system resources.

Tools sub-menu

The system's arsenal of tools for diagnosing problematic solids can be found in this menu. These options are miscellaneous tools to check and correct bodies as well as report and output data. Each item is described below under the category to which it belongs.

Validity Checks

These items do not fix any errors or problems you may be experiencing, but are simply a means to determine whether the body you are using is valid or not. This may aid in troubleshooting any problems you are experiencing with machining solids.

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 will be deselected once the check is complete, allowing the user to identify the problem. An error message identifying the specific problem will also be displayed for each invalid entity.

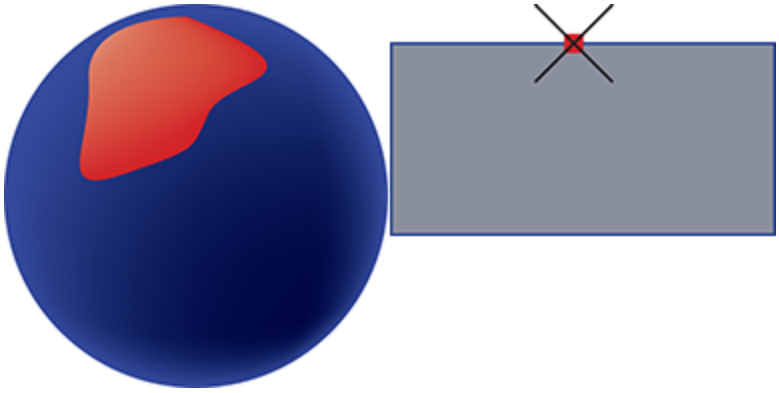
Check Face Validity

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 see if 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 command will inspect selected bodies or faces for any unneeded elements. If two items share the same underlying surface, the body will be simplified. In this example we have a sphere and a cuboid. The sphere (consisting of surfaces) shares a common underlying surface with another sheet. The extra sheet will be removed from the model when running this command. One edge of the cuboid actually consists of two intersecting lines. The lines will be modified to be a single edge.

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 will convert those NURBS back into analytic. The geometry of a body is simplified wherever possible. A body will be defined by lines, circles, spheres, cubes, cylinders and tori instead of B-splines.

Multi-Pass Stitch

Multi-Pass Stitch is a setting used with the Solidify function. When Multi-Pass Stitch is active, the Solidify function will perform successive attempts to solidify sheets, starting at the tolerance specified. Sheets that do stitch at the specified setting will be stitched; any sheets that do not stitch at that tolerance will be attempted at a looser tolerance. This will be attempted a third time at a still looser tolerance. Multi-Pass Stitch is on by default.

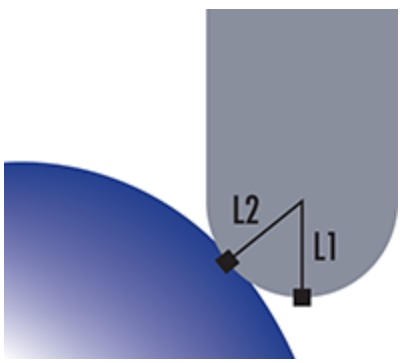
Check Trimmed Surface Polyline

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 Surface Edges

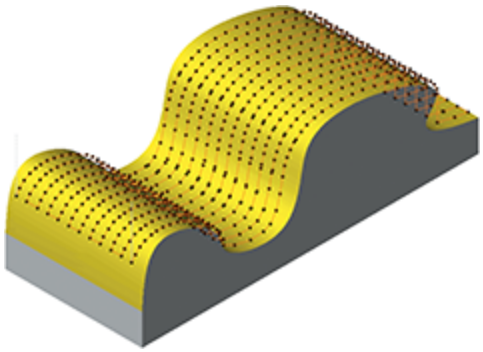
This item verifies the validity of trimmed surface edges to ensure proper machining. Check Trimmed Surf. Edges is only necessary when using the Gen 2 Engine in surfacing operations.

Check Op Gouges



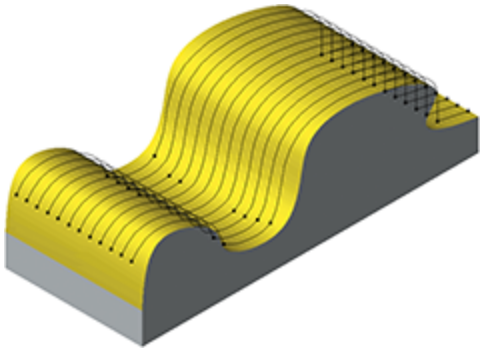
The system will perform a check to see if a selected operation gouges a selected body. A gouge is defined as being off by greater than 50% of the machining tolerance. This command only works with ball endmill tools. If the operation gouges, two lines will be drawn from the tool to show the gouge. The first line runs from the tool center to the tip, and the second line runs to the point of the gouge. This is illustrated in the following image.

Selected Op to Screen Points



This command creates points at the beginning and end of each feature of a selected operation. Any type of operation on a solid may be used with this command. The Selected Op to Screen Points command is useful for dimensioning toolpath. The image on the right shows a lace cut operation composed of line segments. Note the many points created by Selected Op to Screen Points. If the lace cut was defined by arcs or splines, the points would be different.

Selected Op to Splines



This command creates a spline between sharp corners of any surfacing operation. For the Selected Op to Splines command to work the curves dialog must be open. The tolerance of the splines generated from the toolpath is determined by the settings in the curves dialog. This image illustrates splines generated from a lace cut operation defined by line segments.

Selected Op to Lightwork File

This command outputs the selected operation to a Lightwork file.

Selected Op to Text File

This command will output a text file of the toolpath. The text provides the position of all features. A line feature includes the start and end X, Y and Z dimensions. Arcs include start and end point X, Y and Z values, the arc's radius and the arc's direction.

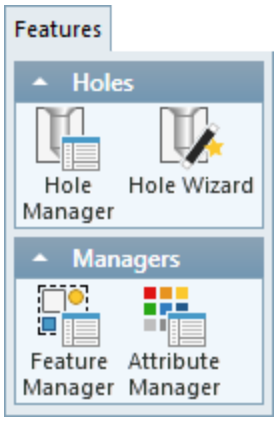
GE to Text File

Geometry Expert must be open to use this function. Selecting this command will output the Geometry Expert's contents to a text file named `GEDump.txt`. The file is created on the Desktop.

Clear Bodies' History

This option will remove the History of any selected bodies, making the body an atomic body. Undo is not available for this.

The Features Menu



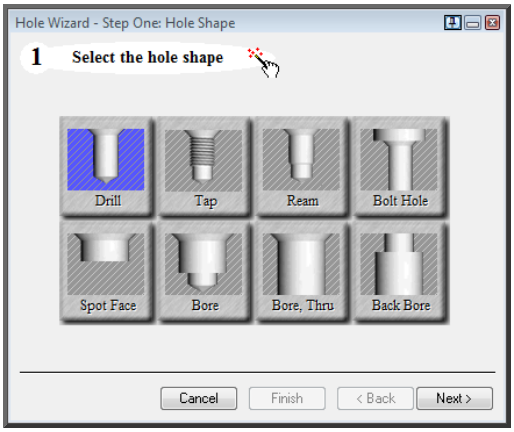
The Features menu provides access to two tools specifically for Hole features and two tools that help you manage user-defined features..

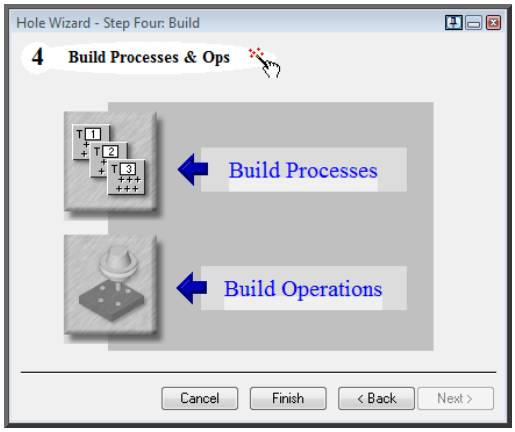
- Hole Manager helps you to identify and machine holes in solid models.
- Hole Wizard quickly creates hole operations for you using the available tools and geometry.
- Feature Manager provides additional capabilities for managing attributes of type Feature and elements that have been tagged with Feature attributes.
- Attribute Manager allows you to create and manage (view/sort/edit/etc.) data tags that attach to multiple elements; attributes can be of type Integer, Real, Text, Color, or Feature.

All options on this menu are fully detailed in the *Features* guide. This guide is provided with the installation media and is available for download from https://online.gibbscam.com.

Hole Wizard

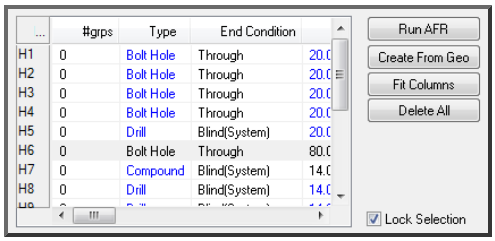
The Hole Wizard automates the process of making holes. The Hole Wizard can make a single drill hole or it can make a pattern of tap holes using multiple tools on any number of holes. The Hole Wizard does all of this in four easy steps and guides you through them.





Two of the Hole Wizard steps

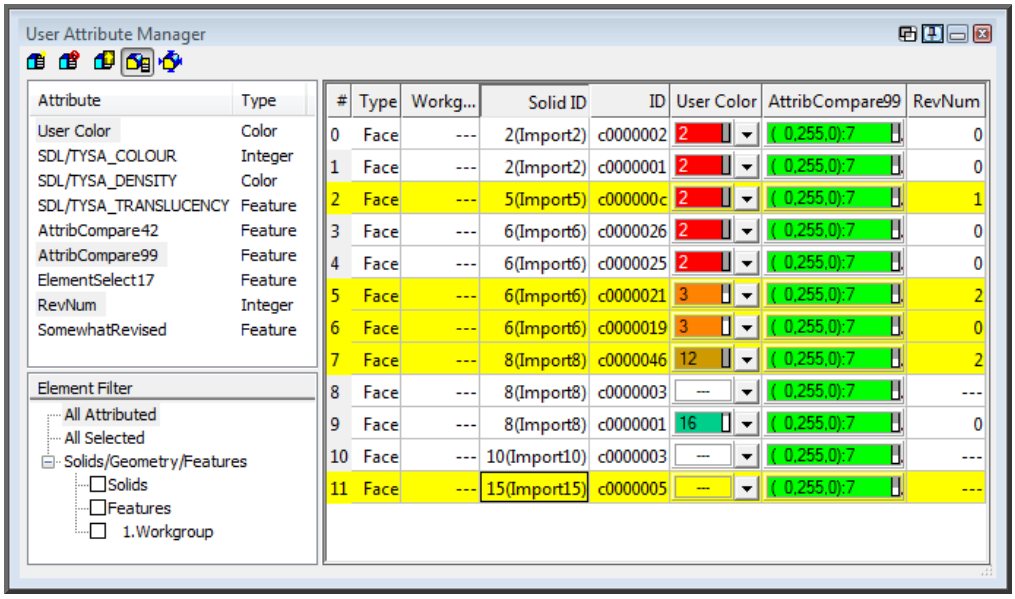
Hole Manager



Hole Manager lets you identify Hole features in a model, specifying geometry, creating groups, and quickly defining the features. From within Hole Manager, you can invoke Hole Wizard (or use Auto Wiz) to invoke it multiple times) to create operations and the list of tools that are needed to create holes.

Hole Manager is intended for models with a large numbers of holes that are primarily solids. However, even if you do not have any of the Solids options, you will find the sorting and grouping of holes very useful.

Attribute Manager

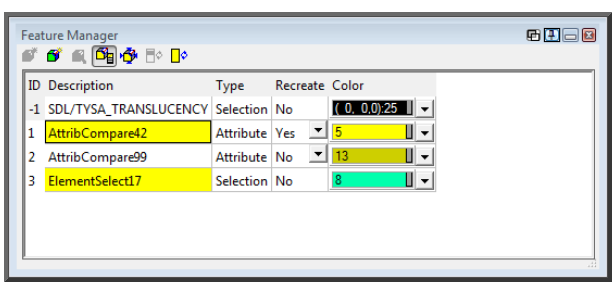


Attribute Manager presents attribute information in three panes:

- In the upper left, a two-column table lists the attributes defined for the current part. You can sort by either column. When one or more attribute names are selected, corresponding columns appear in the element list on the right.
- In the lower left, the Element Filter pane lets you specify what elements will appear in the element list.
- On the right is a table of elements that meet the filter criteria. Each qualifying element appears as a row of values. Cells on the far right show values for attribute whose names that are currently selected.

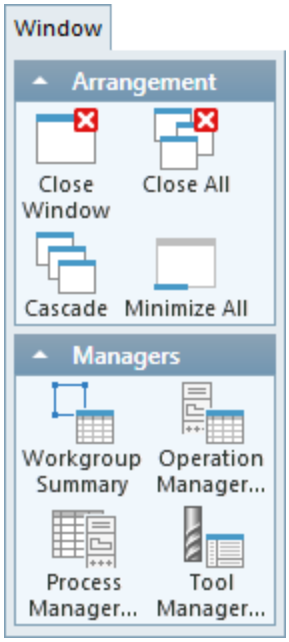
Feature Manager

Feature Manager lists user-defined features in a table that displays their ID, Description, Type (element-selection-based or attribute-comparison-based), Recreate flag, and Color:



To modify a feature's Description, Recreate flag (for features based on attribute comparisons), or Color, click or double-click the corresponding cell.

The Window Menu



The Window menu provides control over windows and dialogs. It also provides access to part summaries including workgroup data, the contents of the tool list, and a summary of the part's operations.

- Arrangement , next
- " Tool, Process , Operation and Workgroup Managers " on page 102

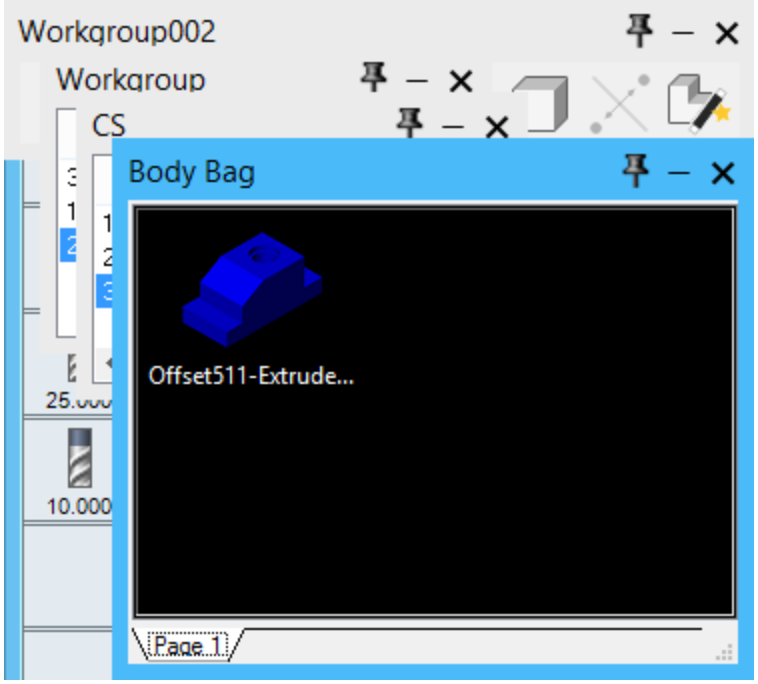
Arrangement

Close All

Closes all open sub-menus.

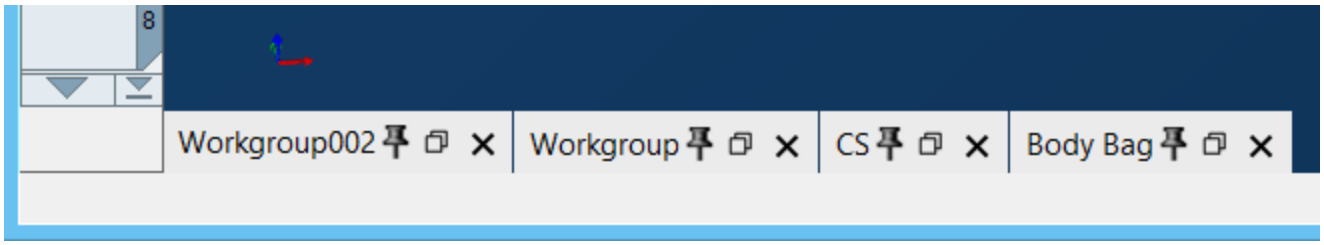
Cascade

Arranges dialogs so that all title bars are visible but dialog bodies overlap, as shown here:



Minimize All

When turned on, this command shrinks all dialogs to title bars only, tiling them along the bottom edge of the workspace window, as shown here:

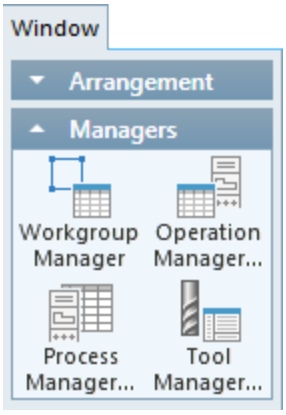


Managing a Large Number of Processes

If the base folder specified by Set Directory contains 500 or more sub[sub[...]]folders that contain processes (or if the total number of processes contained exceeds 2500), the Process menu will not display all folders (or all processes). Instead, the Process menu will display a new menu item: Process Manager. Clicking this item opens a dialog that shows a collapsible hierarchy: the base folder, its applicable subfolders, and so forth. This lets you navigate quickly and easily to the processe you need to access. The top right corner of the Process Manager dialog has a Browse button ... that lets you reset the base folder for your processes.

Tool, Process , Operation and Workgroup Managers

Manager reports can be accessed from several places.



Managers are always available from the Main Menu Window tab.

Right-click the Tool/Process/Operation/Workgroup List and select Manager ... from the menu.

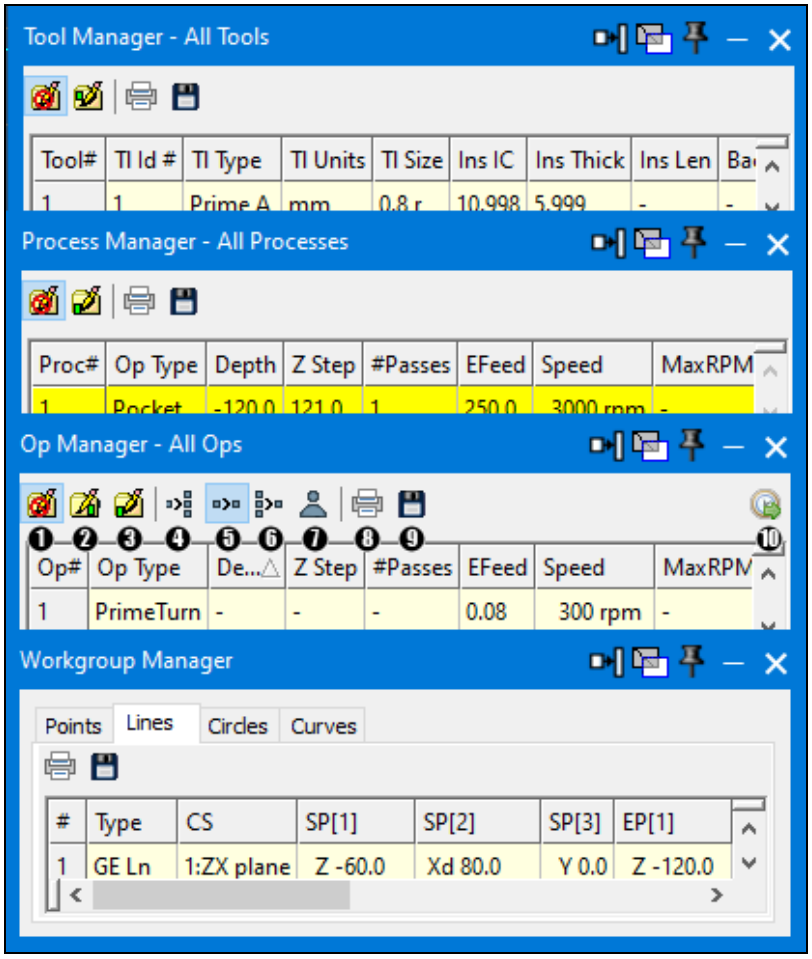
Operation Manager and Tool Manager are located in the Command Toolbar.

Viewing Data

In each of the Manager interfaces, you can do any of the following:

- Choose which columns to display and in which order, by right-clicking the column heading and choosing Customize Headers.
- Sort the display with a simple click on the required column header. You will note that the "sorted by" is indicated by a triangle symbol in the header — pointing up is low-high; pointing down indicates high-low.
- Right-clicking anywhere in a row reveals a menu that lets you:
  - Sort the entire tile list, or find tiles and move them within the list.
  - Open the corresponding dialog.
  - Isolate a subset of rows.
- Print or save the data using the icons provided in the header.

Dozens of data types are available. Tool Manager can display up to 35 columns; Process Manager, up to 50; Operation Manager, up to 79; Workgroup Manager, up to 47. Balloon help will provide a brief description when hovering over a column heading.



1. Disallow editing (Default)
2. Allow editing of Operation fields
3. Allow editing of Process Fields
4. Individual Mode
5. Process Mode
6. Process List Mode
7. Manual Mode
8. Print
9. Save
10. Calculate TP/Len Time

Operation Groups

Op Manager toolbar allows you change the grouping mode, which is synchronized with the grouping mode in the Op Tile list. For more information on active and inactive ops, see “Operations List” on page 211.

- When collapsed, each group shows only the values that are shared between operations in that group. Editing shared data in the operation or process changes the corresponding value for all operations in that group.
- When expanded, each operation’s data is fully populated individually, and the data can be edited for each individual operation.

Editing Data

The Manager dialogs provide powerful editing capabilities. Clicking the Allow Editing buttons (shown above) highlights all editable fields. You can modify the value in an editable cell, or you can apply a single value to many selected items at once.

Multi-Selection

You can select multiple tiles to highlight multiple rows, and vice versa. However — ***important!*** — use extreme caution when using Edit mode, especially context menu item Apply Value to Selected..., which copies the value from the cell under the mouse location (the pointer cursor) into all highlighted light-green cells in the same column, whether or not those cells are visible in the current scroll region. (The Apply Value ... choice is unavailable if any of the highlighted cells in the same column is ineligible to receive the value of the cell that you right-clicked.)

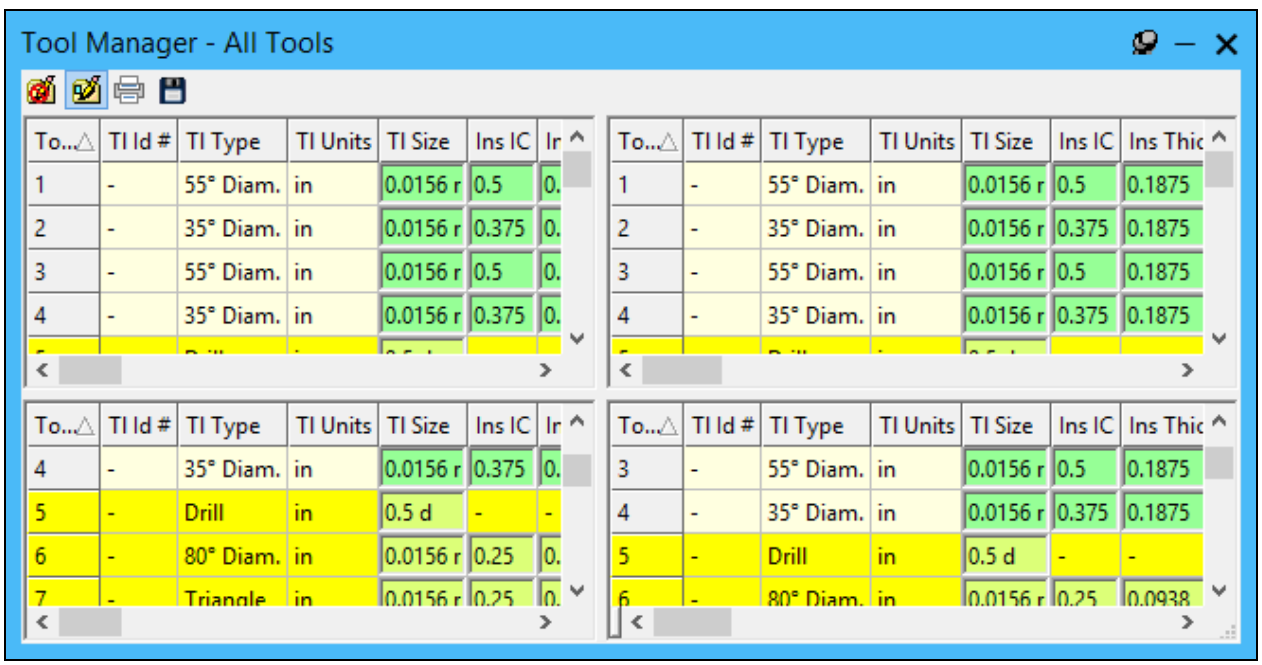
Warning: In Edit mode, a single click can cause far-ranging global changes, and ***there is no Undo capability***.

To edit data:

1. Click Allow Editing of Data Fields. A warning message appears.
2. Click OK. The fields you can edit highlight in green.
3. Double-click the field you want to change and type the information. Or, if you are very confident, you can right-click a cell whose value you want to copy to all editable cells and, on the context menu, choose Apply value to selected operations This should not be used if you have any doubt about the outcome.

Multiple views

The manager allows multiple views, and each view can be scrolled independently.



To split the manager grid:

- To create or reposition the *horizontal* split: Hover your cursor just left of the horizontal scrollbar at the bottom. When the cursor changes to ⟺ or ⫼, `click` and `drag` it to the right or left to create or reposition the horizontal splitter.
- To create or reposition the *vertical* split: Hover your cursor just above the vertical scrollbar on the right. When the cursor changes to ⇕ or ⫶, `click` and `drag` it down or up to create or reposition the vertical splitter.

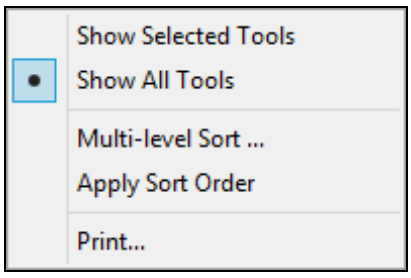
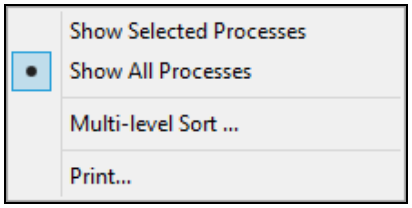
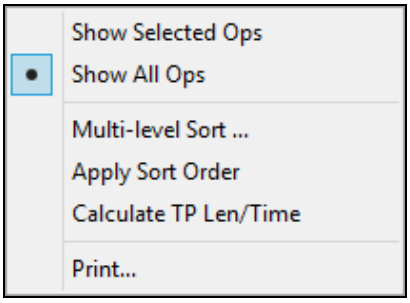
Manager Context Menus

The Manager dialogs provide the following right-click menus:

- Title bar context menu
- Header row customize option
- Tool row context menu

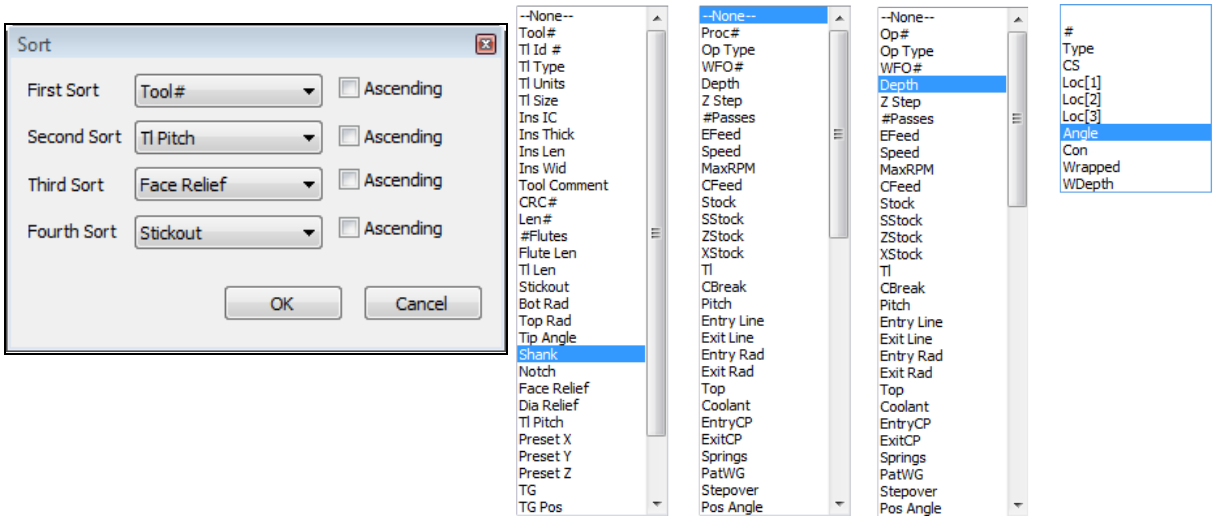
Title Bar Context Menu

This is not available for the Workgroup Manager. To select items from the title bar context menu: `Right-click` the title bar and select the item you want from the menu.

Tool Title Context	Process Title Context	Operation Title Context
		

Show Selected/All:
Displays selected or all items in the Manager.

Multi-level Sort
Selecting the Multi-Level Sort... command opens a dialog where you can select up to four columns you want to use as sort criteria. Check the Ascending checkbox to sort in ascending order, unchecked will sort in descending order.



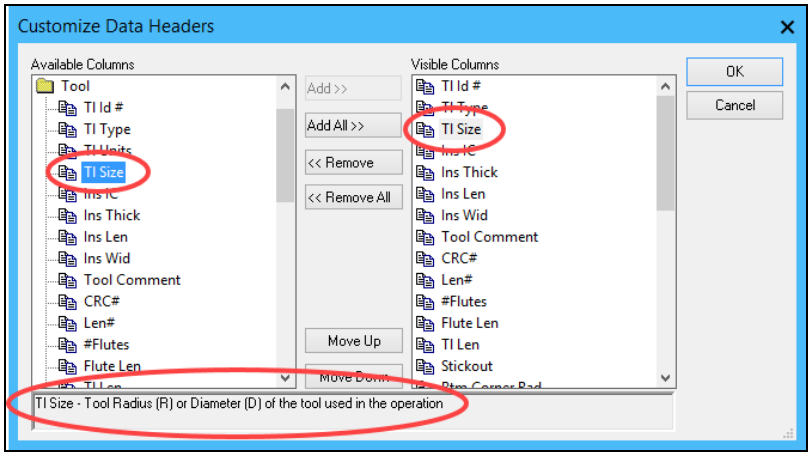
Clicking OK transfers the sorted order of the items in your Tool/Ops Manager to the actual Tool and Operation lists.

> ⓘ Important: Performing a sort changes the order of your operations. If you are not careful you could tell the system to do something you do not want, such as tapping a hole before the hole is drilled. The system does have a reminder to warn you that performing a sort can be harmful but you should still be aware of the potential consequences.

Header Customization

`Right-click` the column title row and select `Customize Headers`. This option enables you to select the columns to wish to show or hide in the Manager reports and place them in the desired order.

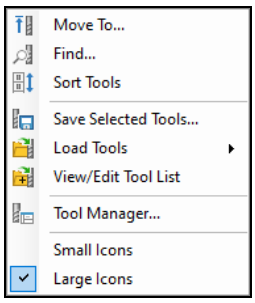
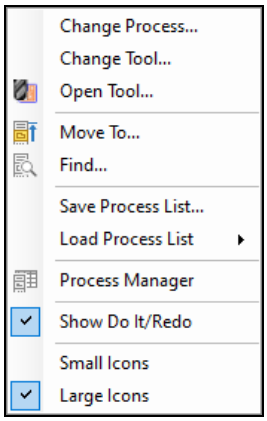
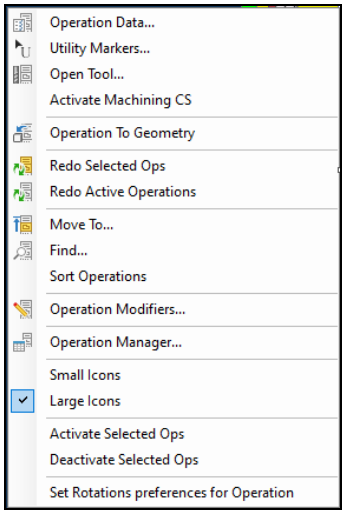
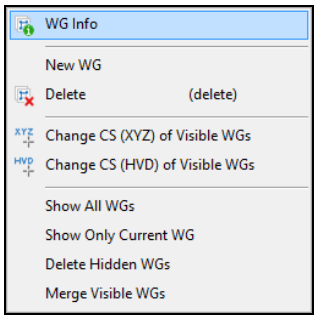
Clicking a data type will provide a brief description of the item at the bottom of the screen. The item is also highlighted in the visible columns window. Please note that data can only appear once in the report.



- *To show a hidden column:* Under **Available Columns**, select a heading and click `Add>>`.

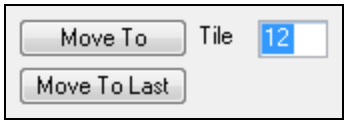
- *To hide a visible column:* Under **Visible Columns**, select a heading and click <<Remove.
- *To move a visible column:* Under **Visible Columns**, select a heading and click either Move Up (to shift left) or Move Down (to shift right).

Report Row Context Menu

Tool Manager	Process Manager	Operation Manager	Workgroup Manager
			

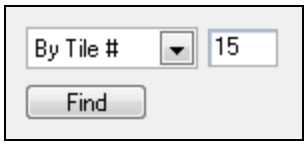
To select items from the row context menu, `right-click` a detail row and select the item you want from the menu.

Move To:



You use the Move To option to move a tile to a specific position by number or to move a tile to the last position in the list.

Find:



You use the Find option to locate a specific tile by number or jump to the last tile. For operations, you can also search by tool number.

Sort:

Sorts all tools by Tool number in ascending order. For Operations, sorting reorganizes the operations by tool number and creation order, from lowest to highest tool number. The system examines all the operations and tries to group them by tool number to minimize the number of tool changes. Any blank spaces in the Operations List are removed.

Operations created in a Process Group (a group of processes to machine the same geometry) are sorted based on their order in the Process List when they were created. For example, finishing

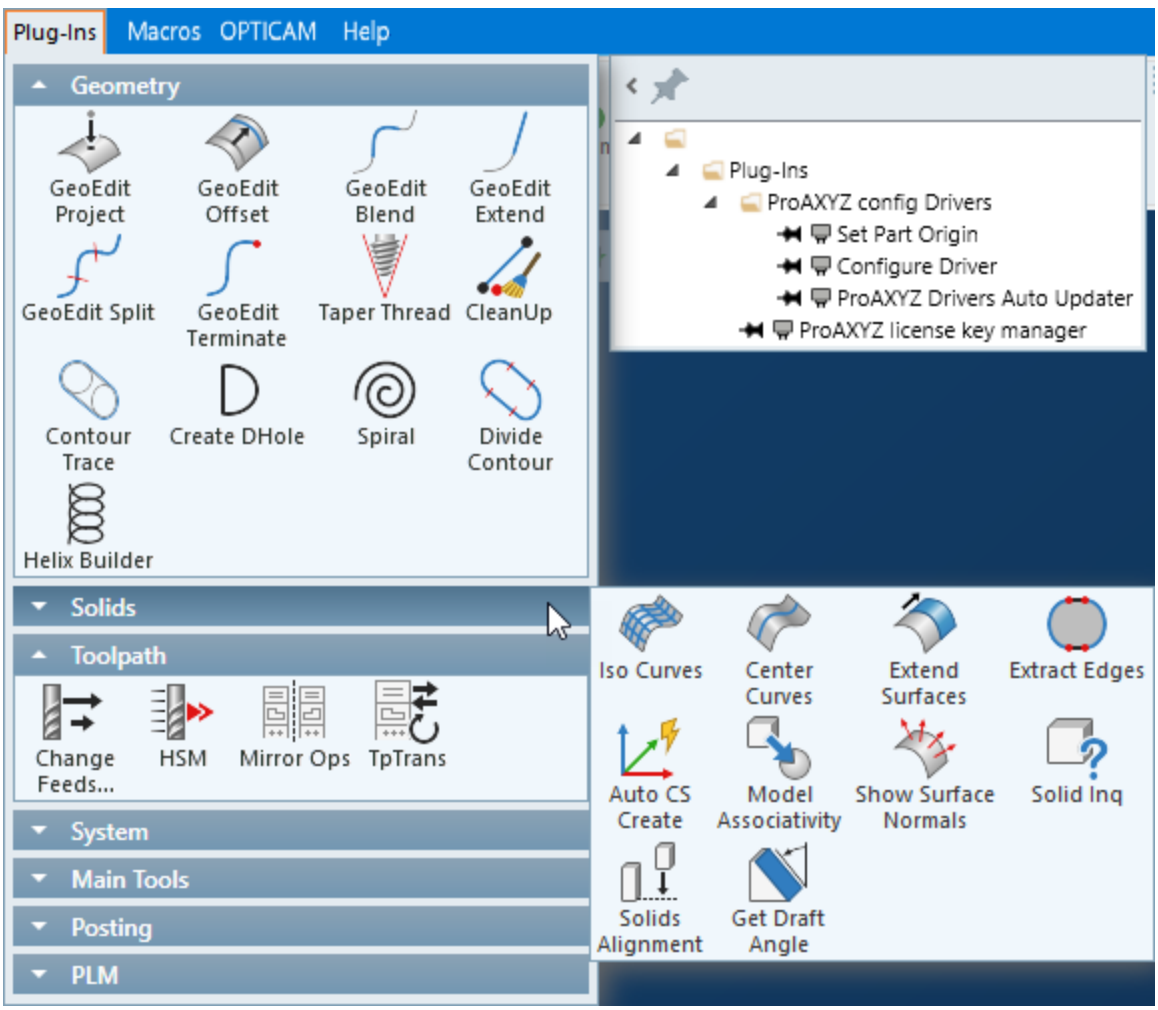
operations are not placed before the roughing operations in a Process List. You can manually reorganize operations by dragging them to the location you want in the Operations List. The order of machining in the finished NC program is the same as in the Operations List. Therefore, the order of tiles in the Operations List is very important. You can organize operations anytime during the programming of the part.

> ⓘ Please be aware that sorting operations can be risky. For example, you could potentially tell the system to tap a hole before it is drilled. Please be sure to review the results of the sort to ensure you get the results you want.

Apply Value to Selected:
To apply a value to selected, select the rows you want to apply the value to, Then `right-click` the cell containing the value you want to apply and select Apply Value to Selected.

Other options in the Operations context menu are described in more detail in the *Getting Started* guide.

The Plug-Ins Menu



Plug-ins are modular functions that are built onto the system to provide enhanced functionality. One large advantage of plug-ins is that third parties can create software that integrates with GibbsCAM. Another strength is the rapid development of plug-ins, allowing clients' needs to be met quickly. Each plug-in can interact with either geometry, solids, toolpath, the rendered part, or some combination of these items. The actual contents of your Plug-Ins menu depends on your system and specific plug-ins installed.

Two drop-down menus provide access to plug-ins: most commonly used plug-ins are on the left, and Legacy and Custom plug-ins are on the right. Also, you can pin your most-used Legacy plug-ins to a pin-pad, or hide a pin-pad by clicking the pin icon at the top.

The layout and contents of the Plug-Ins menu can be customized using Plug-In Manager. This interface allows you to place plug-ins into folders and subfolders (corresponding to submenus and subsubmenus) and to globally activate or deactivate individual plug-ins and plug-in groups. You are able to suppress or display plug-ins in any order and also group them into sub-menus. Changes made by Plug-In Manager take effect the next time you start the system.

For a full description of each plug-in provided by the software installation (including Operation Modifiers that are implemented as plug-ins), see the *Plug-Ins* guide. This guide is provided with the installation media and is available for download from https://online.gibbscam.com.

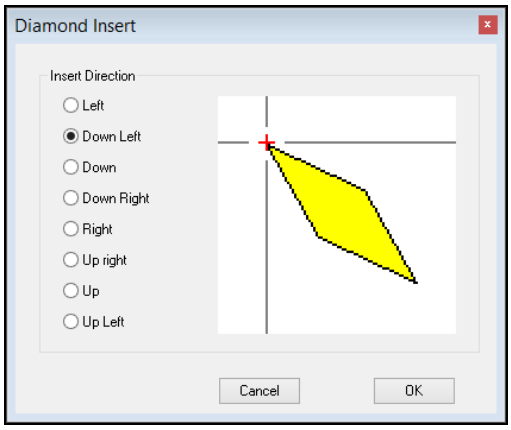
The Macros Menu

The Macros menu provides access to macros that have been installed on your machine's *global data* folder (such as `C:\ProgramData\Gibbs\GibbsCAM\<version>`). By default, one macro, Diamond Insert, is pre-installed to this folder. If you or your administrator have placed other macros under this folder, they will appear under the Macros menu.

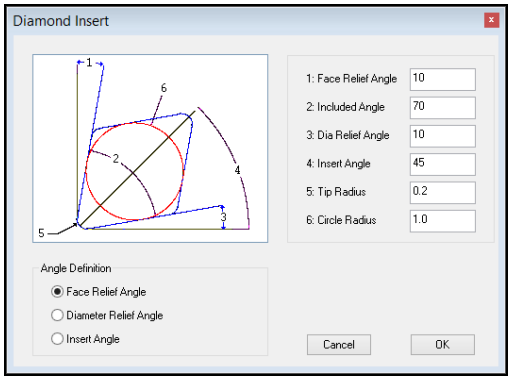
Diamond Insert

The Diamond Insert macro is shipped as a sample, but not pre-configured as a Macros menu item. It lets you create custom diamond-shaped inserts for turning tools. If it is not available on your Macros menu, follow the steps provided in " Configuring the Macros Menu" on page 110.

The initial dialog of the Diamond Insert macro requires you to select a direction for the insert. For example, here is the result of selecting Down Left as the setting for **Insert Direction**.



After you make a choice for **Insert Direction**, click OK. This opens a new dialog box where you define the angle type and specify the angles and radii of the diamond insert.



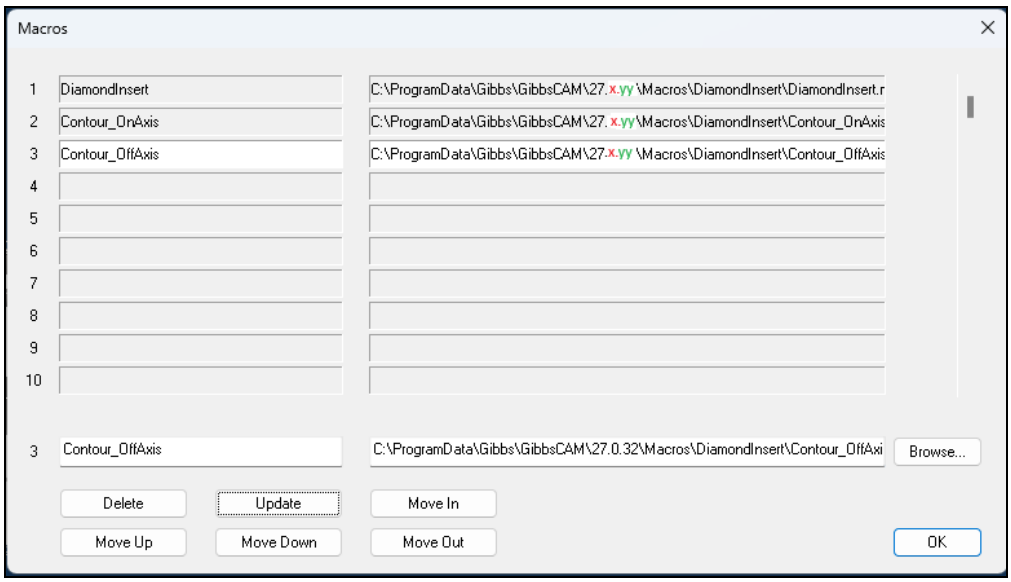
Configuring the Macros Menu

> If you have macros in a previous release of GibbsCAM and you have not yet configured the Macros menu in your current release, you can migrate the old macros using the Migration Tool. If, instead, you *have* already configured your current Macros menu and you want to migrate old macros, see the instructions provided in the Macros wiki: https://Macros.GibbsCAM.com

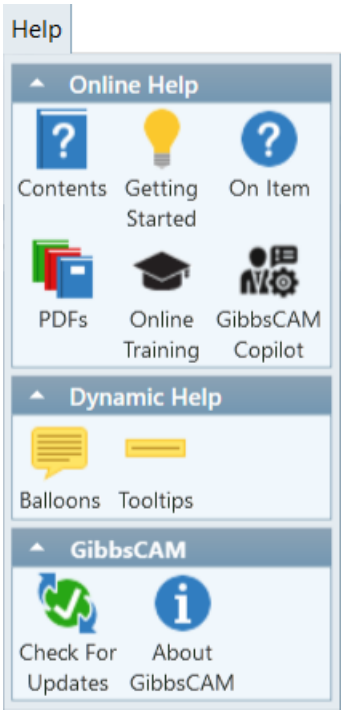
The Macros menu shows all macros that have been installed and configured to appear within the menu. You can use Pathfinder to learn the location of the predetermined Macros Folder — typically under a global data folder like `C:\ProgramData\Gibbs\GibbsCAM\<version>\`**Macros** — but you can use a different location if you prefer. You can add to and customize the menu entries by selecting menu item Configure to open the **Macros** dialog. A sample macro, Diamond Insert, is presupplied under the Macros folder, but the Macros menu will display it only after it has been added to the configuration.

The **Macros** dialog lets you add and organize macros:

- To add a macro, simply select an empty field, enter the name as you want it to appear under the Macros menu, enter the path and filename of the `.mac` file (either by using the Browse button, or by pasting it from the text buffer, or by keying it in), and then click the Update button. Repeat as needed.
- To add a separator line, create an entry whose "name" is a single hyphen (-). Separators help you organize groups of macros.
- To move an entry above or below other entries, select it and then click Move Up or Move Down. Repeat as needed.
- To move an entry subordinate to its predecessor, select it and then click Move In. To move an entry one level higher in a stack of sub[sub]entries, select it and then click Move Out.
- When you are done making changes, click the OK button. This closes the dialog and immediately applies your changes to the menu structure under the Macros menu.



The Help Menu



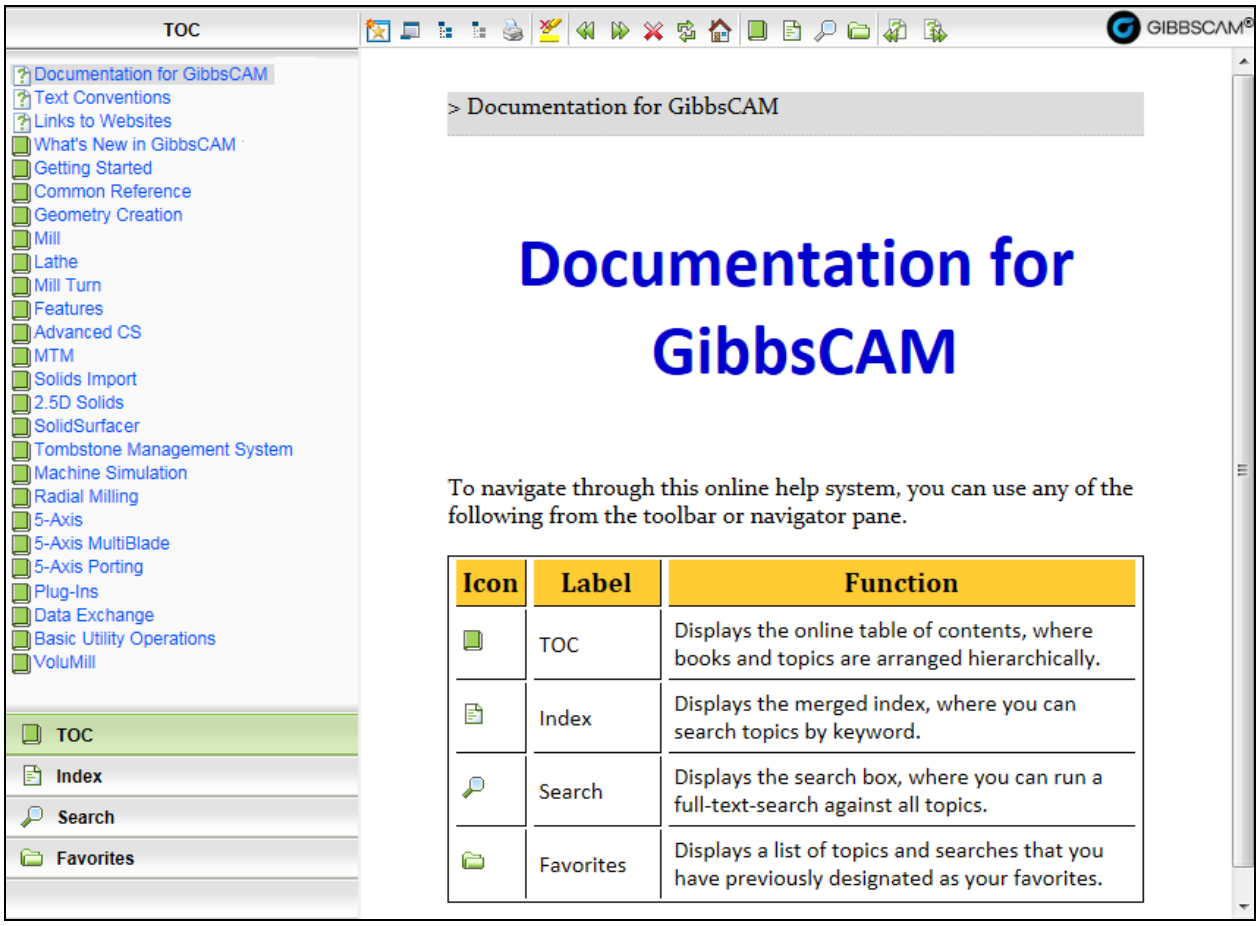
The Help menu contains options to help you use the system and provides additional information about the application.

- Online Help, next
- Dynamic Help
- GibbsCAM
- About the Online Help

Online Help

Contents

This activates the system's online help. A new window opens that lets you view, browse, or search for information.



For details, see About the Online Help.

Getting Started

This launches Adobe Reader and opens the PDF version of *Getting Started*.

On Item

This has the same effect as clicking the help button tool in the GibbsCAM Floating Toolbar.

When activated, the cursor changes to the On Item Help cursor . With this cursor active, clicking an item in a dialog, palette or window in GibbsCAM opens the online help to the section that is relevant to what you clicked. This is known as context-sensitive help.

2027 PDFs

This launches Adobe Reader and opens a “light” version of a PDF organizer that references a few of the books in the GibbsCAM documentation set. (The full version of the master PDF is available from the GibbsCAM website, gibbscam.com, or when you install the complete documentation set.)

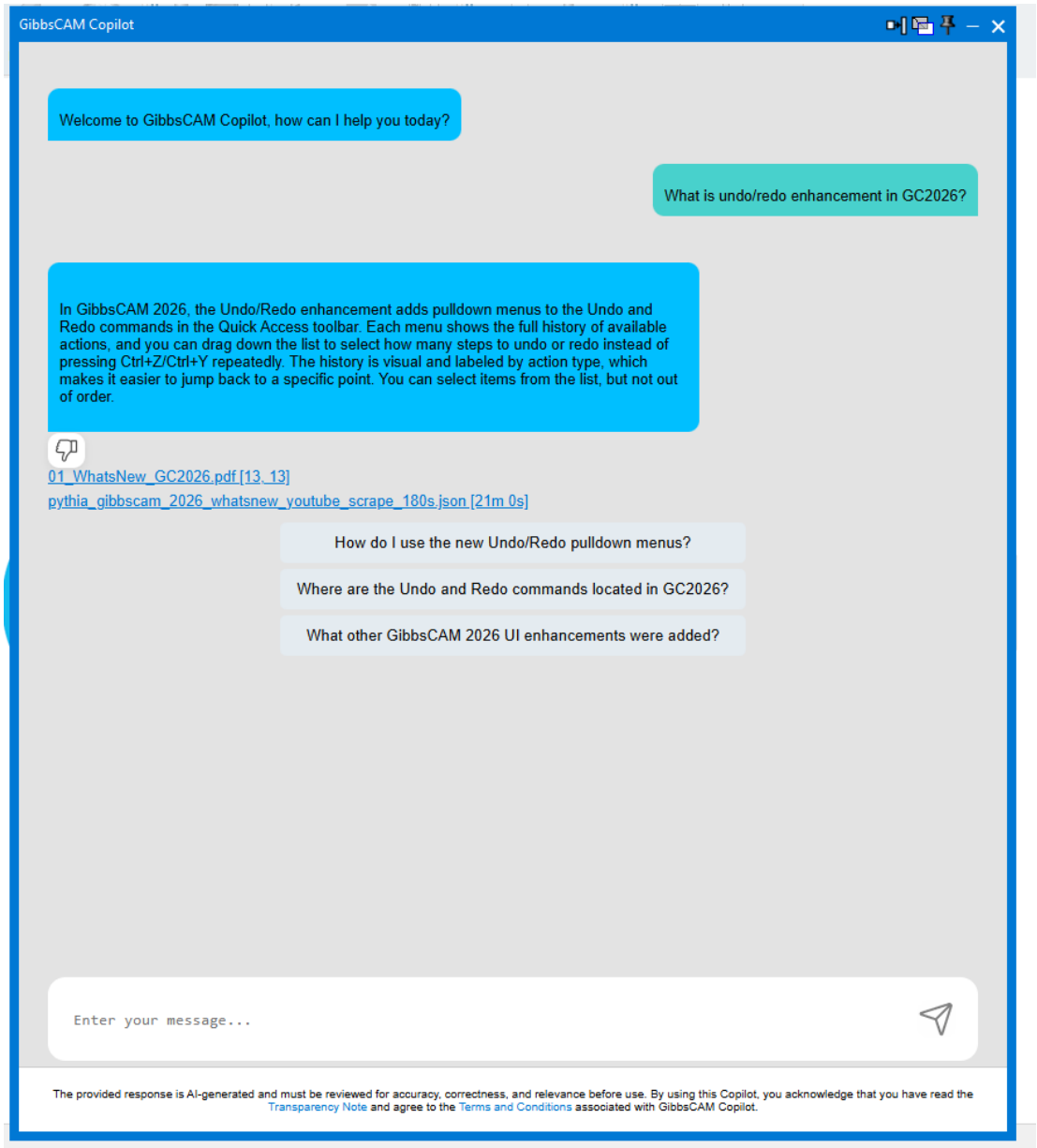
Online Training

This opens a web browser to view the courses available from Cognus Online Training. Free courses include: "Production Milling" "Production Milling/Turning", and "Whats New" for older versions of GibbsCAM. A subscription-based training program is available for advanced options such as 2.5D Solids, 3D Machining, and more.

GibbsCAM Copilot

GibbsCAM **Copilot** is an AI-powered chat tool that helps you quickly access GibbsCAM-related information without searching through documentation or contacting Technical Support. Copilot responds to queries in any language supported by the GibbsCAM user interface and works best with specific, narrowly worded questions.

Copilot provides responses with enhanced formatting for improved readability and includes links to supporting and related documentation. It also offers intelligent suggestions and references to relevant videos, including direct links to the appropriate timestamp, enabling you to quickly locate and understand the required information.



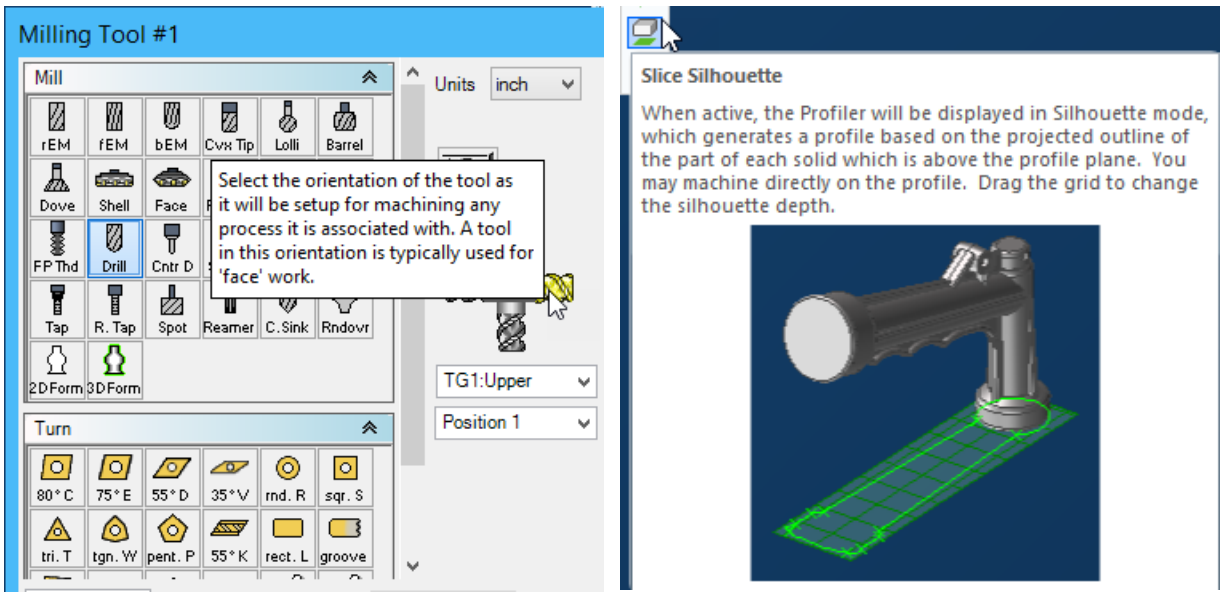
If a response is not helpful, you can use the Thumbs Down feedback option to provide feedback and help improve future responses.

Please note that, because some countries' laws relating to AI are not yet fully analyzed, the GibbsCAM Copilot chatbot is disabled in some countries.

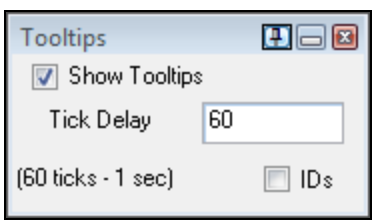
Dynamic Help

Balloons

This button toggles the on-screen help built into the system. When this item is checked, balloons containing reference information will come up on the screen whenever the mouse cursor is moved over an input box, command or other object (and not clicked). Use (Ctrl+B) to turn this option off.

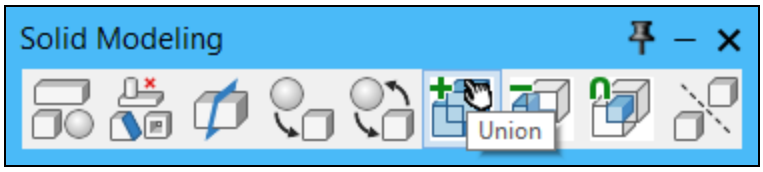


Tooltips



This option summons the **Tooltips** dialog. Tooltips show the name of a button or item that the mouse is held over. The **Tooltips** dialog allows you to set whether tooltips are shown, and to set the delay before tooltips are shown. Tick Delay sets the delay, where each "tick" is 1/60th of a second. The IDs item is not of any real use to end-users: it displays the identifying number of a dialog's components, rather than a description. It is a development tool.

The following image provides an example of a tooltip. It shows the cursor being held over a button in the **Solid Modeling** dialog. The name of the button is displayed in a white box next to the hand cursor.



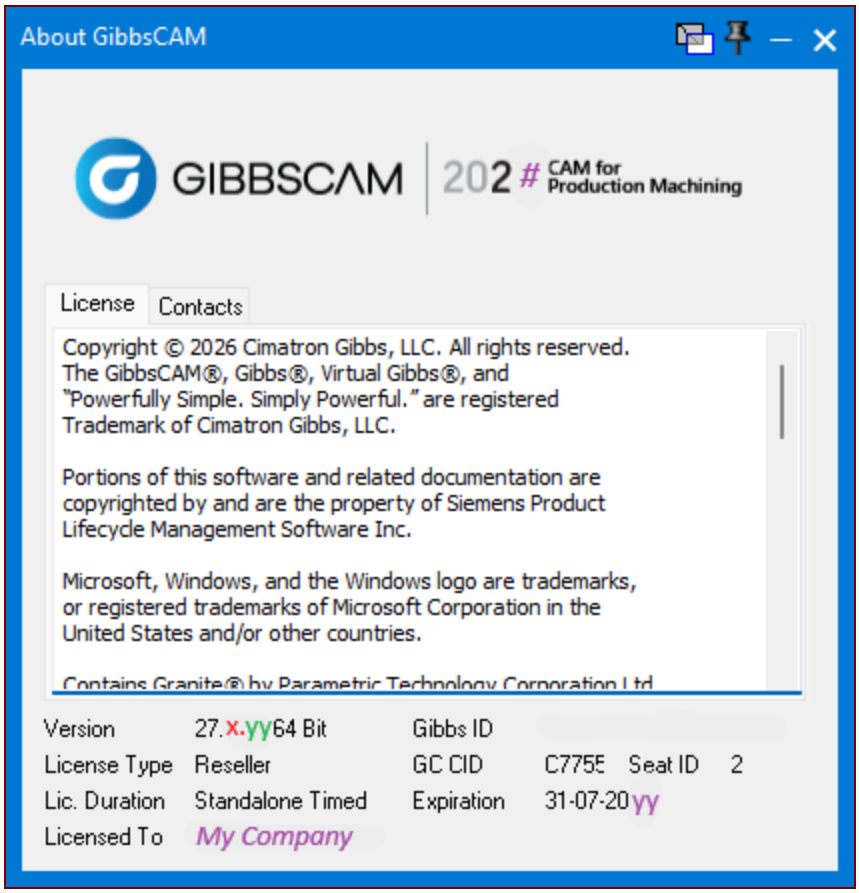
GibbsCAM

Check for Updates

Clicking this item causes the system to perform a one-time check for updated releases. If a later version of GibbsCAM is available, a message dialog appears.

You can configure whether or not to check for updates each time GibbsCAM is started: File > Preferences, Interface tab, checkbox Automatically Check for Updates on Startup.

About GibbsCAM

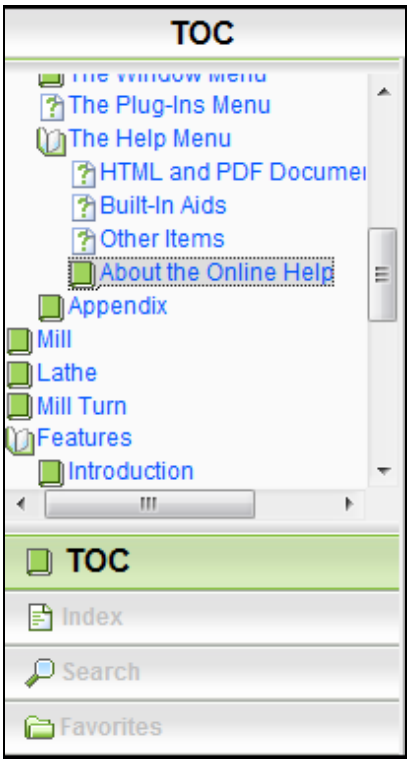


The About dialog provides information about your GibbsCAM software as well as phone numbers and e-mail contacts if have questions or problems.

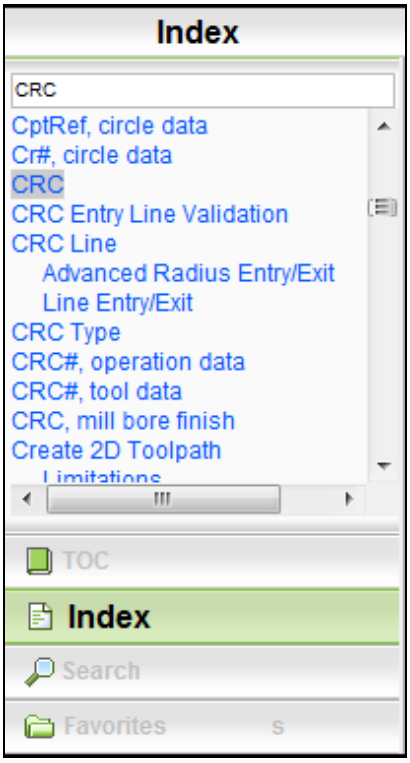
The About box displays your Gibbs ID, which can be copied and pasted directly from this dialog. If applicable, your HASP key expiration is also displayed here.

About the Online Help

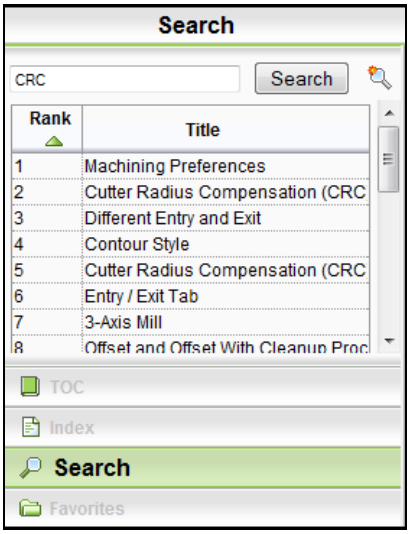
The system provides several navigation panes for working with online help:



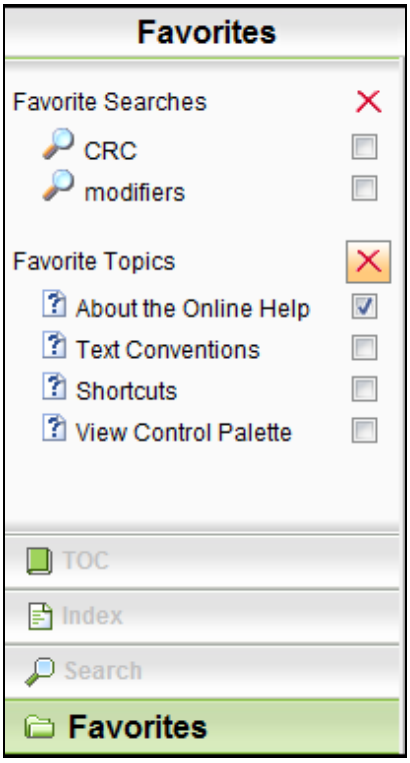
The icon on the Help toolbar activates the TOC pane, which presents a hierarchical list of help topics organized into folders and subfolders like a table of contents.



The icon on the Help toolbar activates the Index pane, which presents an alphabetized list of keywords, each one linked to one or more help topics.



The icon on the Help toolbar activates the Search pane, which uses a term that you enter to construct a list of all topics that contain the term. The list is ranked by the number of times the term occurs within the topic.



The icon on the Help toolbar activates the Favorites pane, which lists searches and topics that you have previously bookmarked.

To add a favorite *topic*, click (on the Help toolbar). To add a favorite *search*, click (in the Search pane). To delete a favorite, select the corresponding checkbox and then click (in the Favorites pane).

Other useful tools in the Help toolbar include the following:

- Add the current topic to your Favorites
- Show or hide all navigation panes.
- Expand all folders and subfolders.
- Collapse all folders at the current level.
- Print the current topic.
- Find a text string within the current page.
- Remove highlighting from search hits on the page.
- Browse back to the topic last visited.
- Browse forward to the topic next visited.
- Stop loading the current topic.
- Refresh the display of the current topic.
- Jump to the home (root) topic of the online help system.
- Jump to the previous topic in sequence.
- Jump to the next topic in sequence.

Troubleshooting the Online Help System

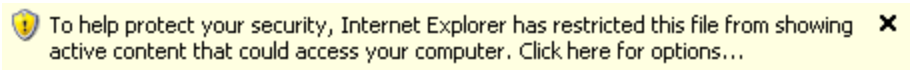
Changing the Text Size

To temporarily increase the text size in online help, press [CTRL +].

To temporarily decrease text size, press [CTRL -].

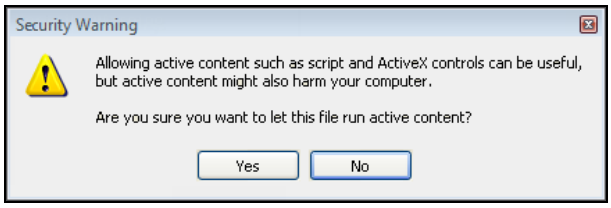
Generally the size of the text in the Help window text is determined by the size of the default text set for Windows.

Allowing Blocked Content

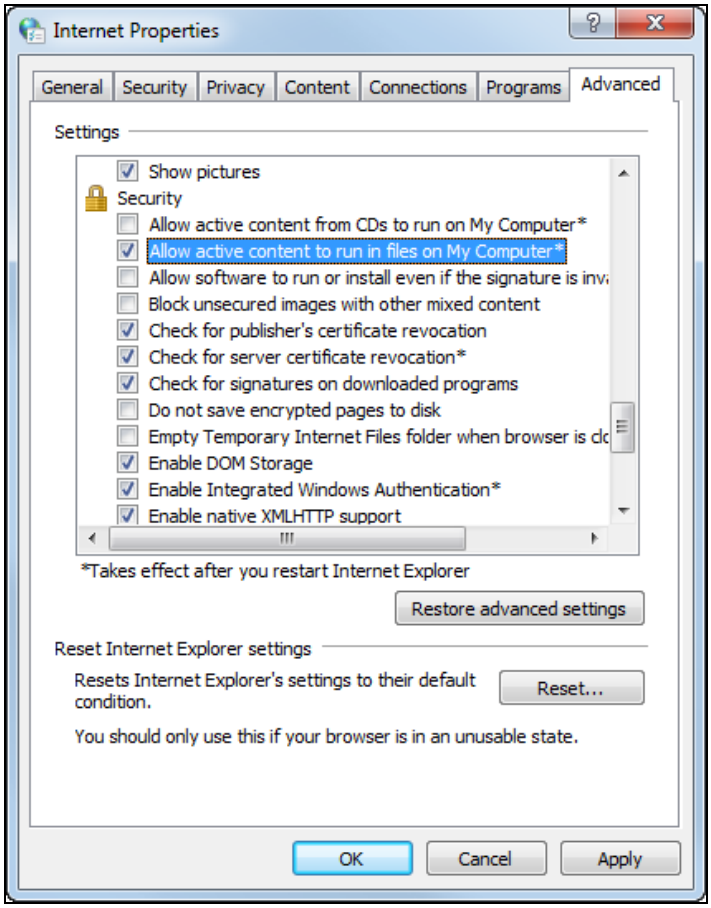


Depending upon the security settings for your operating system you may receive an alert warning you about Active Content. This is generated because of the installation of the JRE. You can easily disable this warning and safely enable the JRE.

- Clicking on the warning brings up a menu. Select Allow Blocked Content...



- You will receive a Security Warning. Click Yes to continue.



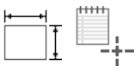
- To avoid this error message in the future, open the Internet Properties Control Panel.
- Select the Advanced tab and scroll down to the Security section (it is at the bottom of the list.)
- Select the Allow active content to run in files on My Computer.
- Click Apply and then click OK.

Floating Toolbar

1. Show Geometry
2. Show Dimensions...
3. Show Holes
4. Show Solids
5. Toggle Render ...
6. Show Section View
7. Indicate Sheet Side
8. Wrap WGs
9. Show Stock & Origin
10. Toggle CS Plane/.../Grid
11. Face Selection
12. Edge Selection
13. Toggle Profiler: ...
14. Pre-Selection Highlighting
15. Hidden-line Toolpath
16. Show Machine Assembly
17. Show Machined Stock
18. Show Tool On Toolpath
19. Toggle TP Color Mode
20. Show Clearance ...
21. Toggle Color Mode
22. Viewport Manager
23. Unzoom (Ctrl + U)
24. Get Help on Item

Show Geometry

When this button is active, all geometry in the currently selected coordinate system and workgroup is shown in the drawing window.

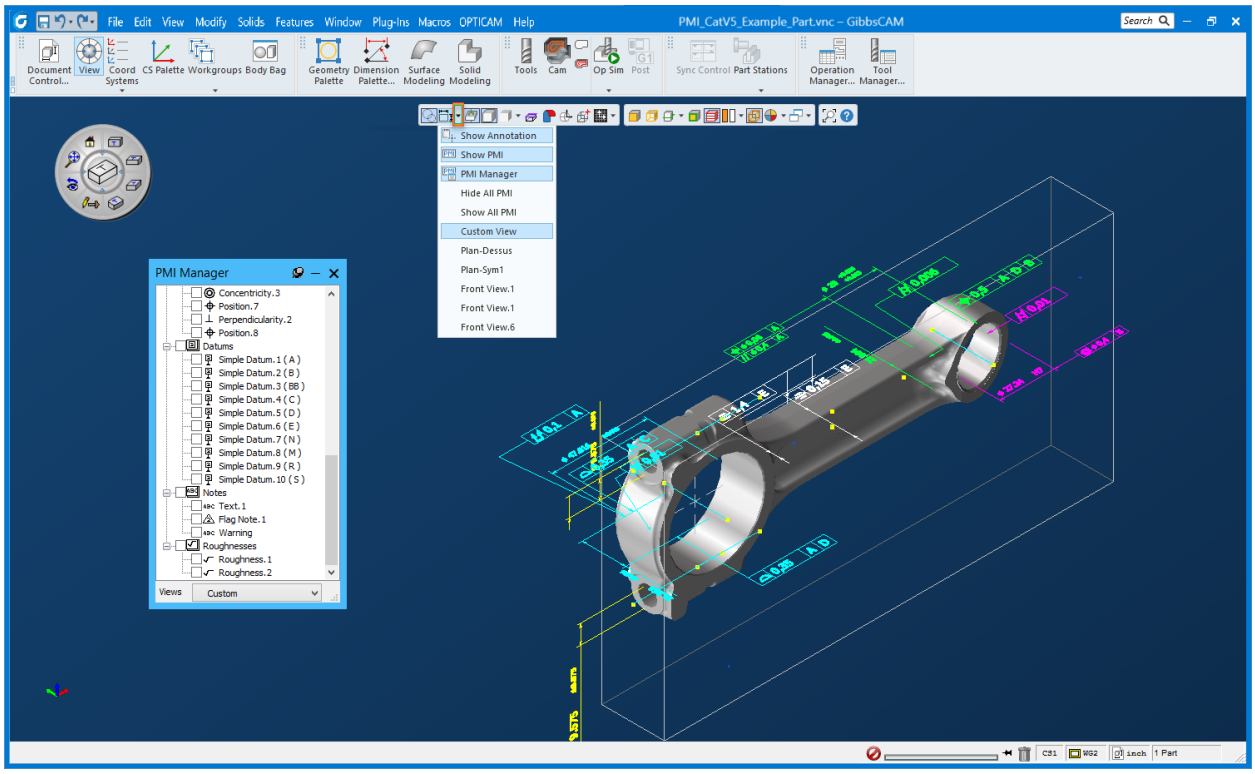


Show Dimensions/Annotations

For a model that has no PMI, when Show Dimensions is active, all dimensions in the currently visible workgroups are shown in the workspace. If Show annotations is active, both dimensions and annotations are shown.

Pull-down menu choices for PMI

When an imported GibbsCAM model contains PMI, you can hover over the **Dimensions** dropdown button to open a menu of options relating to dimensions and PMI.



PMI-related options under the **Show Dimensions** button; **PMI Manager**; and GibbsCAM solid model with PMI displayed

Show Dimensions

When checked, non-PMI annotations are displayed.

Show PMI

When checked, PMI is displayed using the most recently used PMI view.

Manage PMI

Opens the **PMI Manager** dialog box.

PMI Manager displays all PMI items in a tree that is organized into the following categories:

- Dimensions: Dimensioning information associated with the PMI, not with the GibbsCAM model.
- GeomTols: Geometric tolerancing information
- Datums: Datum information for GD&T (geometric dimensions and tolerancing).
- Notes: Annotations associated with the PMI, not with the GibbsCAM model.
- Roughnesses: Information regarding surface texture and finish: smoothness, waviness, and the like.

Tree Controls for PMI Categories

- To expand a branch to see all items in a category, click the symbol.
- To collapse a branch and hide all items in a category, click the symbol.

Display Controls for PMI Categories and Items

- To display all items in a category, or to display a particular PMI item within a category, select the corresponding checkbox.
- To hide all items in a category, or to hide a particular PMI item within a category, clear the corresponding checkbox.

PMI Views

- If the imported model contains specific groupings of PMI items into named views, you can click the Views button to list the defined views. Selecting a view rotates and re-centers the workspace display and shows only those items of PMI associated with that named view.
- If you make changes to the workspace display or the shown/hidden statuses of PMI items or categories and items, the changes are saved in a special view named Custom.

Hide All PMI

Clears all checkboxes in PMI Manager, thus hiding all PMI.

Show all PMI

Selects all checkboxes in PMI Manager, thus showing all PMI.

Custom View

When checked, PMI is displayed using the most recent PMI custom view.

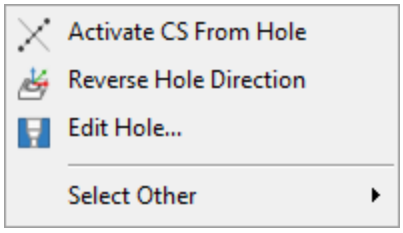
<viewname>

When checked, PMI is displayed using the named PMI view.

Show Holes

When this button is active, hole features are displayed in the workspace.

Holes are displayed in different colors, depending on the hole type. (See " Hole Manager" on page 226.) In the Level 1 Interface, the hole is displayed as a dot. In the Level 2 Interface, the hole feature is displayed and right-clicking the hole feature provides a menu as illustrated.



Activate CS From Hole

When clicked, GibbsCAM activates the CS associated with the hole selected. It will be displayed on the screen and highlighted in the CS list.

Reverse Hole Direction

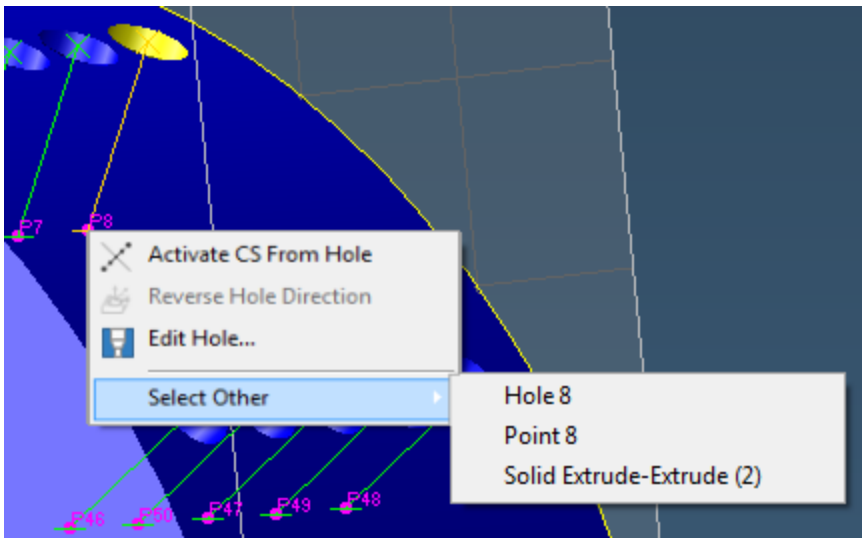
This is not available for blind holes and will be greyed out.

Edit Hole Profile

Opens the Edit Hole Profile dialog.

Select Other

Enables clarification of the selection - if the point is clicked you can choose to select the Hole, Point, or Solid. If the Hole Feature is clicked you can choose to select the Hole or Solid.



Show Solids

This button is very similar to the Show Geometry button, except that it affects bodies (that is, solids and sheets). When turned on, all bodies are displayed in the workspace. If it is off, bodies are not displayed. The display of bodies in the Body Bag is unaffected. This button is available with any of the solids options — Solids Import, 2.5D Solids, and SolidSurfacer.

Render Solid Faces

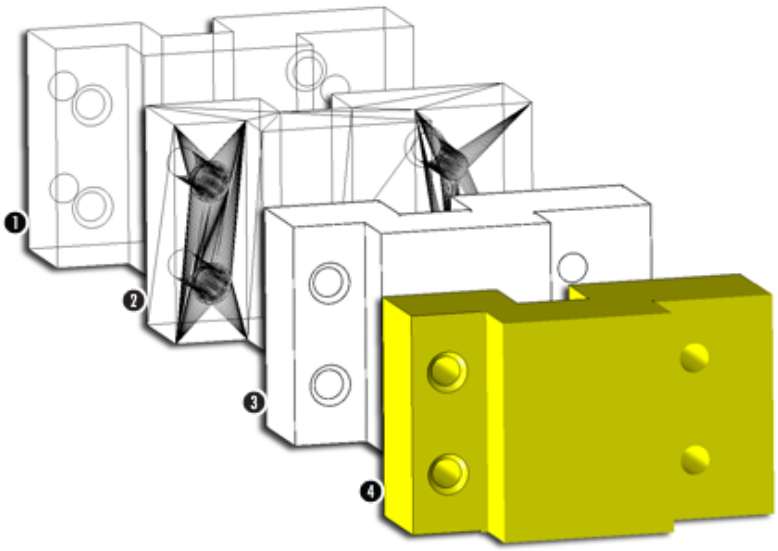
This button has four states that control the rendering of bodies.

Render Solid is the default. It shows solids and sheets as normal rendered bodies. This button is available with any of the solids options.

Render Solid Faces/Edges renders the bodies but shows the edges of the bodies.

Render Hidden Edges renders bodies as hidden wireframe objects — the “hidden” edges are not shown.

Render Wireframe renders all solids and sheets as wireframe objects. This wireframe state will be either edges or facets, depending on the File> Preferences > Display tab > Bodies > Wire Drawing preference setting.

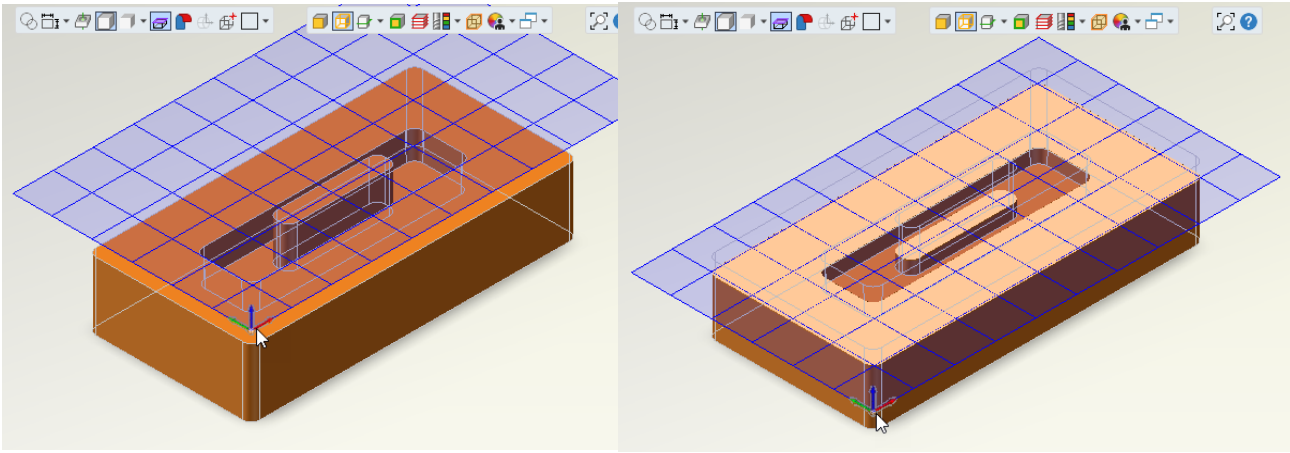


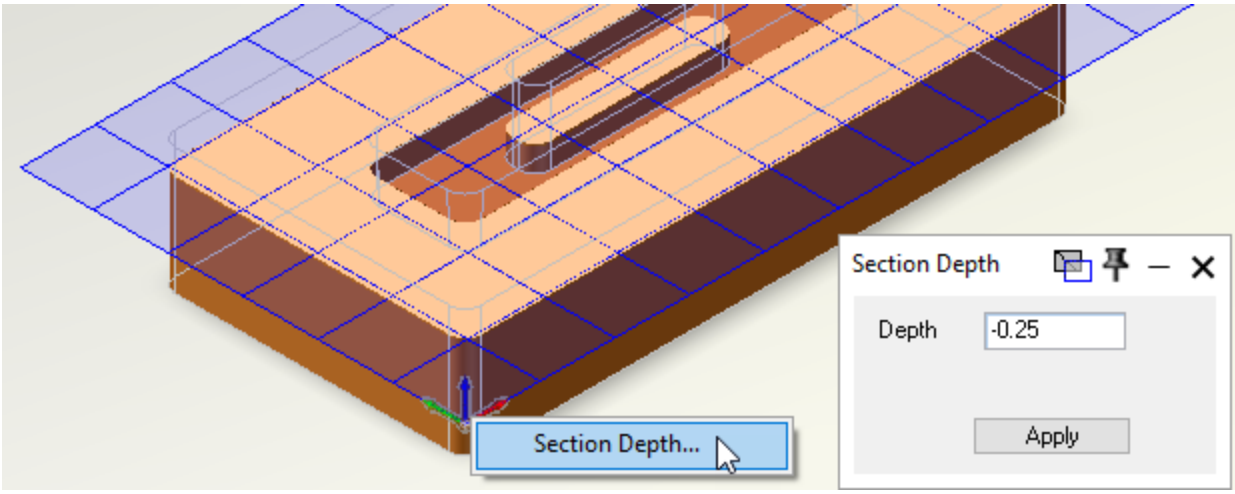
1. Render Wireframe
2. Render Hidden Edge
3. Render Faces/Edges
4. Render Solid

Show Section View

This button is available only with the Solids Import or SolidSurfacer options. It lets you display only the portion of a solid that lies below a defined plane.

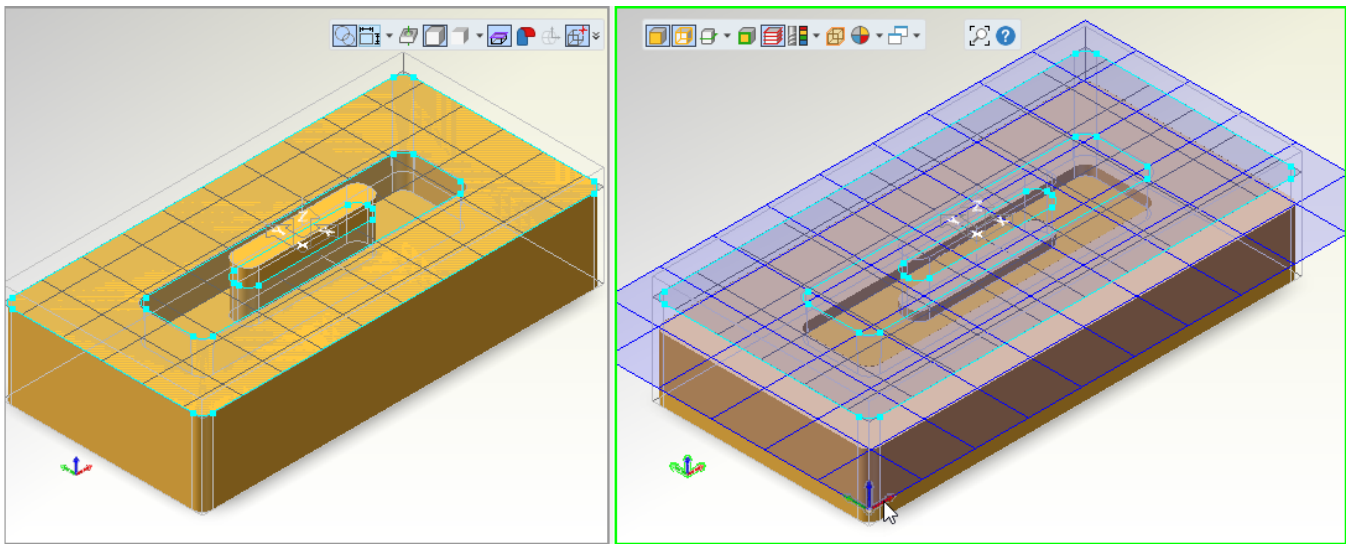
You can dynamically modify this by dragging the plane of the section view and/or by entering a value for Section Depth, as shown in the following illustrations.





Although the orientation and position of the section plane is associated with the CS, you can toggle the section view on or off in individual viewports, as shown in the following illustration.

Left viewport: Section View OFF	Right viewport: Section View ON



Indicate Sheet Side/Annotations

This button is for use with sheets. It differentiates between the outside and inside of a sheet by displaying them in different colors. The outside of the sheet will remain the standard blue that is assigned to all sheets, while the inside will be displayed in red. The outside of a sheet is the side from which the positive direction of the surface normal is projected. The other side of the surface normals are projected out from the inside of the sheet. The sheet side is important when performing such functions as offsetting sheets or solidifying sheets using either the offset or extrude option. This is due to the fact that sheets are offset or extruded in the positive direction of their surface normal. It is possible to toggle the inside and outside of a sheet, using the

Toggle Sheet Side item in the Modify menu. This button is available only with the Solids Import or SolidSurfacer options.

Wrap Workgroups

When Wrap WGs is selected, geometry in the current workgroup at a depth other than Depth 0 is wrapped around a radius equal to the geometry's depth. This button is available if Wrapped is enabled. The Wrapped function provides continuous C-axis rotation on milling operations, referred to as wrapping. When the Wrap WGs button is on, geometry is created and viewed radially, wrapped around the part. Geometry can be defined using XZC coordinates, where X designates the radius or diameter. In order to create and view wrapped geometry, a rotary machine must be selected and the Workgroup must be defined as Wrapped in the WG Info dialog. For more information see the Polar & Cylindrical Milling section of the *Mill* guide.

Show Stock & Origin

When this button is depressed (ON), the stock outline and origin are shown on the screen. This button provides the same function as the Show Stock & Origin option in the View menu.

Toggle CS Plane/.../Grid

This button controls the visibility of coordinate systems and associated CS grids and planes. The button has four states.

Hide CS Grid. This hides the grid and the plane.

CS grid + plane. This shows the CS grid and axis markers for the current coordinate system. The CS grid and plane will be drawn through the origin of the CS.

CS Plane. This hides the grid but shows a shaded plane, representing the coordinate system.

CS Grid. This shows only shows the grid and not the plane.

Face Selection

When this button is depressed (ON), the system is in face selection mode. Clicking a body (that is, a solid or sheet) will select only the face and not the entire body. This button is available with any of the solids options.

Edge Selection

When this button is not depressed (OFF), the workspace does not display edges of solids or sheets, such as curves where faces intersect. When it is depressed (ON), edges are displayed and can be selected. This item is available with any of the solids packages.

The button has two states:

- Visible Edge Selection shows only edges that are not obstructed. Edges that are behind the body are hidden, making it easier to select the correct edges on a complex solid model.
- Edge Selection shows all edges.

In order to select edges for such functions as blending, geometry extraction, and body unstitching, you must be in Edge Selection mode; in other words, one of the states must be ON. Also, when using the stitching function, Edge Selection mode is useful for viewing the external edges of a body.

If an edge is double-clicked, the system will attempt to select an entire loop that contains the selected edge. This edge selection will stop when it has more than one good choice at a vertex. In some instances, it may take a few double-clicks on different edges to select an entire loop, but it is still much faster than attempting to select edges individually.

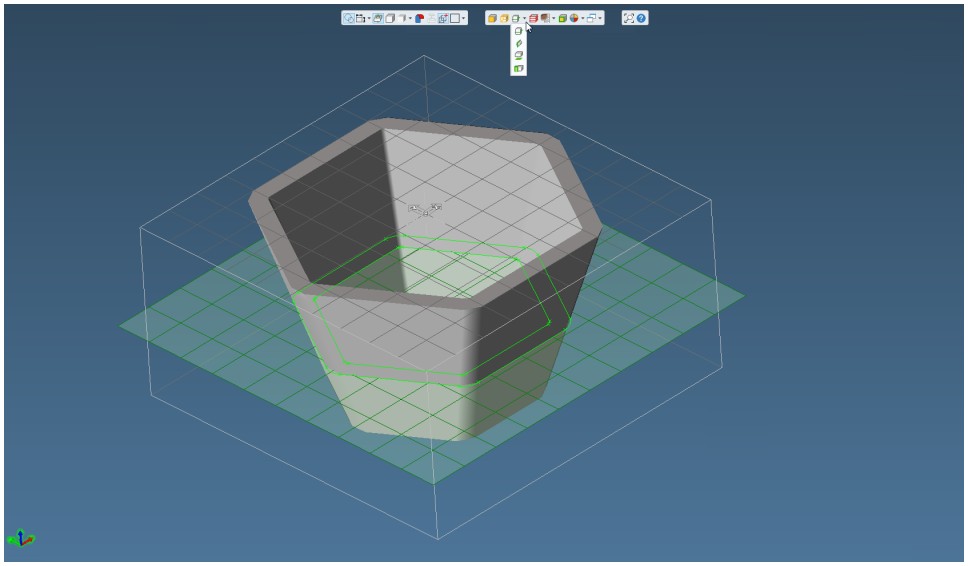
Toggle Profiler: Slice Plane / Slice Spun Body / Slice Silhouette / Slice Cylinder

The Profiler is a moveable cross-section tool that can be toggled off or on in any of four different modes.

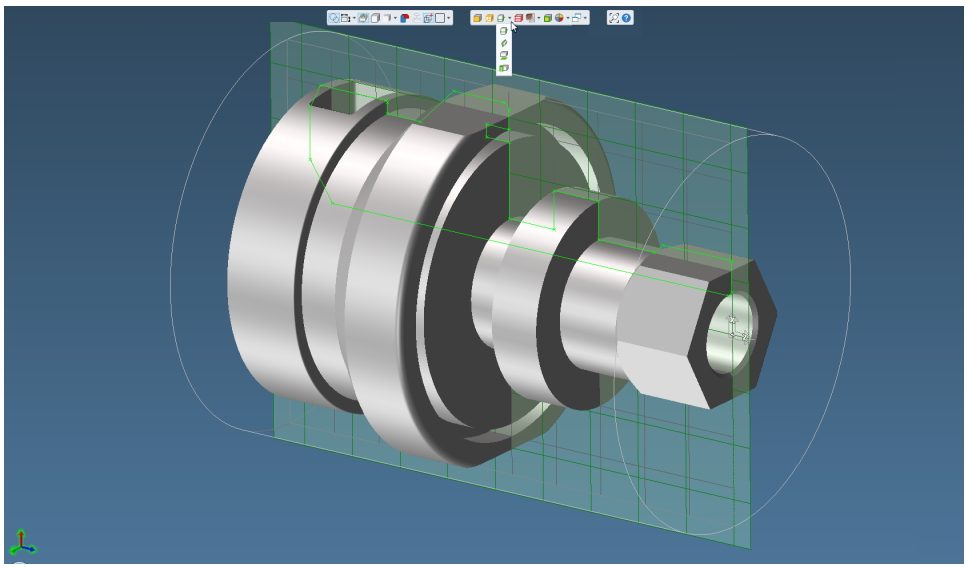
The Profiler's main use is to set machining markers on a solid. The Profiler can also be used to create geometry: Drag the profiler and release it; right-click a green geometry profile and, on the context menu, choose Extract Geometry; when you click the dialog's Do It button, the system creates geometry for the selected geometry profile. This item is available with any of the solids packages.

To change modes, hover over the dropdown arrow and choose from the list.

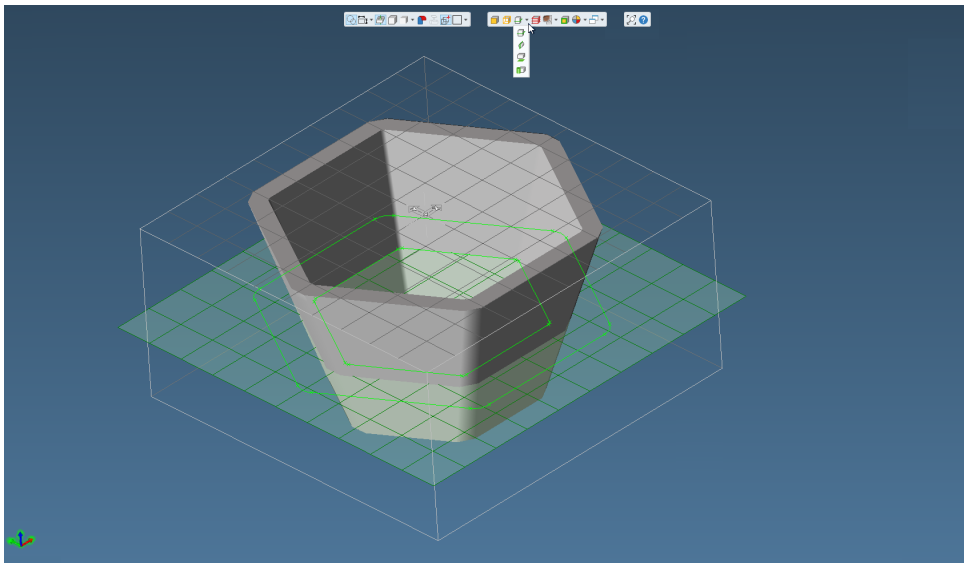
In Slice Plane mode, the Profiler is planar to the current CS. To dynamically display the part's cross-section geometry, drag the slicing plane up or down along the depth axis.



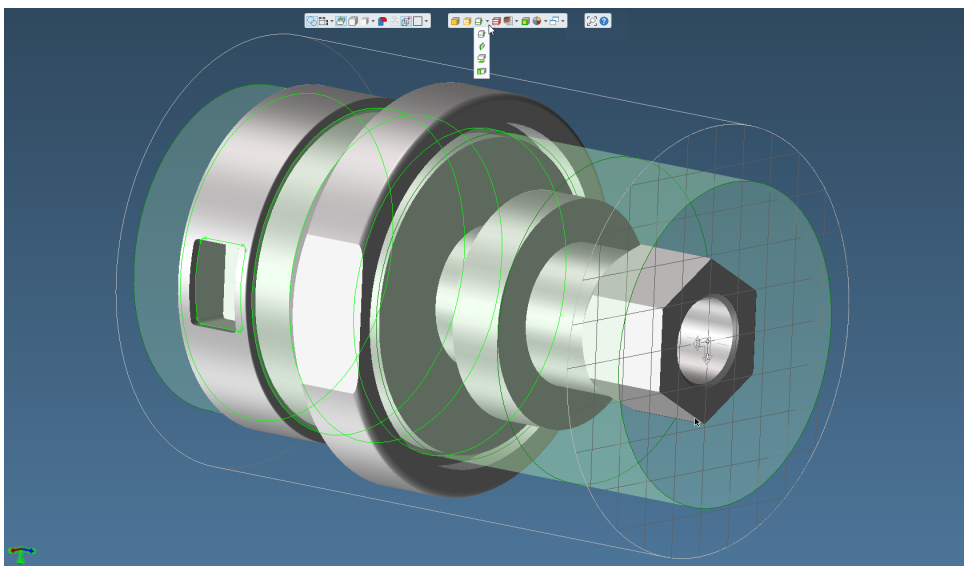
In Slice Spun Body mode, the Profiler is planar to the current CS, and slices the outline of the body that would result from spinning all elements around the rotary axis.



In Slice Silhouette mode, the Profiler is planar to the current CS, and displays the outline of the "shadow" cast by all bodies from rays of light normal to the plane and shining down on it. Each body (even a multi-lump body) casts a single shadow with a single outline. Multiple bodies cast separate shadows with possibly overlapping outlines.



In Slice Cylinder mode, the Profiler is cylindrical perpendicular to the current CS. To radially expand or contract, drag the slicing cylinder outwards or inwards.

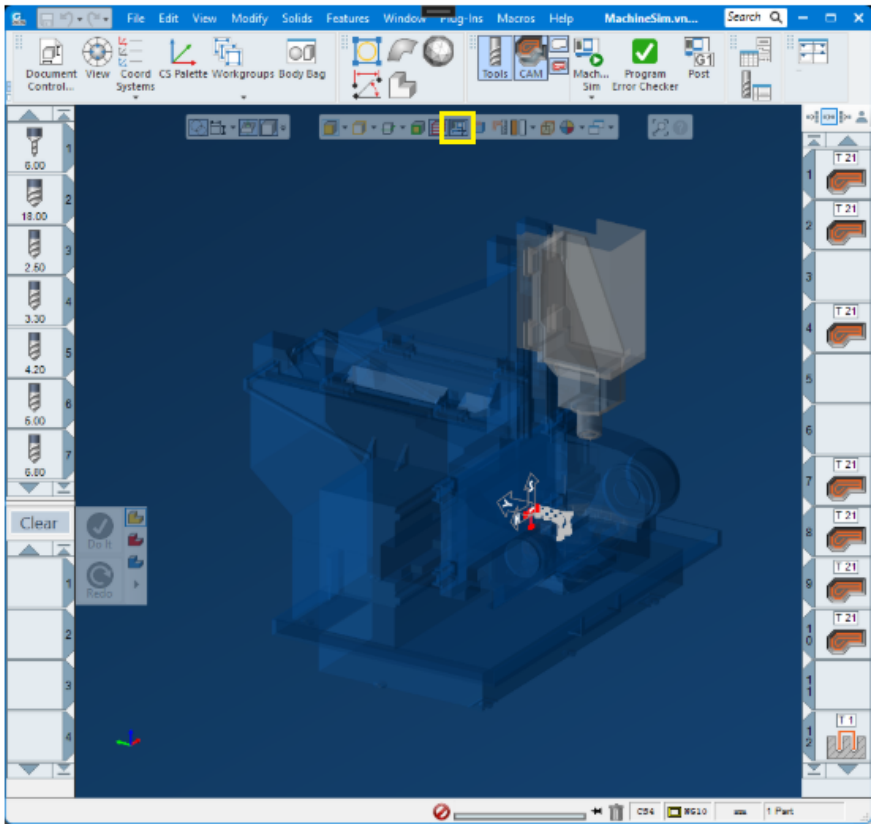


Hidden-Line Toolpath

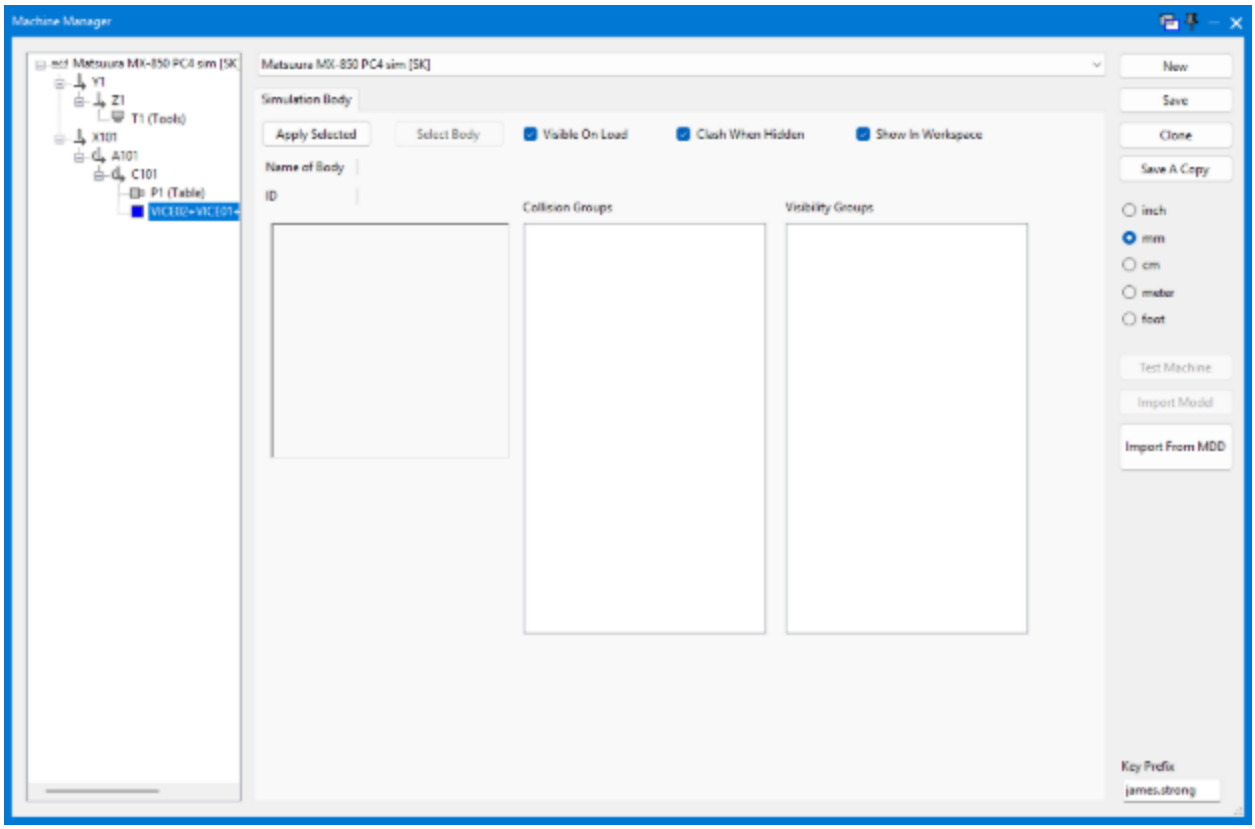
This button sets the mode to hide/show toolpath with hidden-line removal activated. When on, toolpath that lies behind solid objects is not shown.

Show Machine Assembly

By clicking on Show Machine Assembly command, the machine model is displayed in the workspace.



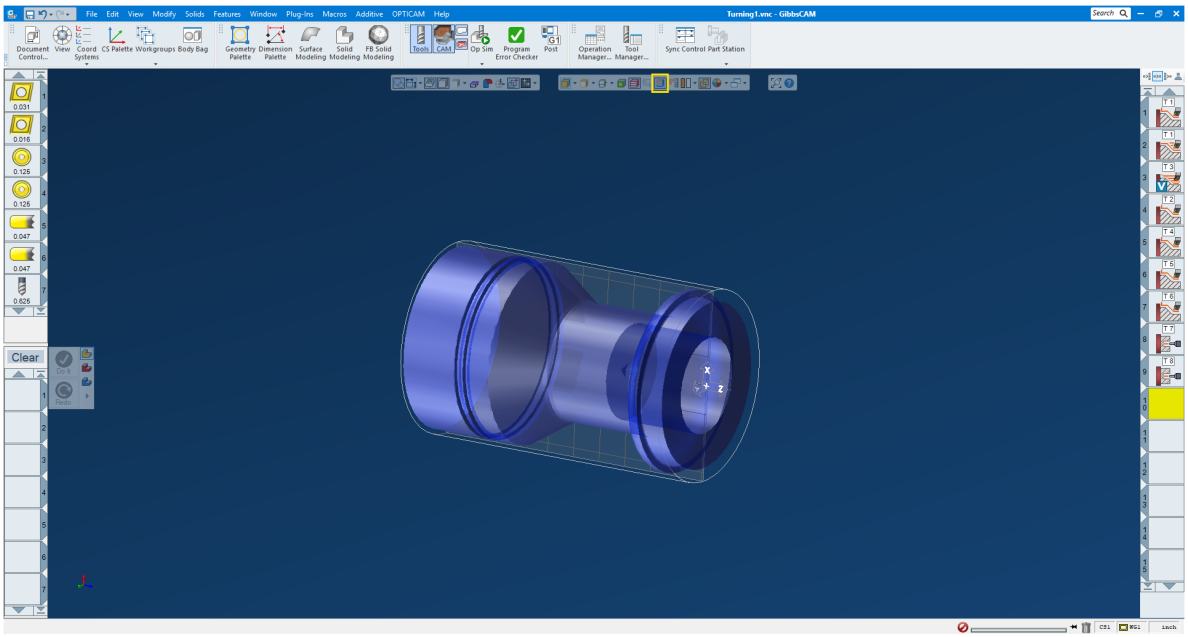
By default, all machine assembly components are displayed in the workspace. However, specific components can be excluded from the visualization using below method -
Create a machine assembly visibility group named Workspace and add only the required machine components to this group. When the Workspace visibility group is present, any components not included in the group will not be rendered in the workspace.



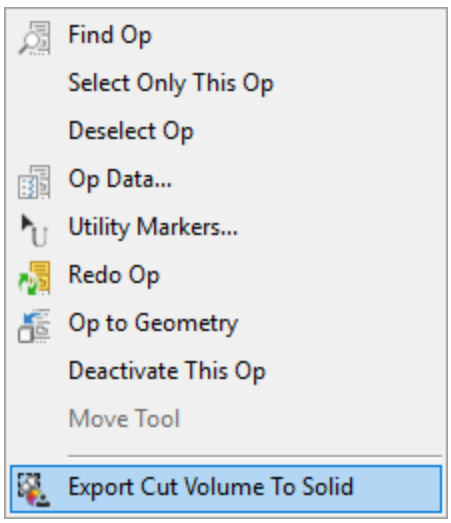
Show Machined Stock

The Show Machined Stock command displays the machined stock for the highest-numbered selected operation or the entire program when no operation is selected.

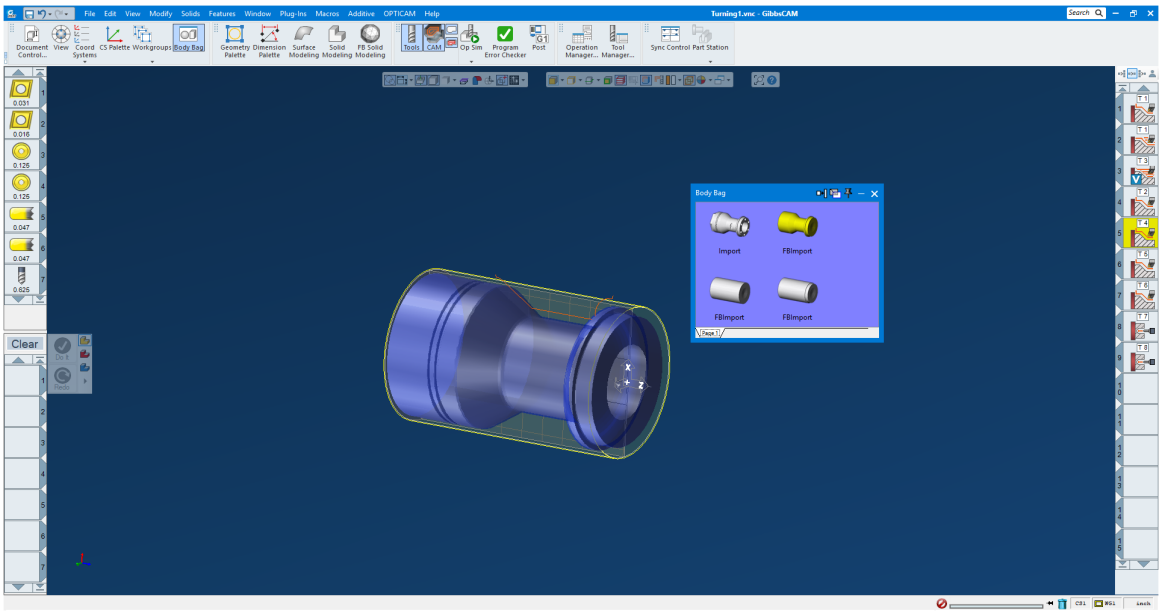
The Show Machined Stock option displays the work-in-process stock state after the selected operation directly in the workspace without running a full Machine Simulation.



By Right clicking on the Machine stock while one of the operation is selected these options appears :



You can Export the cut volume of stock to solid with its corresponding operations selected.



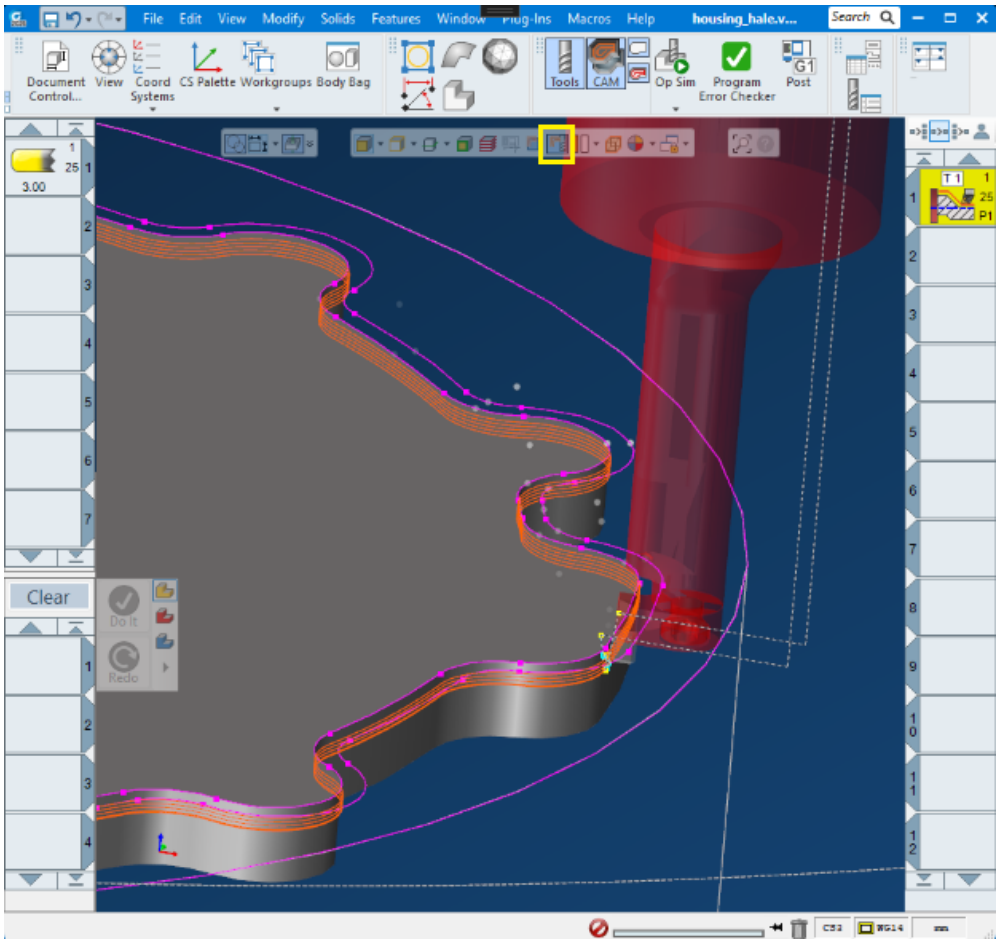
Note:

1. The Machine Stock option gives priority to custom stock instead of system (DCD) stock when both stock definitions are available.

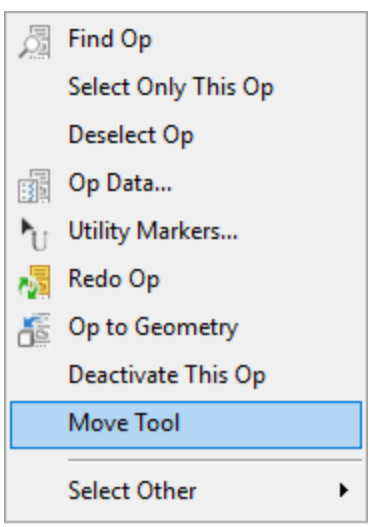
2. **Show Machined Stock** does not render Lathe Threading, Linear Broaching, Rotary Broaching, or Thread Whirling operations within the Workspace. This functionality is intended to align with Material Only visualization and should not be interpreted as equivalent to, or a substitute for, Operation Simulation (Op Sim).

Show Tool On Toolpath

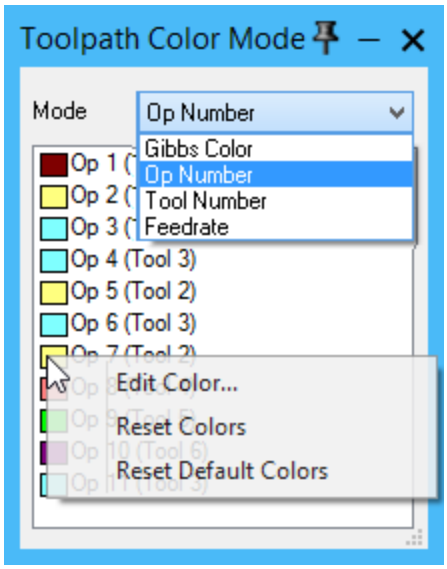
The Show Tool on Toolpath command displays the tool for the selected operation in the workspace and allows interactive movement along the toolpath.



Click the desired location on the **Toolpath**, then right-click and select Move Tool. The tool moves to the selected location on the Toolpath.



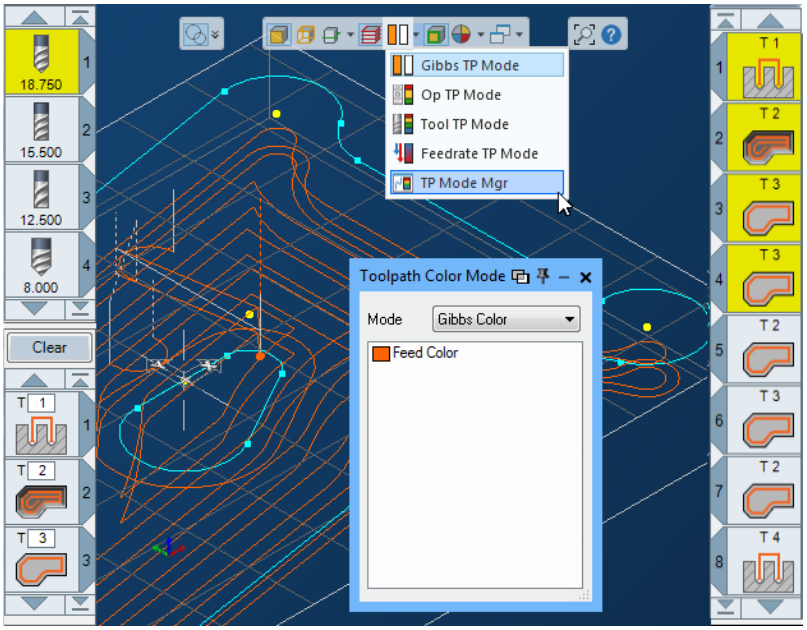
Toggle Toolpath Color Mode: Gibbs/Op/Tool/Feedrate / TP Mode Manager



In addition to the standard red GibbsCAM toolpath color (Gibbs TP Mode), toolpath can be displayed with the Operation numbers (Op TP Mode), Tool numbers (Tool TP Mode), or Feedrate (Feedrate TP Mode) highlighted in user-editable colors.

The colors for each mode are managed by the Toolpath Mode Manager, as shown in the following illustrations.

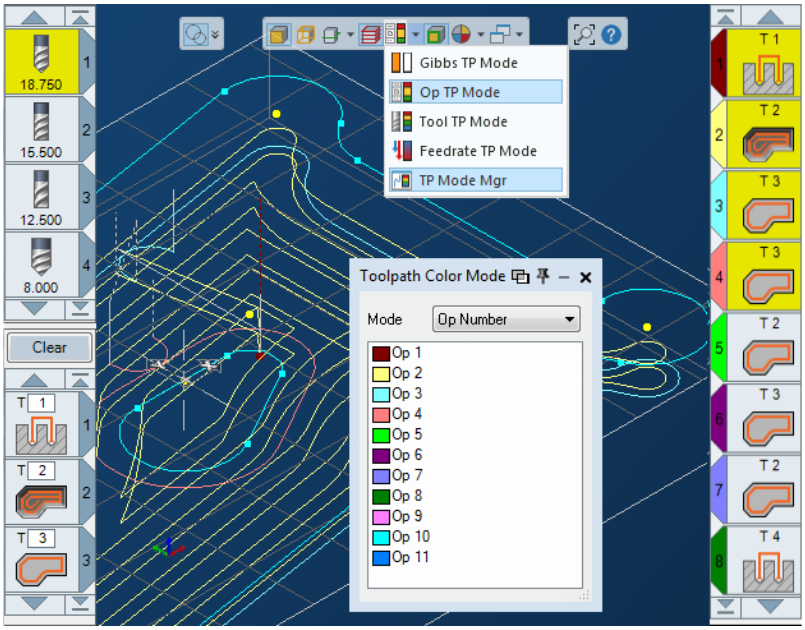
Gibbs Toolpath Color Mode



Gibbs TP Mode is the toolpath display used in previous versions of GibbsCAM:

- Interop and entry/exit moves are all in one color. (default: white)
- Op moves are all in another color. (default: orange)
- Rapid moves are all in the same color as feed moves, but drawn with dashes.

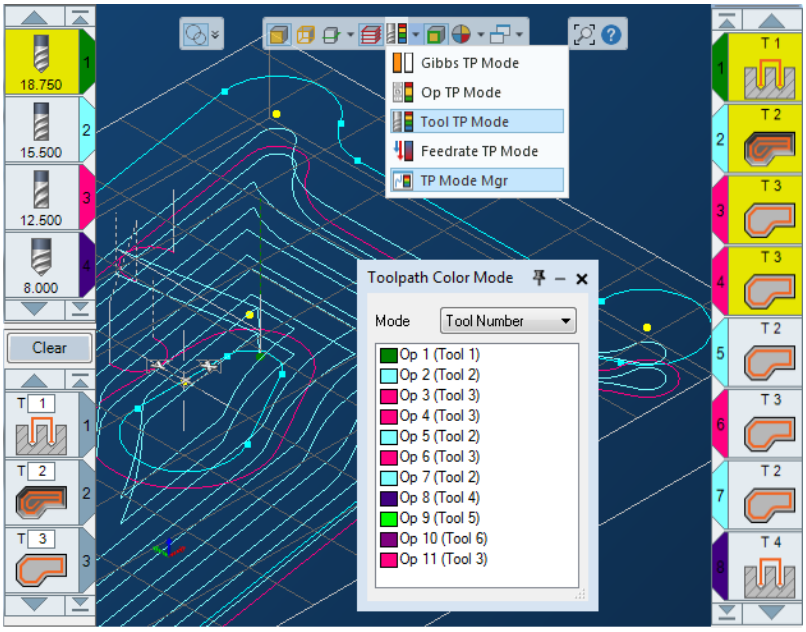
Op Number Toolpath Color Mode



Op TP Mode is useful for multiple operations, so you can see which ops create which toolpath:

- Interop and entry/exit moves are all in one color. (default: white)
- The toolpath for each operation is drawn in that operation's "Op Color" — that is, the same color displayed when Op Sim is in Op Color mode.
- Rapid moves are in the same color as feed moves, but drawn with dashes.

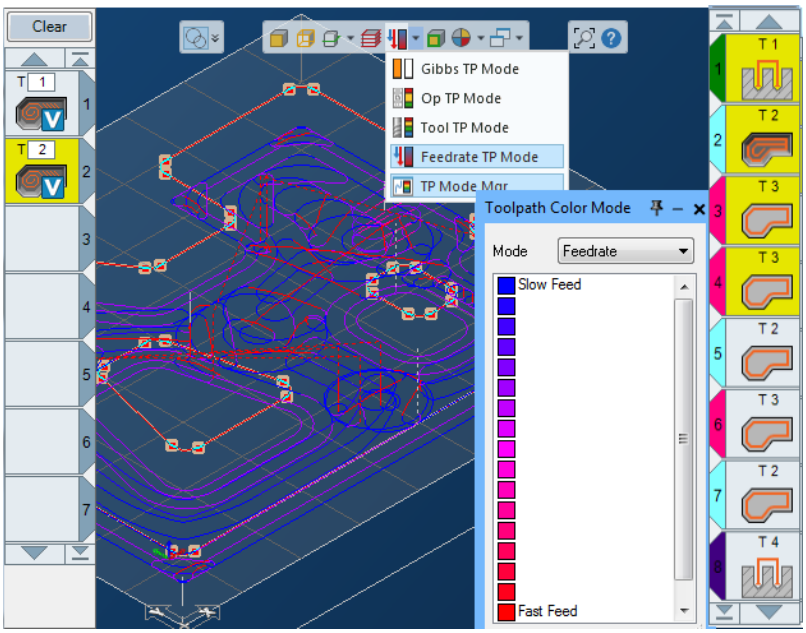
Tool Number Toolpath Color Mode



Tool TP Mode lets you see which tools create which toolpath:

- The toolpath for each op is drawn in that op's "Tool Color" — the color displayed when Op Sim is in Tool Color mode.
- Interop and entry/exit moves are also in the Tool color, so you can easily identify tool changes.
- Rapid moves are in the same color as feed moves, but drawn with dashes.

Feedrate Toolpath Color Mode



Feedrate TP Mode lets you see when moves vary in speed so you can identify fast feed moves that might be dangerous:

- The toolpath for each operation is drawn in a color that ranges from cool "slow feed" to hot "fast feed"
- Entry/exit moves are drawn in the same color as an op toolpath move at that feedrate.
- Interop moves are in Gibbs color (default: white); interop feed is solid, and interop rapid is dashed.
- Intra-operation rapids are drawn in a dashed red.

Pre-Selection Highlighting

This mode highlights objects on the display as if they were selected whenever the cursor passes over them. The preselection face and edge colors, halo transparency, and thickness can all be set by the user in preferences on the Color Tab.

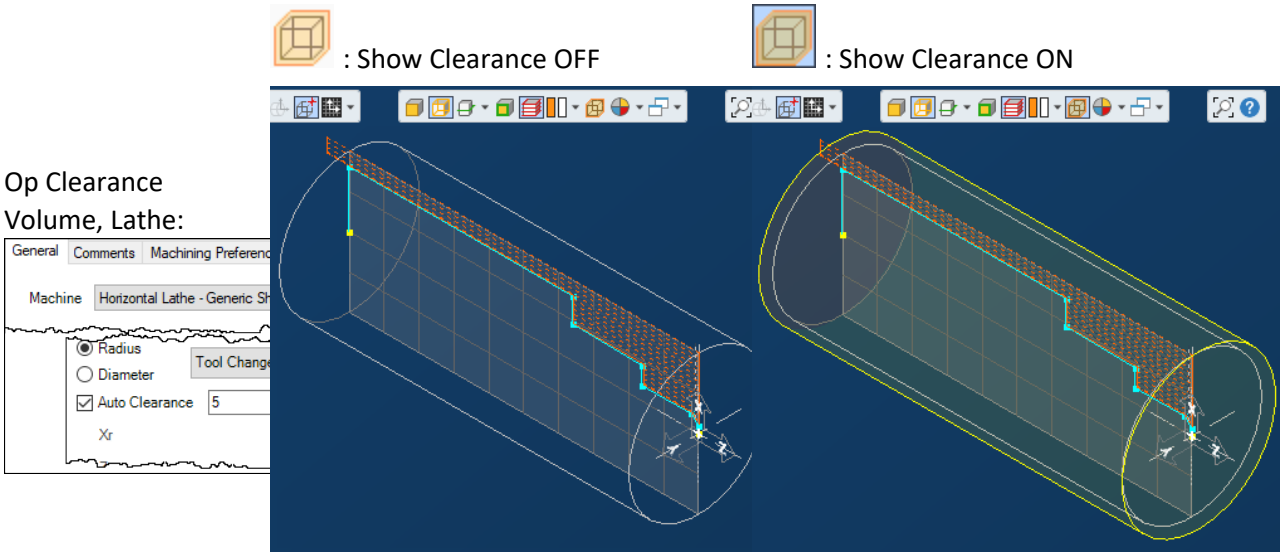
Show Clearance Planes / Clearance Volumes

When this button is not depressed (OFF), the workspace does not display clearance volumes or clearance planes.

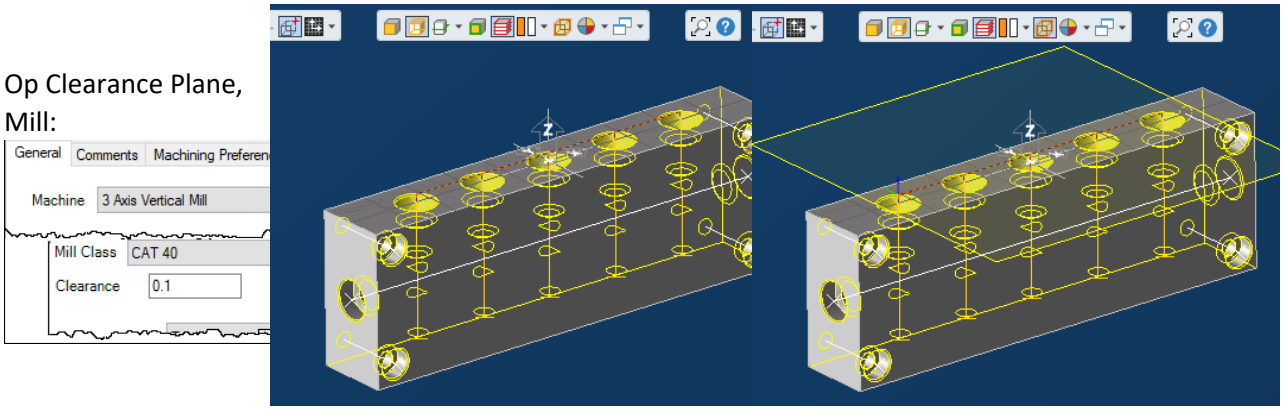
When this button is depressed (ON):

- If the MDD defines clearance volumes, the workspace displays them.
- If the MDD does not defines clearance volumes, the workspace displays the clearance plane for the current operation.

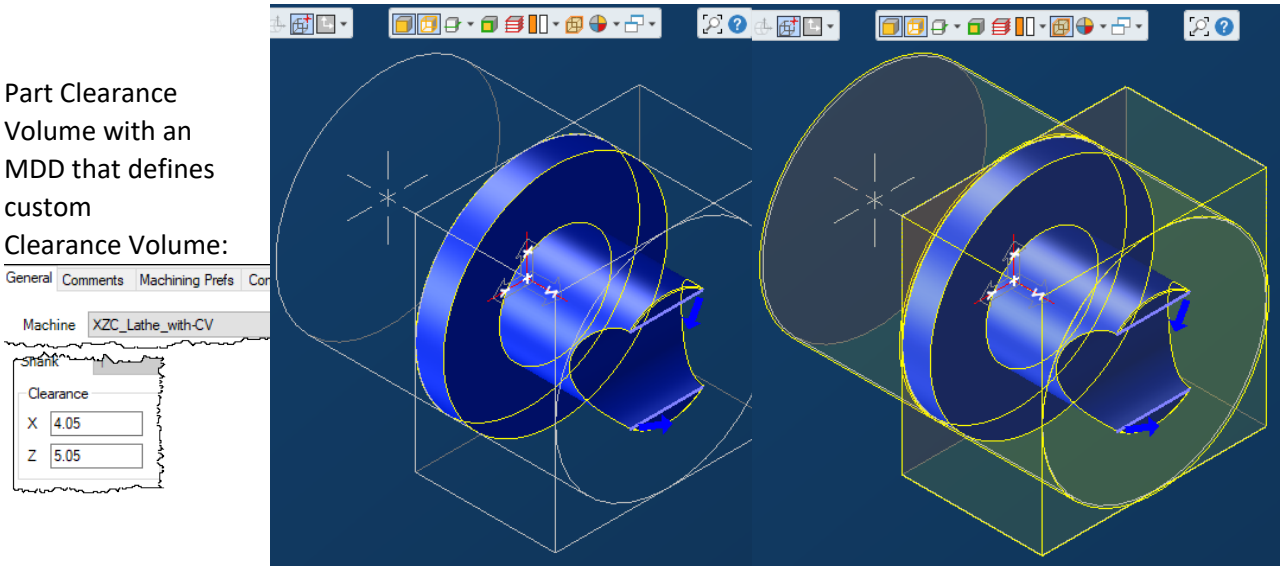
See the following illustration.



Op Clearance Plane, Mill:



Part Clearance Volume with an MDD that defines custom Clearance Volume:



Toggle Color Mode: Gibbs Color / Feature Color / User Color

This mode determines the color mode currently in effect for the workspace. Hover over the dropdown button to display all modes that can be selected.

Gibbs Color

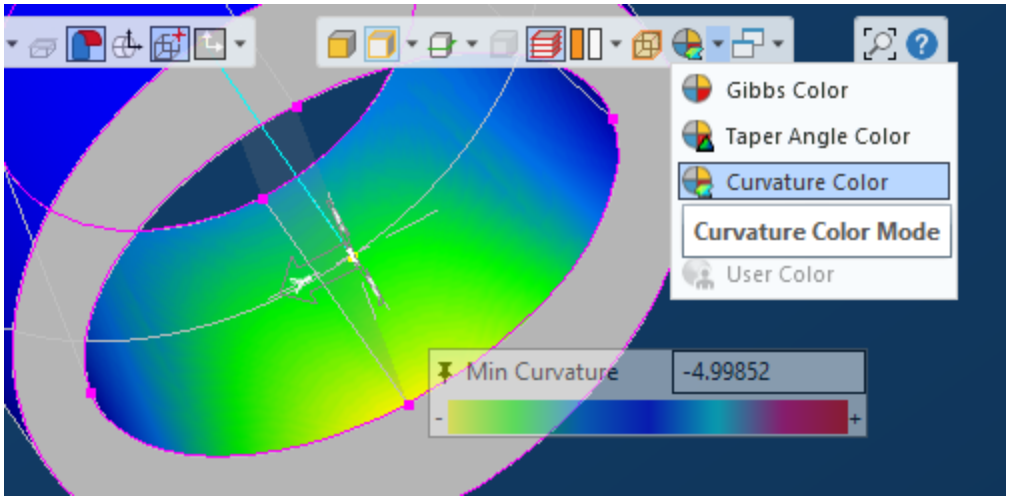
This mode is the default system color mode. For more information, see the extensive discussion of color under “Appearance” on page 17, especially the Colors tab.

Feature Color

This mode exists only if the model contains features defined in Feature Manager to have feature color attributes. For more information, see the *Features* guide.

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.

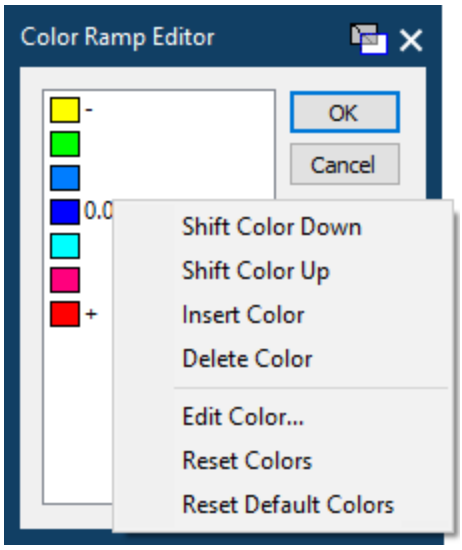
To switch between signed and unsigned: Right-click the title bar of the Curvature dialog and use the context menu to toggle your preference, as follows:

- 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” on page 17.

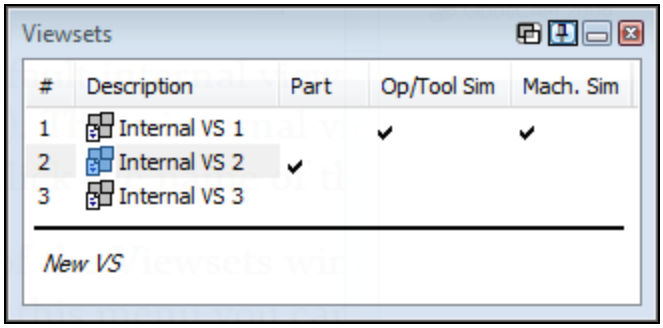


User Color

This mode exists only if the model contains features defined in Attribute Manager to have user color attributes. For more information, see the *Features* guide.

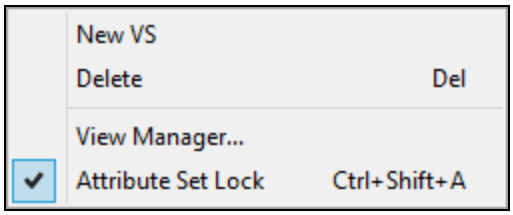
Viewport Manager

This button activates the Viewport Configuration Manager, which lets you to switch between viewsets, create a new viewset, bring up the view manager, indicate a default view for a viewset, or lock the attributes of a viewset.



The system maintains three default internal viewsets, each of which maintains its own set of viewports (panes and windows). These internal viewsets are tied to a particular part. To switch between viewsets, simply hover over the dropdown arrow and click the name of the desired viewset.

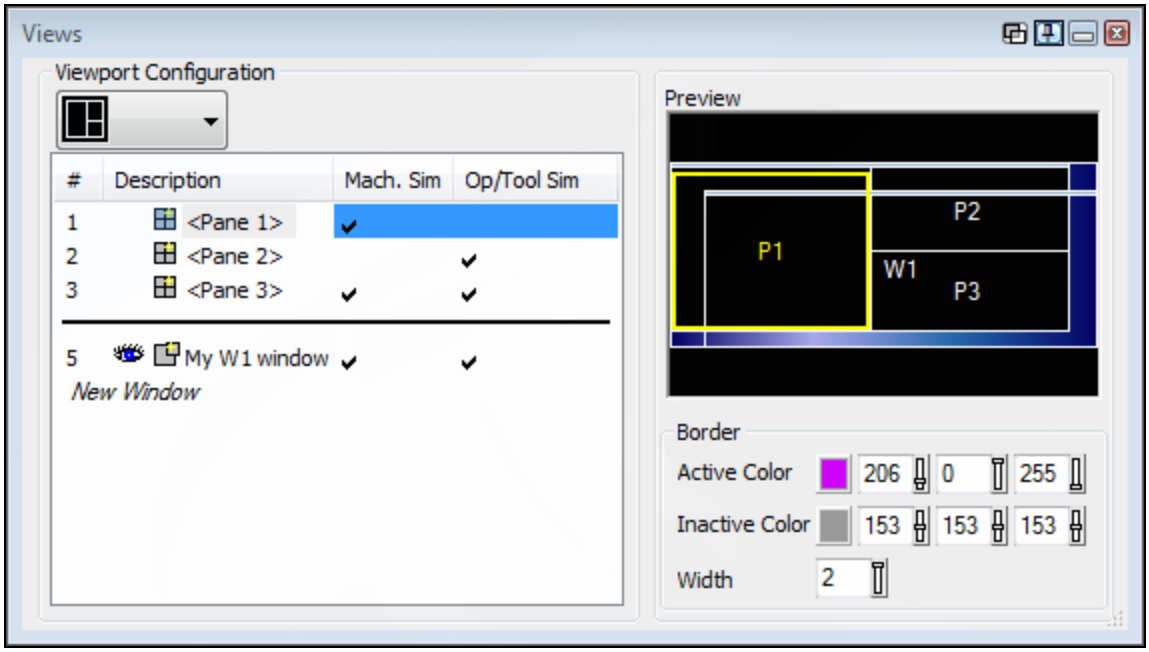
Right-clicking the title bar of the **Viewsets** window, or right-clicking the name of a viewset, displays a context menu. From this menu you can create a new viewset, delete a viewset, access View Manager, or lock the attribute set.



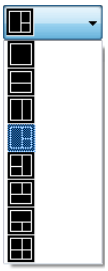
If Attribute Set Lock is checked, any changes made (turning on edge selection, render wireframe, etc.) will affect ALL viewports, active or inactive. If Attribute Set Lock is unchecked, changes will affect only the active viewport.

Additionally, you can define if a given viewset will default to Part, Op/Tool Sim, or Machine Sim. These checkmarks can be active in only one viewset at a time. They specify which viewset will be used for each GibbsCAM mode (Part, Op/Tool Sim, or Machine Sim).

The Viewsets context menu includes an option to access View Manager, which displays the **Views** dialog box.



The pull-down button lets you select one of the eight pre-defined pane layouts. Panes are always visible. Each pane or window will always show in part mode, and display the part.



View Manager also allows you to define the look of each viewport. Each viewport can have Machine Sim and Op/Tool Sim toggled on or off. If the Machine Sim and/or Op/Tool Sim options are

checked, the pane or window will also show a rendering session (Machine Sim or Op/Tool Sim) when using these modes.

Windows behave slightly differently from panes here, as they can be toggled to be seen or unseen by clicking on the eye icon.

Unzoom (Ctrl + U)

Returns screen to full visibility.

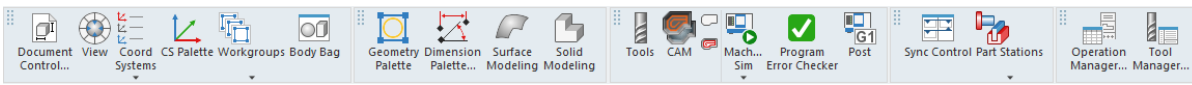
Help on Item

This item is used to activate context-sensitive online help. When you select this item, the cursor switches to the Help cursor. When you click on an item in the interface with the Help cursor, the online help system opens to display the help topic for that the selected item. As soon as you click on an item, the cursor switches back to its normal state.

Commands toolbar

This area of the User Interface is highly customizable: individual commands can be added or removed, and individual command groups can be moved out of the toolbar to the bottom or sides of the workspace — whatever suits your needs. (See Customizing the User Interface)

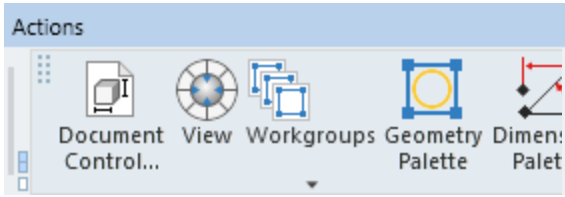
The standard display is as shown below:

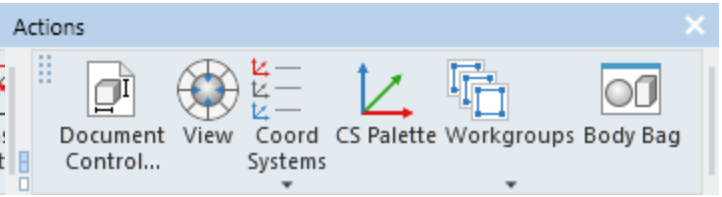


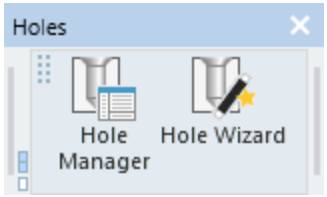
The toolbar can be minimized by using the small box controls situated on the left of the menu. (In the example below the command text has also been removed).

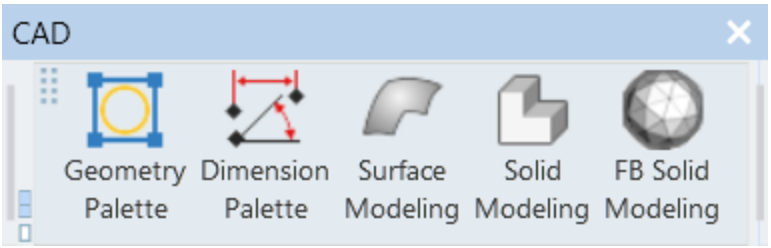
The contents of the default toolbars change according to the current Interface Level set in the File>Preferences>Interface Tab.

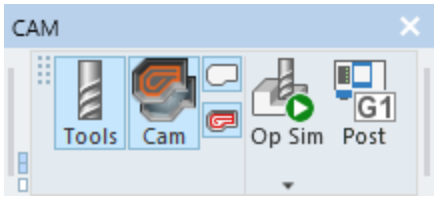
Level 1 Interface	Level 2 Interface

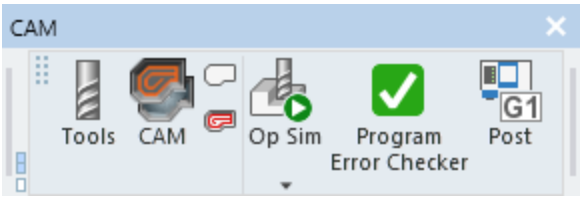


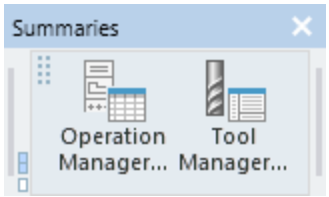
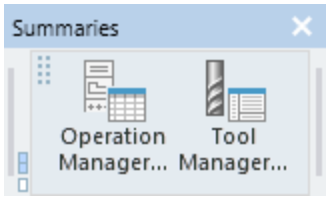
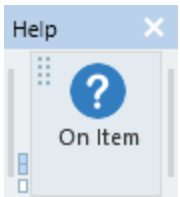
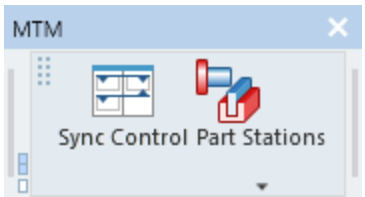




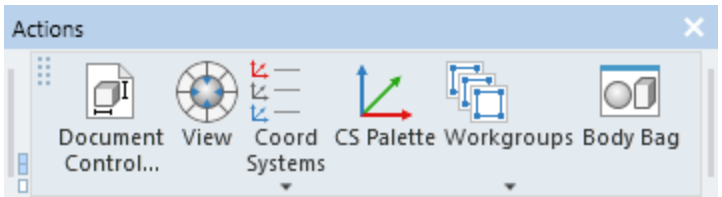






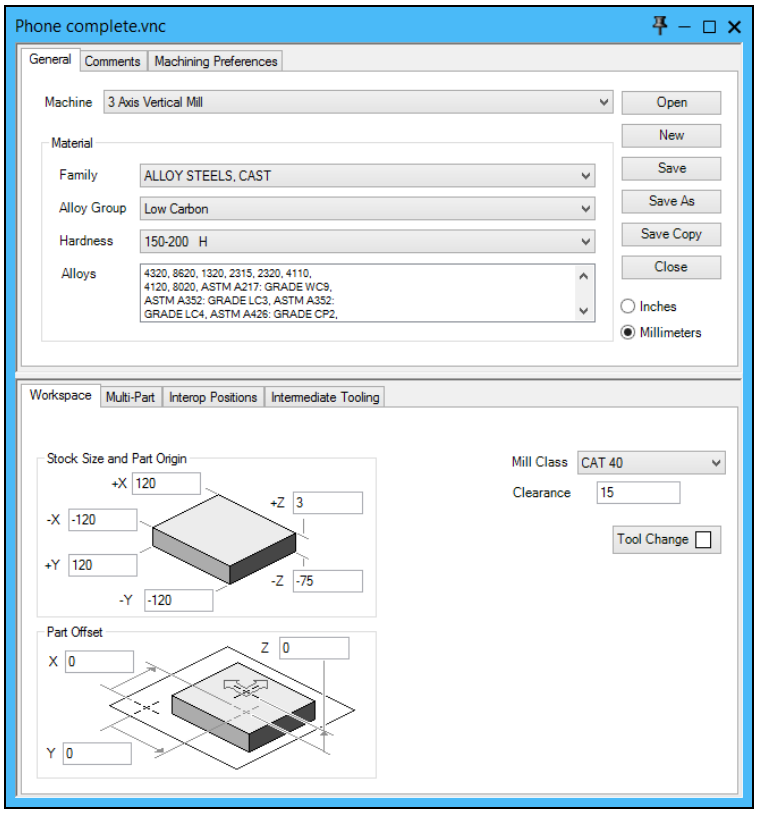
Level 1 Interface	Level 2 Interface
	
	

Actions Command Toolbar Group



Document Control Dialog

Displays the Document control dialog where the major settings for the part are specified. For more information, see the section on "Setting up a part" in the *Getting Started* *Mill* and *Lathe* guides.



View Control Palette

The View Control palette lets you easily change the current view of the part. The eight buttons around the outside of this circular palette provide quick access to standard views and lets you redraw or unzoom. Holding the Alt key and clicking one of the buttons shows the opposite view.
The trackball is moveable; drag its edge to move the trackball around the workspace.

Click and hold the center ball with the mouse and it will operates like a trackball. If you use the left mouse button, the part moves only when you release the button. Use the right mouse button and the part moves dynamically. The “T” always identifies the Top surface of the part.

The track ball has four small solid black triangles around it, inside the ring of buttons (located at 12:00, 3:00, 6:00, and 9:00). These are called axis “handles”. Drag any of these handles in a circular motion for Z axis view change. Or drag the top/bottom triangle vertically (up and down) to rotate the part about the Y axis. The left/right handle dragged horizontally (left and right) rotates the part about the X axis.

If your mouse is equipped with a scrollwheel, you can `click` anywhere in the workspace and `drag` the mouse to dynamically move the stock and geometry. Scrolling with the scrollwheel will also make the part display larger or smaller.

Keyboard control

Rotating

Rotating the part is controlled by pressing the `Shift` key and the appropriate arrow keys on your keyboard. Rotating can also be accomplished using `Ctrl+click-dragging` the third button of the mouse (producing the same results as rotating the part using the rectangle in the track ball).

Shift ↑ This key combination rotates the part upward by about 10%; south moves toward you and north recedes.

Shift ↓ This key combination rotates the part downward by about 10%; north moves toward you and south recedes.

Shift → This key combination rotates the part to the right by about 10%; west moves toward you and east recedes.

Shift ← This key combination rotates the part to the left by about 10%; east moves toward you and east recedes.

> ⓘ The point of rotation depends on the current view. Whatever intersects with a ray that projects from the center of the GibbsCAM window is what the view rotates about. If the ray intersects with a solid, the part rotates about the center of the intersection. If the ray intersects with the part stock, the rotation is about the center of that intersection. If the stock is not in the center of the window, the rotation is about the intersection of two planes that define the stock boundary and the ray.

Panning

Panning is controlled by pressing the `Ctrl` key and the appropriate arrow keys on your keyboard. Panning can also be accomplished by a `Ctrl+Drag`.

Ctrl ↑ This key combination moves the part up by about 10% of the screen.

Ctrl ↓ This key combination moves the part down by about 10% of the screen.

Ctrl → This key combination moves the part to the right by about 10% of the screen.

Ctrl ← This key combination moves the part to the left by about 10% of the screen.

Zooming

Mouse. You can `drag` a marquee around the region to zoom in. Additionally, if your mouse is equipped with a scrollwheel, each `turn` of the scrollwheel is equivalent to a 10% zoom in or out. For scrollwheel zoom actions, the center of the zoom is determined by an interface preference that you can set using File > Preferences > Interface: If the Zoom to Mouse Cursor checkbox is selected, the zoom center is the cursor; otherwise, the zoom center is the window center.

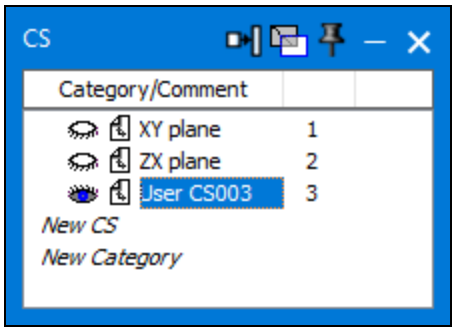
Keyboard. You can zoom in and out of the part is by using the `Ctrl` key with the `+` or `-` keys.

Ctrl + This key combination zooms in on the part by about 10%.

Ctrl - This key combination zooms out from the part by about 10%.

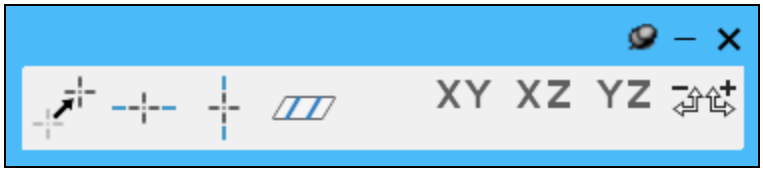
CS List

Clicking this button activates the Coordinate Systems dialog. If you click on the small arrow in the bottom corner of the icon, a selectable list of all the current CS's is displayed. Coordinate systems are used for 3D geometry creation, rotary part orientation for machining, multiple work fixture offsets, and as a basis for solid modeling. For more information, see the "Coordinate Systems" section in the *Getting Started* guide.



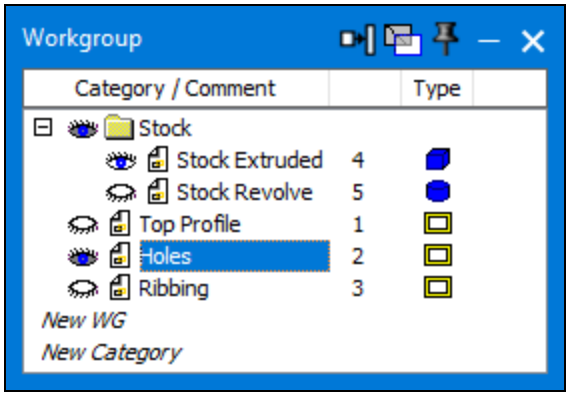
CS Palette

This activates the Coordinate System Palette. For more information on Coordinate Systems see the *Advanced CS* guide.



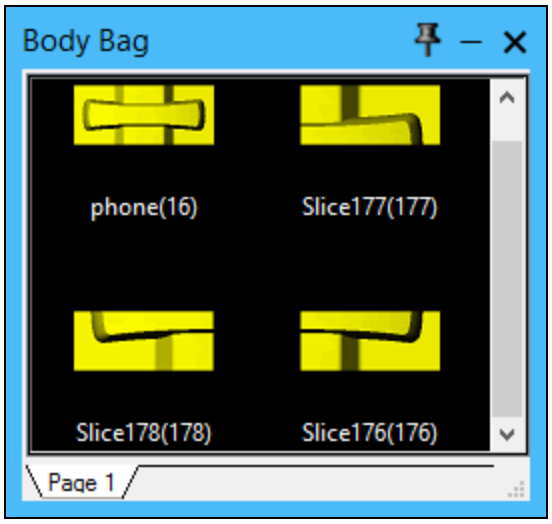
Workgroup

Clicking this button activates the Workgroup dialog. If you click on the small arrow in the bottom corner of the icon, a selectable list of all the current Workgroups is displayed. Workgroups are separate layers used to separate different groups of geometry, including custom stock. For more information, see the "Workgroups" section in the *Getting Started* guide.

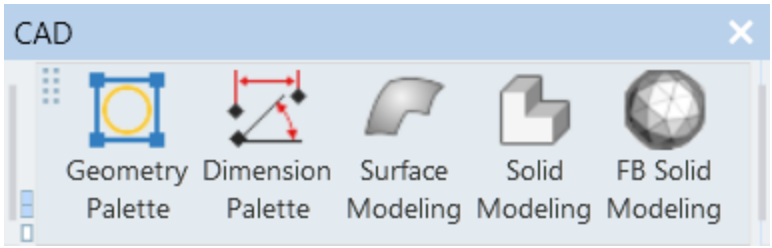


Body Bag

This opens the Body Bag. The Body Bag is used to organize the Workspace by storing bodies during part creation. For more information on the Body Bag, see the "About the Body Bag" section of the *SolidSurfacer* or *2.5D Solids* guides.

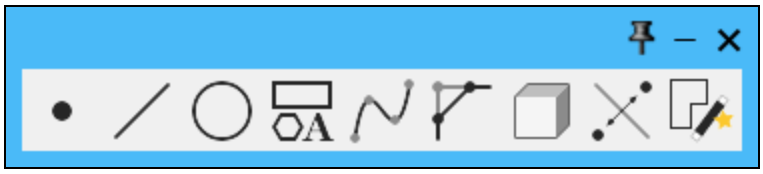


CAD Command Toolbar



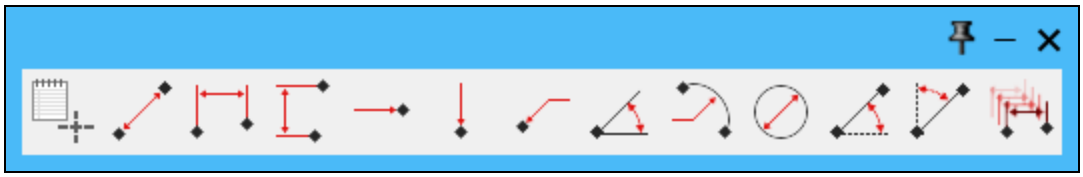
Geometry Palette

This opens the main Geometry creation palette. For more information, see the *Geometry Creation* guide.



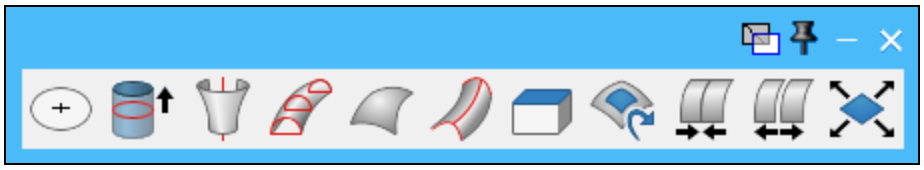
Dimension Palette

This opens the Dimensioning Palette. For more information, see the "Dimensioning" section in the *Geometry Creation* guide.



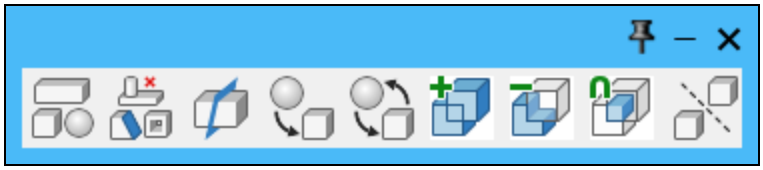
Surface Modeling

This opens the Surface Modeling Palette. You use the Surface Modeling palette to perform sheet, or surface, modeling. For more information on surface modeling, see the "Surface Modeling" section of the *SolidSurfacer* or *2.5D Solids* guides.



Solid Modeling

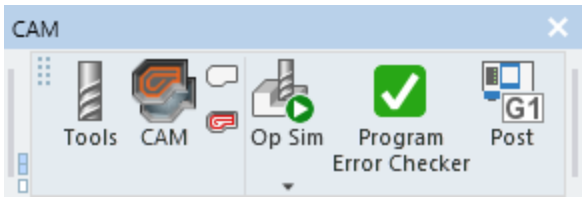
This opens the main Solid Modeling palette. For more information on solid modeling, see the "Solid Modeling" section of the *SolidSurfacer* or *2.5D Solids* guides.



Solid Modeling

This opens the Facet Body Solid Modeling palette. For more information on facet body modeling, see the "Facet Body Solid Modeling" section of the *SolidSurfacer* guide.

CAM Command Toolbar

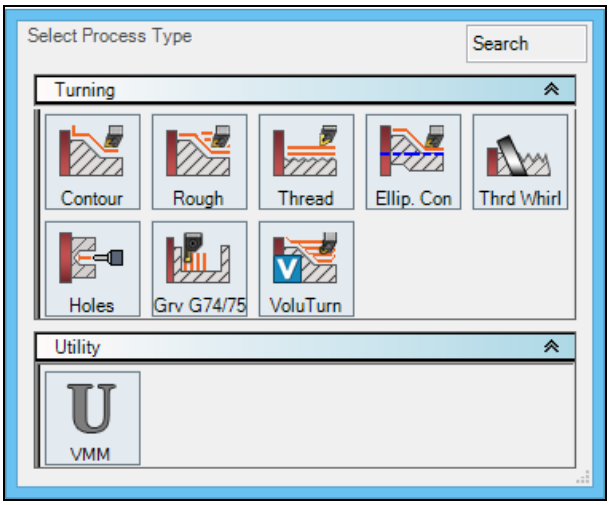


Tools

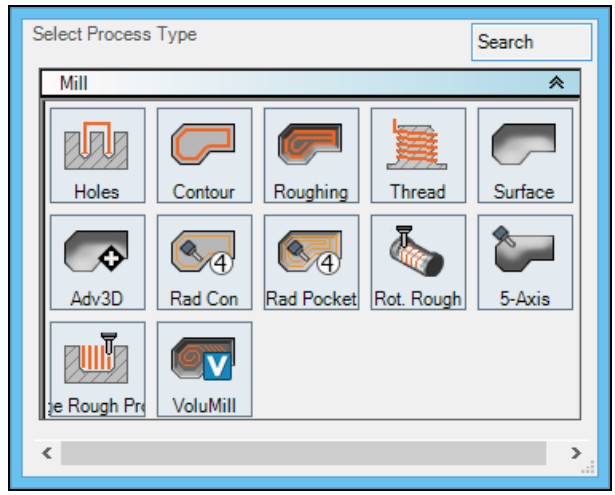
This button activates the Tool List on the top left of the screen, containing the tiles that you use to define tools. For more information see "Tool Creation" in the *Getting Started* guide.

CAM

The Machining button activates the Process and Operations lists and the Machining palette for the type of machine specified in the Document control dialog. You can move the Machining palette to any location on the screen. This palette contains Function tiles and buttons. The Function tiles are moveable objects that you can drag to the Process List to create operations.



Lathe Machining Palette



Mill Machining Palette

Note: The processes that appear on the palette depend on which product options are licensed and active. They also vary according to the Machine Definition Document (MDD) associated with the Machine type currently specified in the Document Control dialog.

The Select Process Type dialog can be customized. See the Process dialogs section in the *Mill* and *Lathe* Guides.

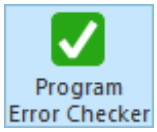
Tile Lists

When you click the CAM button, both the Process and Operation Tile lists are displayed. The Processes and Operations buttons will toggle the individual lists on and off.

Cut Part Rendering and Simulation

For more information see the section on Simulation Cut Part Rendering and Simulation:

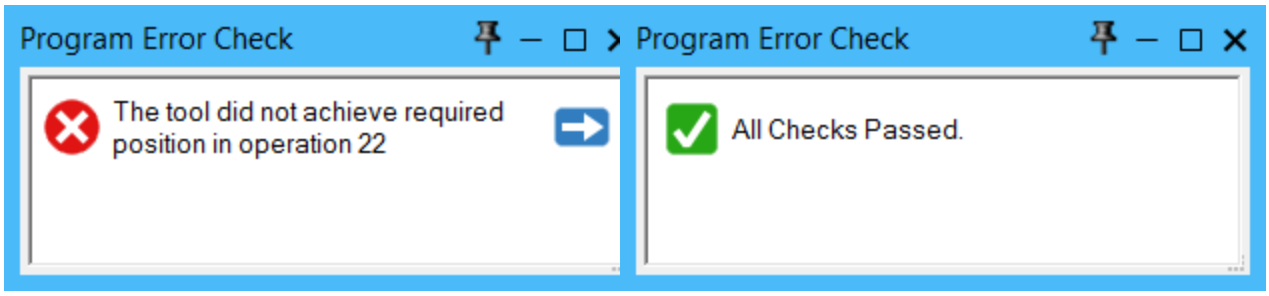
Program Error Checker

When you open a pre-V12 *.vnc file in V12 or later for the first time, an 

icon appears on the Main Palette:

To run Program Error Check, click the button. The system will then test the current *.vnc file for errors and incompatibilities and displays any problems found, or else the message “All Checks Passed.”

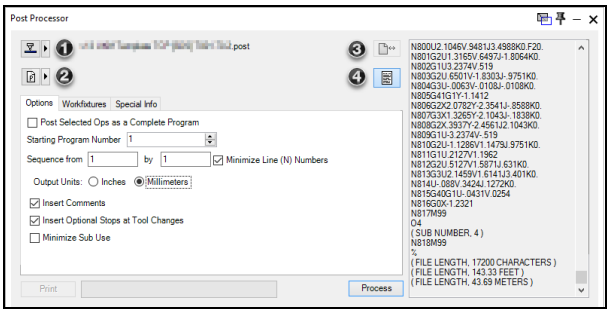
> **Please Note:** This applies to active operations only; inactive operations are unaffected. For more information about active and inactive operations, see *Common Reference*, chapter "Miscellaneous", section "Lists", subsection "Active and Inactive Operations".



Post Processor dialog

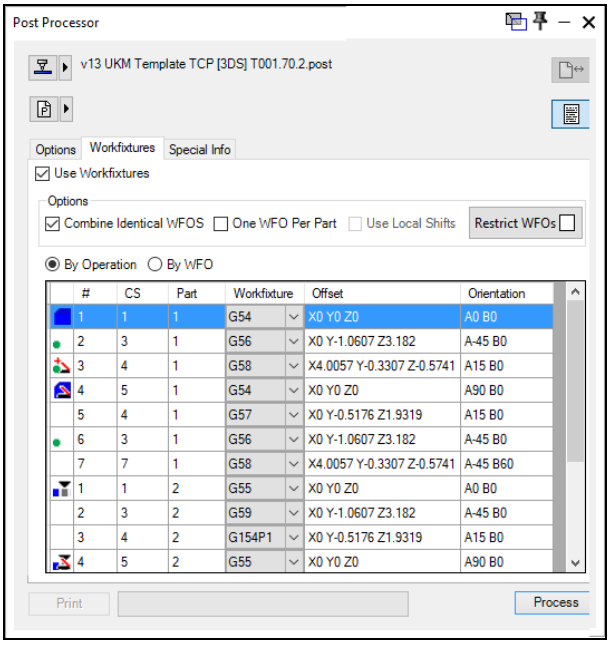
The Post Processor button in the Command Toolbar becomes active once machining operations have been created in a file. Clicking Post Processor will display the **Post Processor** dialog, which allows you to select a Post Processor, an .NCF Program name, and the format of the output. The Document Control dialog is used to set up Multiple Parts or TMS, if available.

This applies to active operations only; inactive operations are unaffected. For more information about active and inactive operations, see *Common Reference*, chapter "Miscellaneous", section "Lists", subsection "Active and Inactive Operations".



For details, see Options tab

1. Post Processor Selection
2. Program Name
3. Communications
4. Text Window



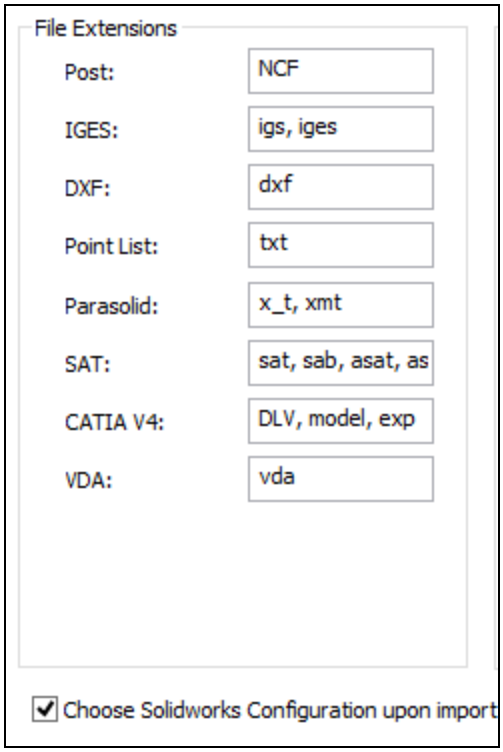
For details, see Workfixtures tab

The text file will be saved under the file name entered and is displayed to the right of the Program Name button. If a file with that name already exists the system will ask if you want to replace the existing file. If yes, clicking the Process button will erase the old file and replace it with the new one. To view the text file as it processes, click the Text Window button. When this button is depressed, a window will appear that displays the posted text file as it is created. The program will scroll by in the window as it is being generated. The Pause button lets you stop the scrolling of the output as it is being processed. The Print button becomes available after the program file is finished processing. Clicking the Process button generates the NC program for the file currently open.

Before posted output can be generated, the post processor and program file name must be specified.

To select the post processor, click the Post Selection button. An **Open** dialog allows you to access the directory or folder where the post processors are stored in the system.

The file extension of a post processor is `*.post` (or, in much earlier releases, `*.pst`). The full filename displayed in the **Open** dialog usually reflects the control and the machine.



When a file is post processed, a text file is created with the extension specified in the Post text box in the File > Preferences > Import/Export > File Extensions dialog.

By default this extension is set to *.NCF.

A file can be post processed multiple times and saved with different *.NCF file names. If changes are made to the part file, it must be post processed again in order to incorporate the changes into the posted output. As a default, the text file uses the part file name with an *.NCF extension (e.g. EXAMPLE1.NCF). This file name can be changed by clicking the Program Name button and entering a new name.

Options tab

The items in the first tab of the **Post Processor** dialog change the format and content of the finished output. Each item can react differently with different post processors. Below is a description of the effect they will have with most post processors.

Post Selected Ops as a Complete Program

Checking this item will cause the output to only include operations that are currently selected in the Operations list. Items that are unselected will not be output.

This is not recommended for MTM part files or any part containing rotary axis rotations. Instead, post-process all operations together, to avoid possible large differences between posted output of operations using Post Selected Ops as a Complete Program and the same operations when posted together with others.

This applies to active operations only; inactive operations are unaffected. For more information about active and inactive operations, see *Common Reference*, chapter "Miscellaneous", section "Lists", subsection "Active and Inactive Operations".

Inactive ops will not be posted to the program. Active and inactive ops functionality adds a more reliable solution to disclude ops, improving consistency and posted code accuracy.

Starting Program Number

This is the number of the program as it will appear in the control. It is also the starting number for all sub-programs. If the Starting Program Number is one, the first sub-program will be two, the second three, etc.

Sequence from

This is the starting number for the block numbers or “N” numbers. The by box determines the increment.

Minimize Line (N) Numbers

If this option is turned on, the post processor will only output block numbers on tool change positions.

Insert Comments

This option will output information about each operation and tool used as well as file length. If any additional comments have been entered by the user about tools or operations, they will be output as well.

Insert Optional Stops at Tool Changes

If this option is on, the software will output a machine operator selectable program stop at every tool change. This is generally used as an aid to the setup man.

Minimize Sub Use

When this option is checked the post processor will prefer longhand output when possible.

Workfixtures tab

Note that the post processor that you have chosen influences what the Workfixtures tab presents. The post is compiled with many settings that apply to your particular CNC. Therefore those settings dictate how the WFO algorithm proceeds as it calculates the operation/WFO list to display in the Workfixtures tab.

If your post is from before GibbsCAM 12, the interface simply lists CS’s and WFOs, and the rest of this discussion does not apply to you.

For WFOs that are not in one-to-one correspondence with CS's, items in the Workfixtures tab of the **Post Processor** dialog let you do the following.

This applies to active operations only; inactive operations are unaffected. For more information about active and inactive operations, see *Common Reference*, chapter "Miscellaneous", section "Lists", subsection "Active and Inactive Operations".

- Distinguish at a glance a part station from a multi-part part instance, with markers for whether the WFO is manually changed, the op is shifted from the WFO origin, op oriented differently from the WFO definition, or any combination, as shown in this table.

	No shift, same orientation	Op shifted from WFO	Op oriented differently from WFO	Shifted and oriented
Part Station				

Multi-Part				
(marker: manual change)				
Part Station, manually changed				
Multi-Part, manually changed				

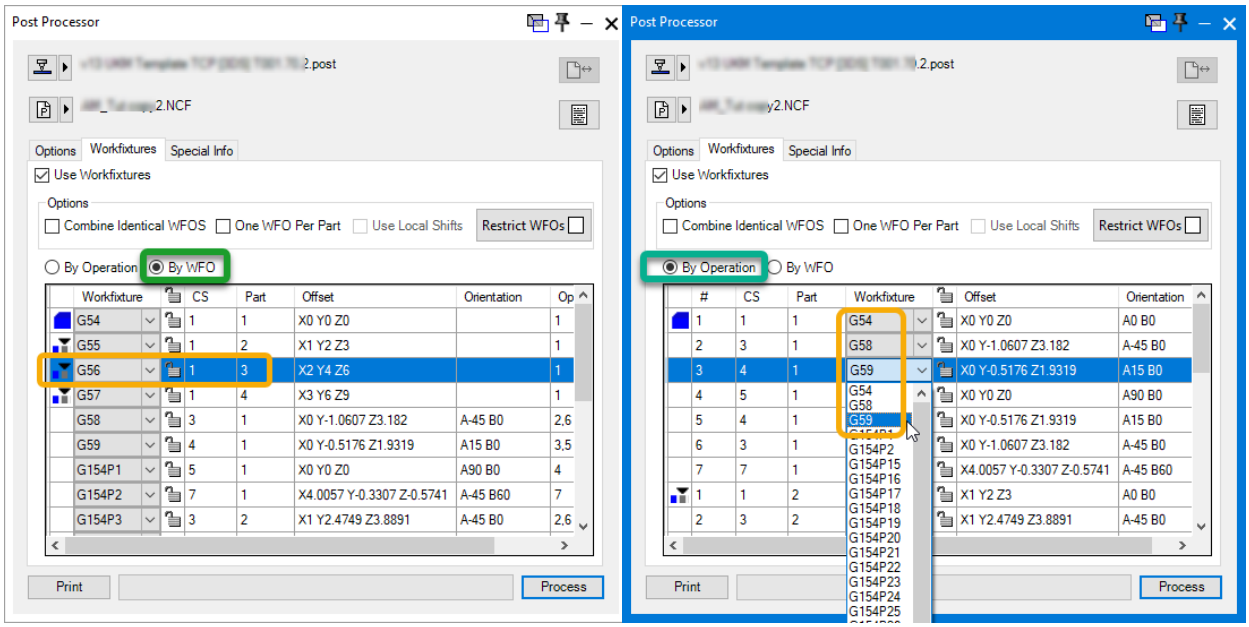
- Assign a WFO per op rather than per CS (including multi-part instances).
- See the WFO list before you post.
- Tell the system how to handle cases where a WFO does not match a CS.

Although all columns in the table can be sorted by clicking the column header, the column presentation and organization depend on which option button is chosen: By Operation or By WFO.

The **Workfixture** column displays only the WFOs that are currently relevant. Relevance can depend on the current MDD, DCD, post processor, and choices and settings in the dialog.

For example, when Part Duplication is in effect, if Part 1 is assigned to G55 and Part 3 is assigned to G56 in the By WFO view, then in the By Operation view, when you expand the drop-down menu for an operation that uses Part 1, you will not see G56 as an option (because it is held by Part 3), as shown in the following illustrations:

The Workfixture drop-down menu



In the **By WFO** view, assigning a Workfixture drop-down menu item to a particular part ...

... makes that drop-down menu item unavailable to other parts in the **By Operation** view

By Operation

When this is chosen, the focus is on operations. Workfixtures can be applied or reverted.

The columns are presented in this order: (icon), Op #, CS, Part # (or Part Station # for an MDD that defines multiple part stations), Workfixture, locked status, Offset, Orientation.

Locks

A WFO that is locked in the By Operation view becomes (in effect) a user-modified WFO (green dot) and will retain that WFO selection. Unlocking a row has the same effect as performing a Revert on that row (see below, under Right Mouse Menu). In other words: Not only does the icon change from locked to unlocked, but the row is reverted: The WFO algorithm is invoked to find the first available default WFO.

If you click the lock icon in the column header, all rows will lock or unlock so as to match the header.

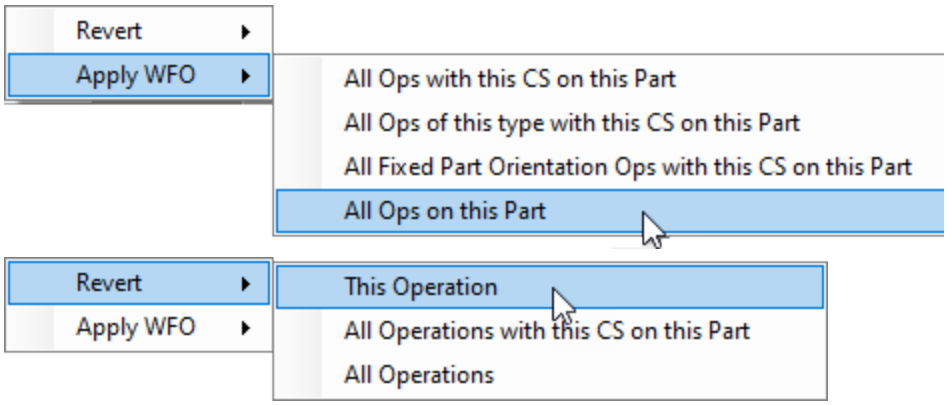
Right Mouse (Context) Menu

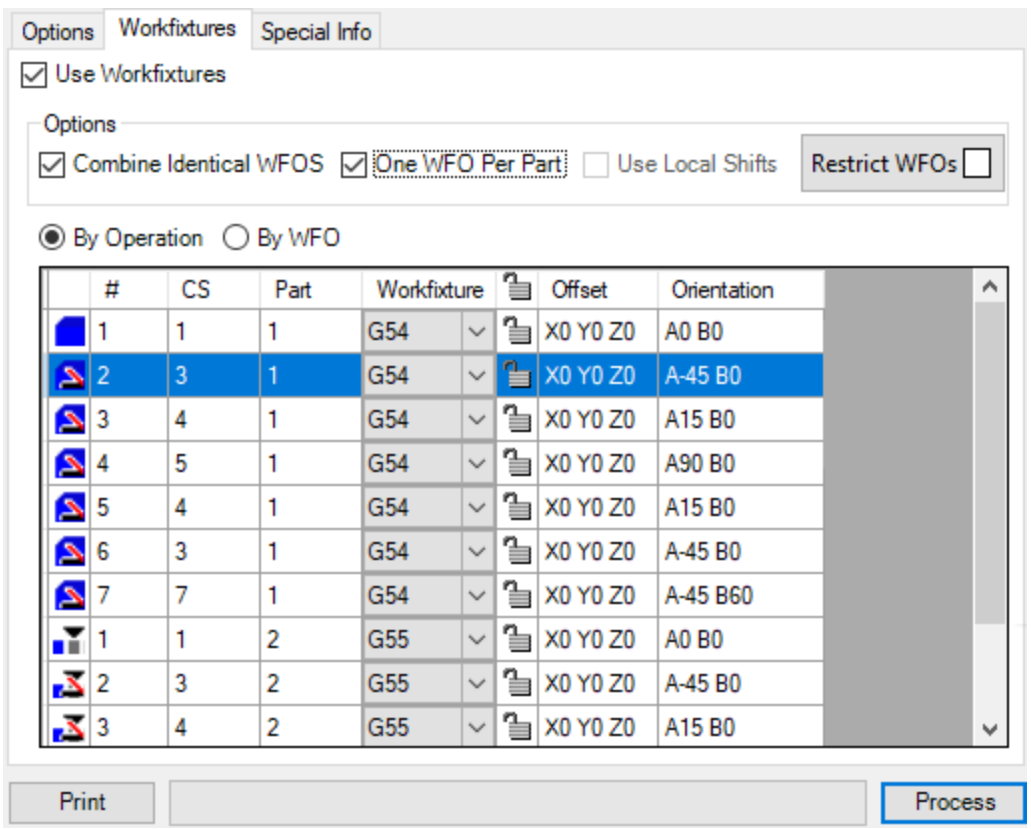
When you right-click a row, the context menu presents two possibilities:

- **Revert** (remove the green dot from the icon): Undo manual changes and re-run the WFO algorithm to re-assign each row using the appropriate default values.
    - This Operation
    - All Operations with this CS on this Part

- All Operations

- **Apply WFO** (add a green dot to the icon): Take the Workfixture assignment of the selected row and propagate it to one or more of the other rows (according to the based menu item you choose). Each menu item represents a different filter for propagating the selected Workfixture. This allows you to modify multiple WFO rows instead of changing each row individually, which is particularly useful when Part Duplication is in effect.
  - All Ops with this CS on this Part
  - All Ops of this type with this CS on this Part
  - All Fixed Part Orientation Ops with this CS on this Part
  - All Ops on this Part





By WFO

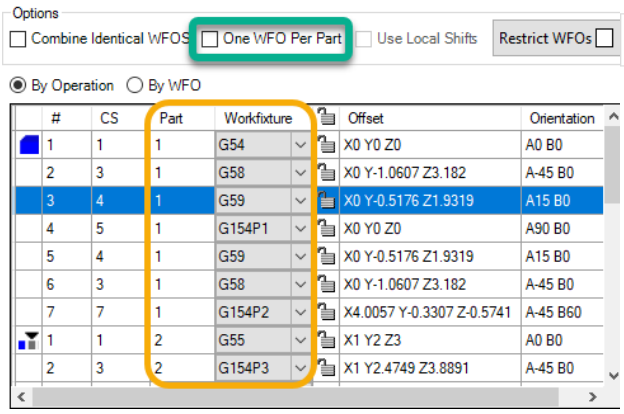
When this is chosen, the focus is on workfixtures; operations are combined, and the icons do not reflect if a WFO has been applied to an operation or if an operation is shifted or oriented differently to a WFO.

The columns are presented in this order: (icon), Workfixture, locked status, CS, Part # (or Part Station # for an MDD that defines multiple part stations), Offset, Orientation, Operation #s.

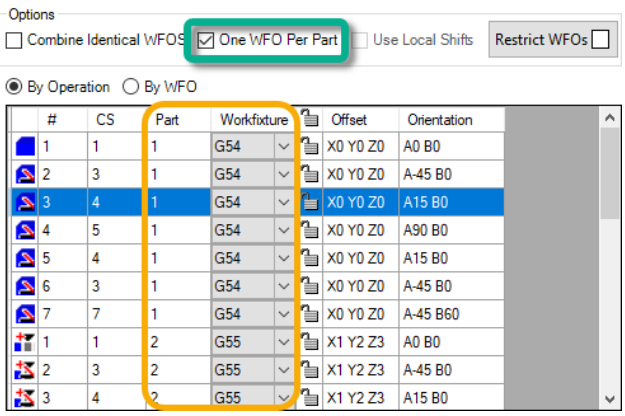
One WFO Per Part

When handling multiple parts, checking the One WFO Per Part box collapses Part/Workfixture assignments so that each part has only one WFO assignment, as shown in the following illustrations.

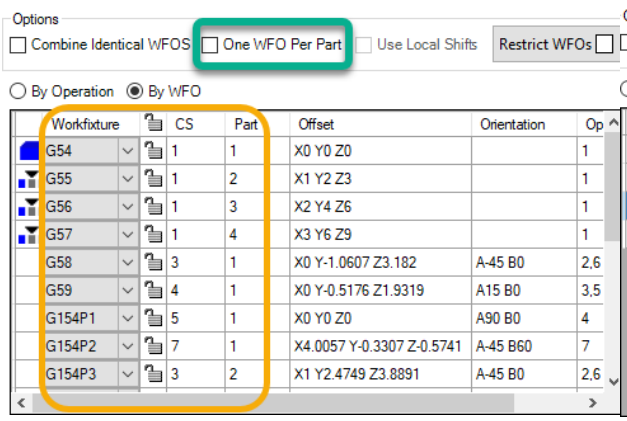
Checkbox: One WFO Per Part



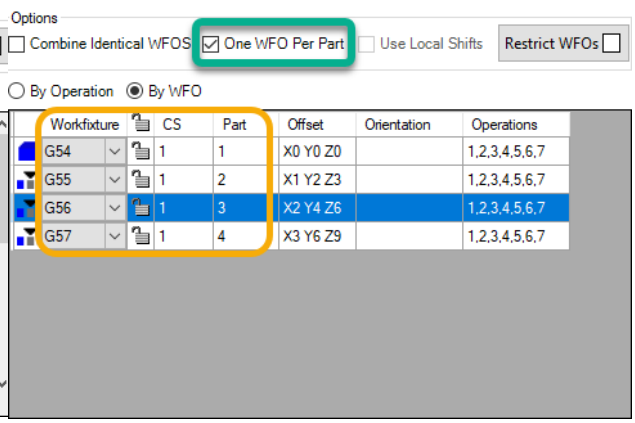
The **By Operation** view,
with as many WFOs as needed per part



The **By Operation** view,
with **One WFO Per Part**



The **By WFO** view,



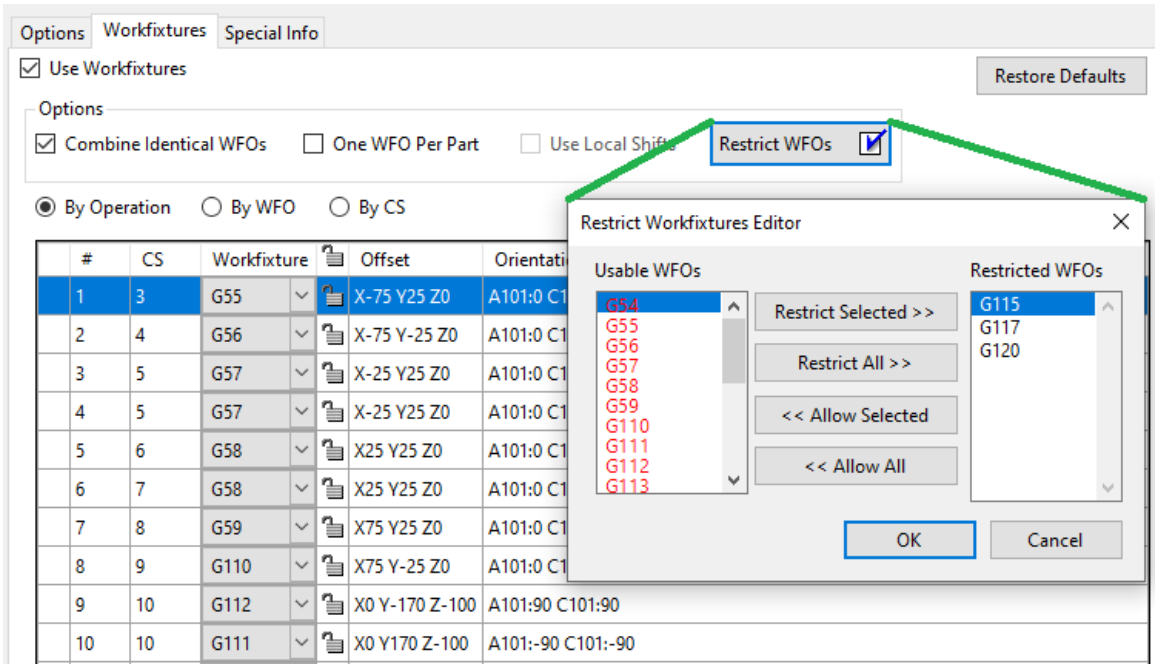
The **By WFO** view,

Checkbox: One WFO Per Part	
with as many WFOs as needed per part	with **One WFO Per Part**

As you can see, the most condensed view is **By WFO** with **One WFO Per Part**.

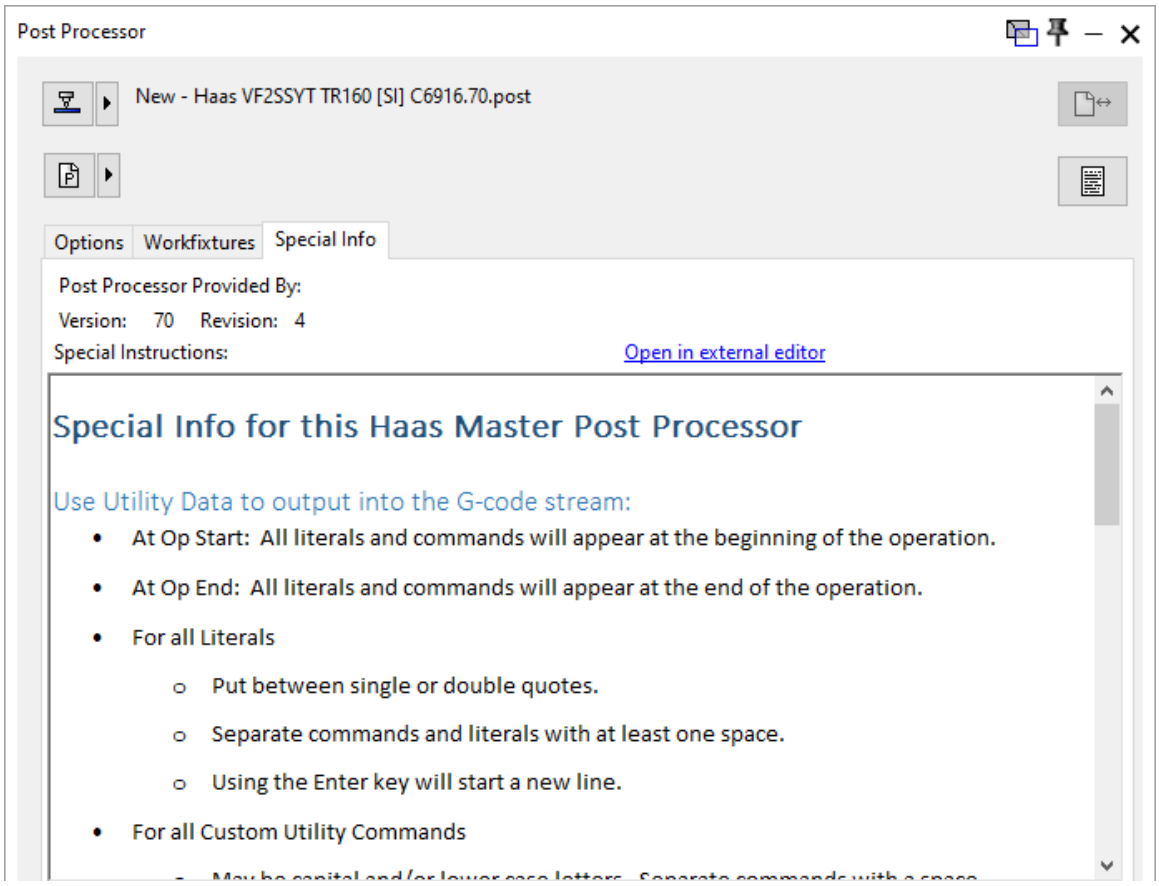
Restrict WFOs

The Restrict WFOs button opens the **Restrict Workfixtures Editor**, illustrated below. If the button has a checkbox (as shown), then at least one of the possible workfixture offsets (WFOs) has been made unavailable.



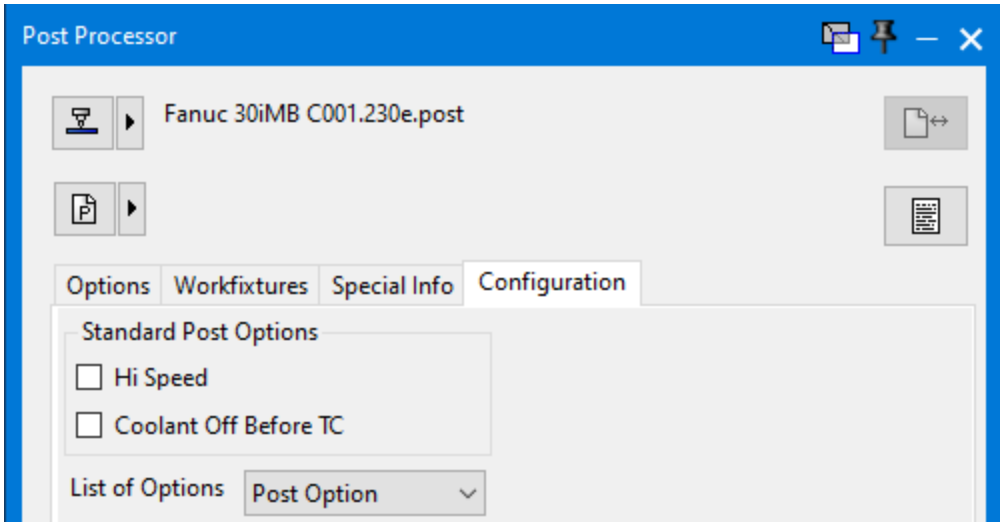
Special Info tab

If the **Post Processor** dialog has a Special Info tab, it contains information written by Gibbs Post developers that is specific to that post. Before requesting help on your post, or a post update, read this information. The following illustrates one possibility.



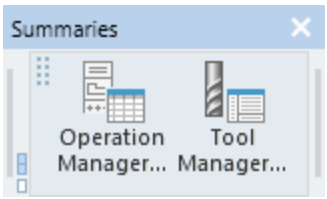
Configuration tab

If there is a Configuration tab, it contains a user interface created by Gibbs Post developers that allow end users to make selections or supply values can be used by the post. Configuration parameters differ from one post to another, and most posts do not have them. The following illustrates one possibility.

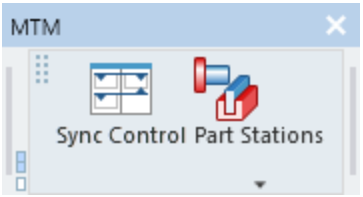


If you have a post that you think could be made improved using user input, talk to your post developer to see if there are existing options that can be exposed to make interacting with the Post Processor easier.

Summaries Command Toolbar

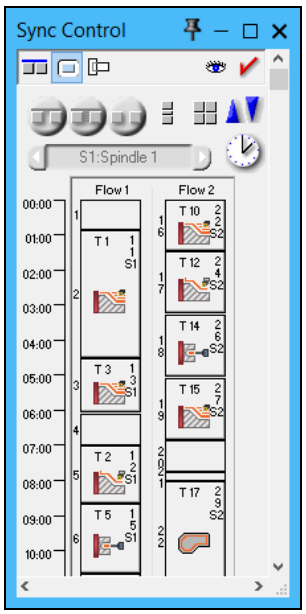


MTM Command Toolbar



Sync Control

This button activates the sync dialog if your part uses multiple spindle operation. It displays operation tiles sized according to their individual run-times in the order in which they will be executed. All other times the option will be greyed out. For more information on Sync Dialogs see the *MTM* guide.



Part Stations

This button activates the Part Station list if your part contains multiple spindles. It is unavailable on parts with only one part station. For more information on part stations (spindles), see the *MTM* guide.

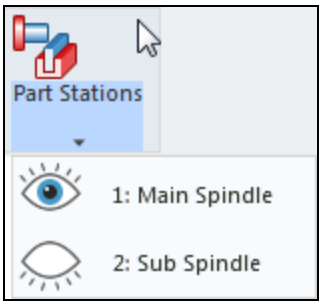
Cut Part Rendering and Simulation

For Simulation, the rendered image is an OpenGL-based 3D part capable of zooming, panning and rotating without restarting the render. The last-used render type becomes the active button in the Floating Toolbar.

This applies to active operations only; inactive operations are unaffected. For more information about active and inactive operations, see *Common Reference*, chapter "Miscellaneous", section "Lists", subsection "Active and Inactive Operations".

Op Simulation display is part-centric toolpath rendering, showing the part with material cut away by the operations.

Tool Simulation shows toolpath movement on the stock without any material removal, displaying inter-op moves as dashed lines. The use of transparent stock display is recommended. In Tool Sim mode, the Render Control has two additional sliders. As with other render modes, the first slider (from top to bottom) controls rendering speed. The second slider moves the tool back and forth through the render process with a fairly coarse granularity. The third slider provides a much finer granularity of control. The Show Toolpath options control the display of the toolpath.



Machine Simulation can include an actual machine model. To use Machine Mode a machine file will need to be selected using the Load Machine option in the Render Control menu. However, if you do not have Machine Sim as a part of your license, you cannot load a machine sim model. If this is the case, machine sim will create a temporary machine kinematic model (no visible components) based on the MDD, and will do the simulation with that. In Machine mode to aid in the visualization of spindles on turning machines, the spindle will rotate while cutting during any lathe turning operation. This rotation is only to assist in showing the direction of rotation.

Rapid Sim rendering is only available for Milling operations and is a very quick rendering method.

Legacy CPR is often faster to render, but if you change the view, the rendering restarts.

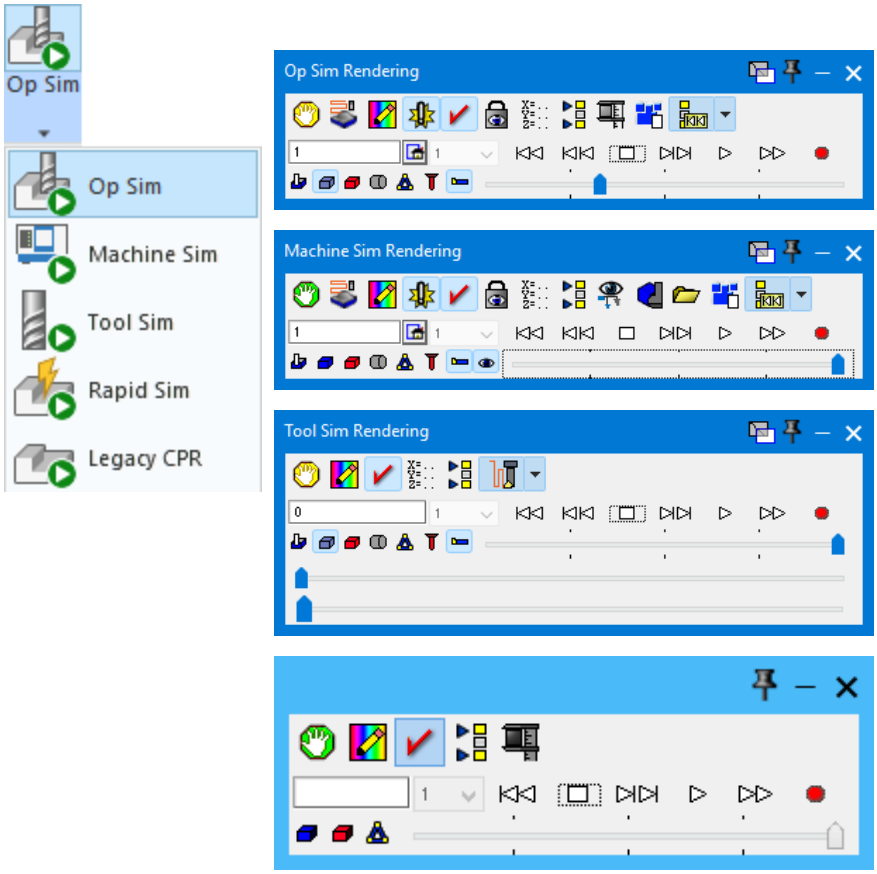
Simulation, in any mode, will show all of the part instances in a TMS multi-part setup, whereas Legacy CPR only shows the single part as programmed in the VNC file. You may find that when checking your part file you use several or all of the rendering types, depending on your needs.

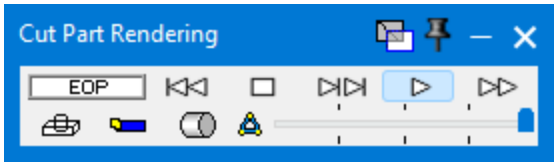
All operations are rendered in their current order. Selected operations render in shades of yellow, and all others render in shades of grey. Shades of red appear when non-cutting tool surfaces or rapid moves hit the material. Selected tools appear in shades of yellow. Deselected tools appear in shades of grey. Smaller pictures render faster and use less computer memory.

- Cut Part Rendering and Simulation Options
- “ Playback Controls ” on page 169
- “CPR/Rendering Visibility Controls” on page 171
- “Simulation Control Icons” on page 174
- “Simulation Context Menus” on page 185
- “Settings for Op Sim, Tool Sim, and Machine Sim ” on page 31

Cut Part Rendering and Simulation Options

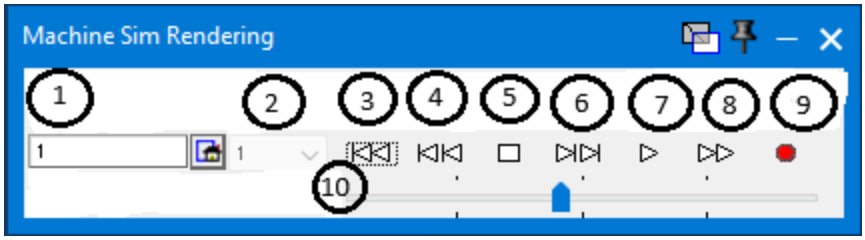
Hovering over the icon on the Command Toolbar, provides a dropdown menu for selecting one of the five methods of Rendering/Simulation. Click to select the required option. The last-selected rendering type remains in effect.





Playback Controls

The playback controls shown below are common to all Simulation palettes.



1. Op/Time Display
2. Active Flow Number
3. Rewind
4. Previous Step
5. Stop
6. Next Feature
7. Run
8. Fast Forward
9. Record (not Legacy CPR)
10. Speed control

Note: If you click the Stop or Next Feature button to pause the rendering, and then close and redisplay the Render Control palette, rendering pauses at the same location. The system remembers where you pause the rendering. Click the Run button to continue rendering.

Op/Time Display

This box displays the number of the current operation being rendered or the current runtime. (Available for all rendering modes)
This displays the current run time or Operation number. If the rendering is stopped, the display is the time it was stopped or the number of the previous operation. A value of zero means that rendering has not yet begun. Available on the Machine and Operation Rendering, the central "home" button, when clicked, aligns the active Flow cutting plane to the home view.

Active Flow Number

The second dropdown box, on the right hand side, available for all Rendering modes except Legacy, is to select the active flow display.

Rewind

Click this button to return playback to the first operation.

Previous Step

Click this button to revisit the saved simulation state which is saved by Previous Step Mode

Stop

Click this button to pause rendering.

Next Feature

Click this button to render the next feature of the current operation, and then pause.

Run

Click the Run button to render the part from the current feature of the current operation in the Current Display box. Rendering continues until you click another button or the last feature of the last operation renders. The Stop and Next Feature buttons pause the rendering. If you click the Play button during rendering, the rendering pauses. The Fast Forward and Rewind buttons change the current operation being rendered, but do not stop the rendering process. When the last feature of the last operation is complete, rendering stops. If rendering is paused, you can click the Run button to resume.

Fast Forward

If rendering is in progress, clicking Fast Forward finishes rendering the current feature, skips the remaining features for that operation, and continues rendering the next operation. If rendering is paused, clicking Fast Forward advances the operation number in the Current Display box to the next operation. Click the Run button to continue rendering.

Speed Control

Shows the current location of the Speed Control slider. The Speed Control slider sets the speed of rendering. You drag the speed control slider to the left to slow the rendering speed or to the right to increase the rendering speed. You can drag the slider while rendering is in progress and the rendering speed adjusts accordingly.

Record Video

For all rendering modes except Legacy CPR, an additional red button is provided. Clicking it opens a dialog that lets you save the current simulation as a video file. The output video can then be played and viewed independently of GibbsCAM.

Output File

You can designate the path and filename of the output file.

Record

Click this button to start recording the Machine Simulation video.

Encoder

From the pull-down list, you can choose one of the following video encoders:

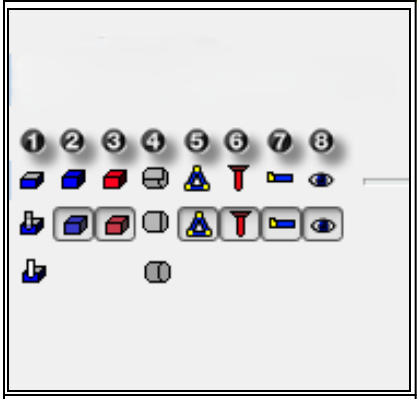
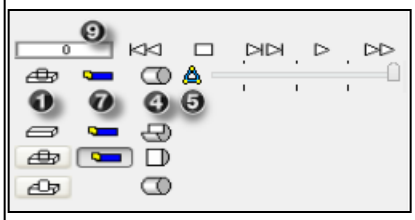
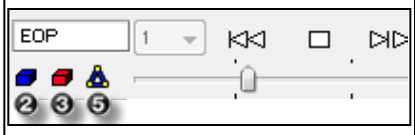
- H.264 Video
- Windows Media Video 7
- Windows Media Video 8
- Windows Media Video 9
- SMPTE 421MVideo

Frame Rate

Set the speed of the output video, in terms of how many frames to capture per second.

Video Dimensions

Set the width and height of the output video, either by entering the number of pixels or by using the current window view size.

Simulation	Legacy CPR	Rapid CPR
		
1. Tool Visibility Control 2. Stock Visibility Control 3. Fixture Visibility Control 4. Lathe Stock Cutaway	5. Overlay Geometry 6. Show Rapid Tool 7. Show Tool Holders 8. Show Machine	9. CPR/Rendering Visibility Controls

CPR/Rendering Visibility Controls

Tool Visibility Control

(Available for Operation, Tool, Machine Simulation, and Legacy CPR)

As with standard rendering, the hidden tool produces the fastest rendering; the opaque tool is the slowest of the options.

/ Invisible Tool:

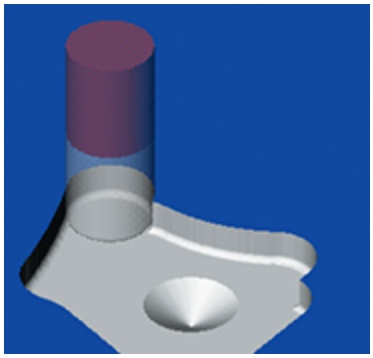
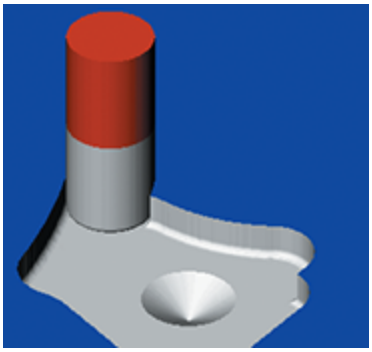
If this button is selected, the tools do not appear during the rendering process, although the material as a result of the tools cutting is removed (see example). Radii render more smoothly, and the part renders faster with this choice selected.

/ PartiallyTransparent Tool:

If this button is selected, transparent tools appear during the rendering process.

/ Opaque Tool:

Select this button to display opaque tools during the rendering process.

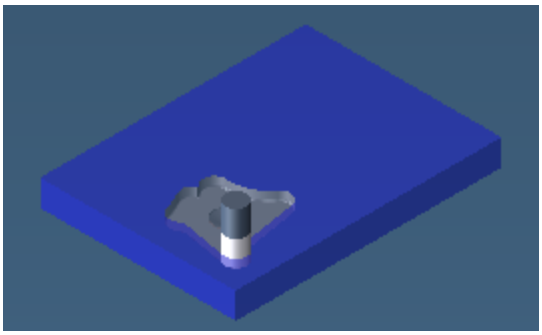
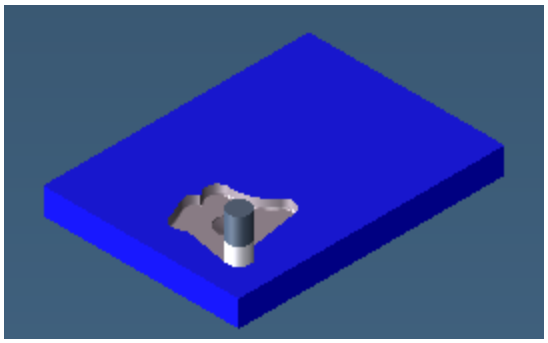



Invisible Tool	Partially Transparent Tool	Opaque Tool

Stock Visibility Control

(Available for Operation, Tool and Machine Simulation and Rapid CPR)

For Rendering, stock can either be displayed as Partially Transparent or Solid .

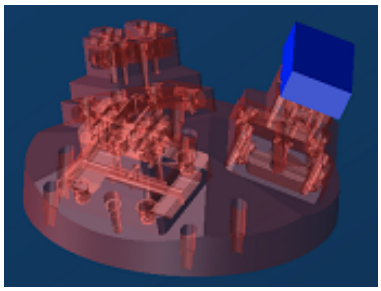
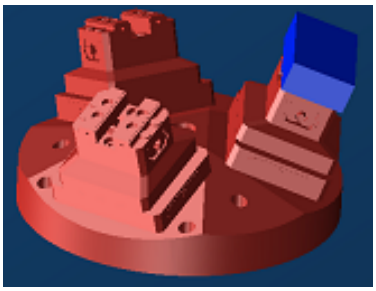



Partially Transparent Stock	Opaque Stock

Fixture Visibility Control

(Available for Operation, Tool and Machine Simulation and Rapid CPR)

For Rendering, fixtures can either be displayed as Invisible , Partially Transparent , or Opaque .

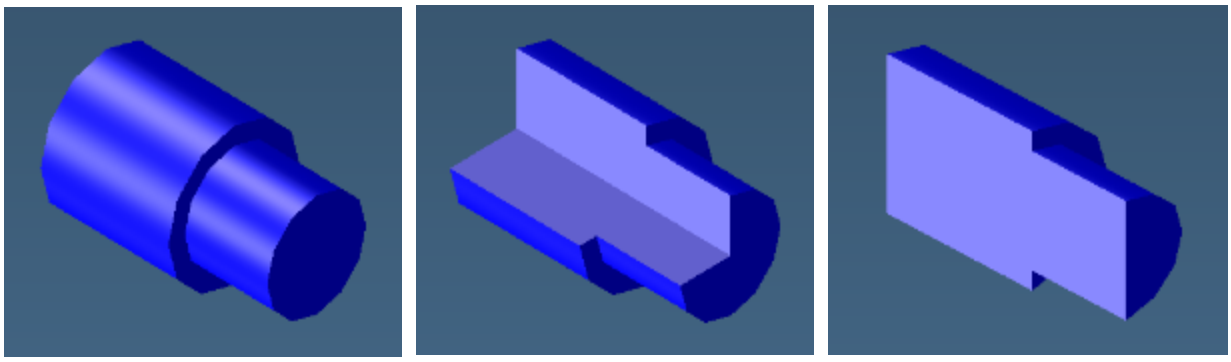



Invisible Fixture	Partially Transparent Fixture	Opaque Fixture

Lathe Stock Cutaway

(Available for Operation, Tool, Machine Simulation and Legacy CPR)

When rendering parts in the Turning, Mill/Turn, or Multi-Task Machining modules, there are three cutaway states for the stock cut part rendering. The three cutaway states are: No cutaway; 1/4 cutaway; and 1/2 cutaway.

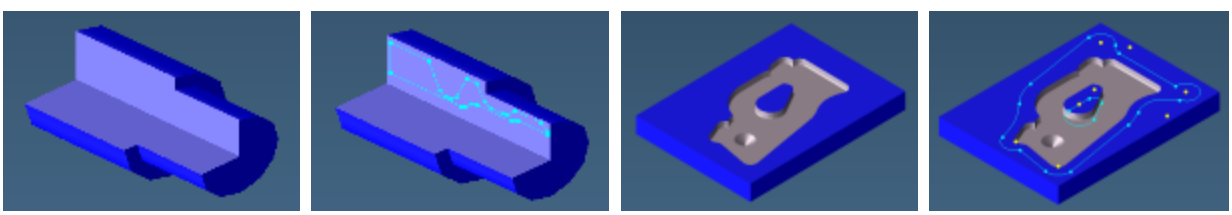


No cutaway	1/4 Cutaway	1/2 Cutaway

Overlay Geometry

(Available for all rendering options)

This option hides / shows workgroup geometry while rendering the part.



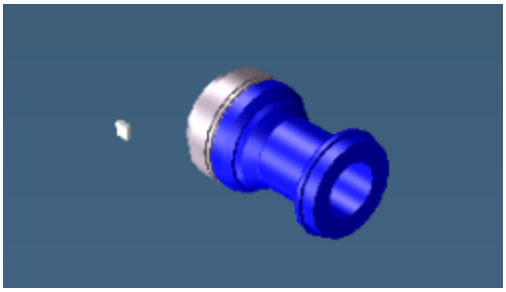
Turning: No Geometry	Turning Geometry	Mill: No Geometry	Mill Geometry

Show Rapid Tool

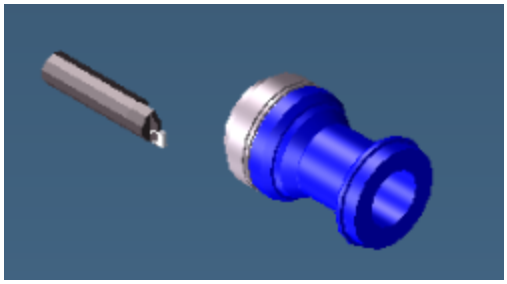
(Available for Operation, Tool and Machine Simulation)
The tool is rendered in a different color to show when it is Rapiding when this option is enabled.

Show Tool Holders

(Available for Operation, Tool, Machine Simulation and Legacy CPR)



No Toolholder



Show Toolholder

Show Machine
This option is only available in Machine Sim Rendering.

Simulation Control Icons

For full information on all controls in all Simulation dialogs, see the *Machine Simulation* guide.

The Rapid, Part, Tool and Machine Simulation palettes have various control options. To select an option, click the icon. More than one icon can be selected at once.

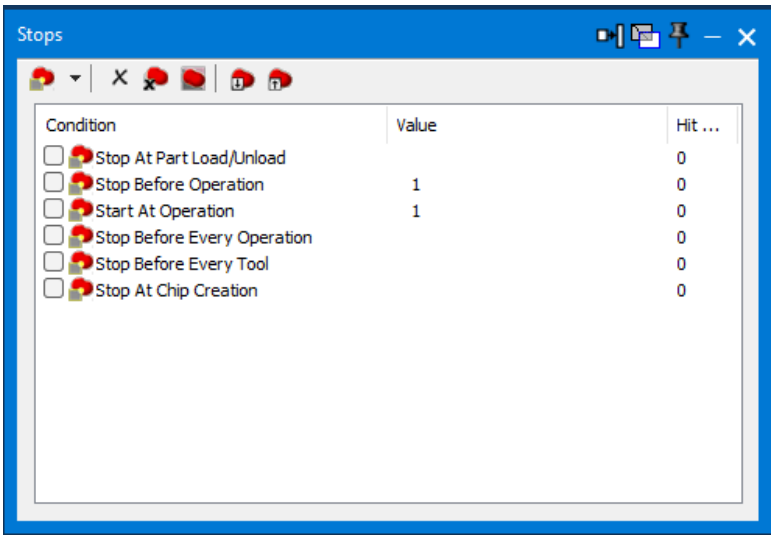
Stops

(Available for Rapid, Operation, Tool and Machine Simulation)

Clicking the Stops icon opens the **Stops** dialog, which lets you set one ore more conditions at which rendering should stop. Use the checkboxes to add/deselect options. Rendering will stop upon reaching the condition value specified in the **Stops** dialog.

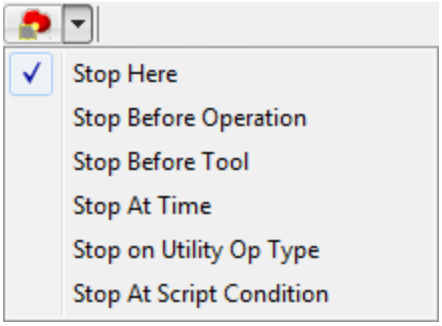
The **Stops** dialog initially displays stop options as shown in the following illustration. Select a stop using the checkbox, then double-click the option to input conditions. All options except Stop at Part Load/Unload provide a window to input specific operation/tool numbers, times, and so forth. When Stop At Chip Creation enabled, the simulation stops whenever the stock splits during material removal. So, can review the newly created chips and optionally define chip removal events at that point, although doing so is recommended but not required.

When a stop is active, it is highlighted with a red box, as shown below. If multiple stops have been set, use the Play button to go to the next stop.



Stops dialog

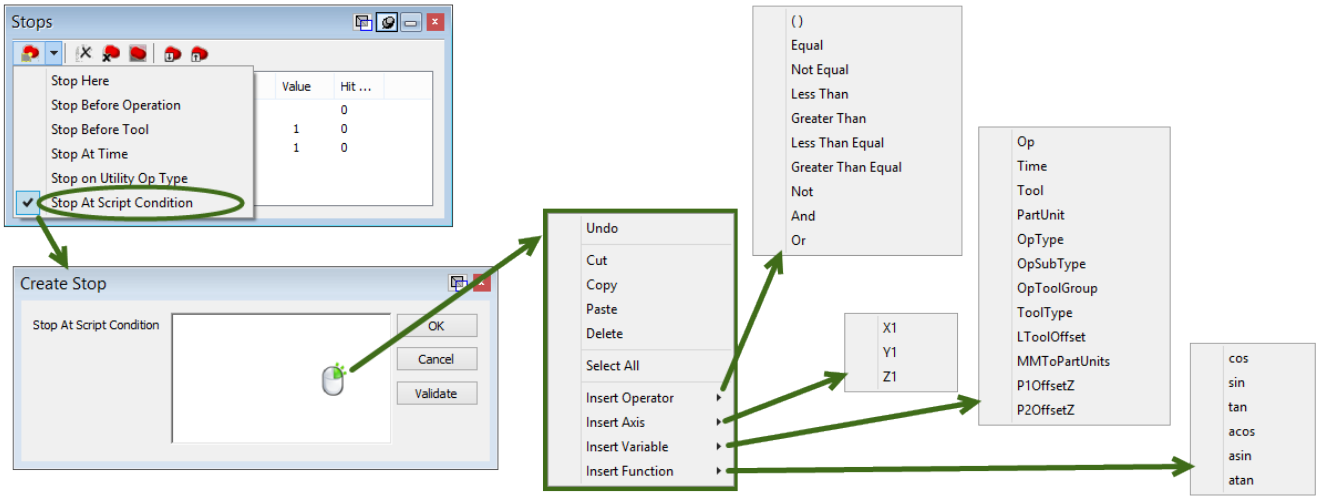
1. Dropdown menu
2. Delete selected stops
3. Delete all stops
4. Enable/disable selected stops
5. Import stops (`*.smd` file)
6. Export stops (`*.smd` file)



(1) Dropdown menu items for **Stops**

Additional stop options can be chosen from the dropdown menu. Scroll down the menu and select the desired option, then input required values in the dialog window. The Stop at Script Condition option has a right-click menu within the input window, providing the opportunity to input more specific conditions. Use the Validate button to ensure there are no script parsing errors.

Stop at Script Condition Options



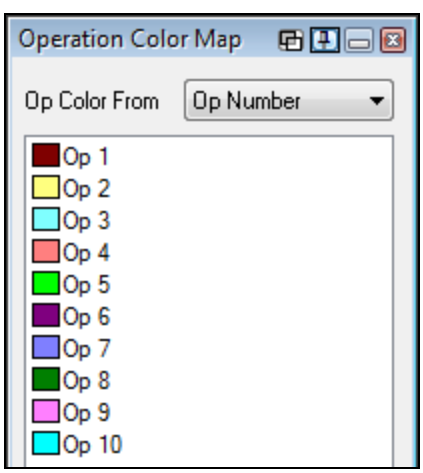
Tool motion on target body

(Available for Operation and Machine Simulation)

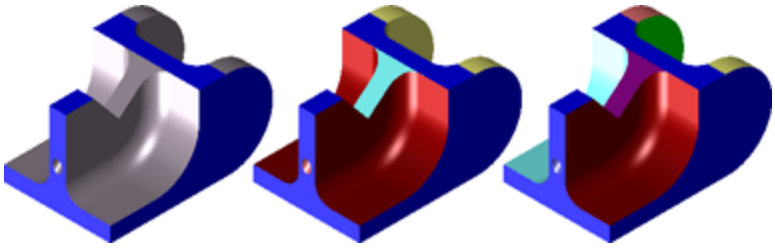
This option provides a non-cutting simulation of the tool motion. Selecting this option will increase the speed of the simulation, where the display of material removal is not as important as what the tool is doing.

Cut Color Mode

(Available for Rapid, Operation, and Machine Simulation)



This option lets you use color to enhance the functionality of CPR and Simulation. The corresponding options sub-dialog has three color modes: Cut Color (the default behavior, matching earlier versions of the software), Tool Number, and Op Number. The color of the remaining material changes with each operation or tool. In this way, you can see what areas of the part were machined by each operation, tool, or selected op. The color palette is generated automatically to create a set of colors that are readily distinguishable from one another. Double-clicking a color allows you to change it.



Example of Using Cut Color, Tool Number, and Op Number

Collision Checking

Available for Operation and Machine Simulation)

This option enables checking of collision events. The result of a collision event is controlled by setting the Collisions/Limits parameters. For details on these preference settings, see the documentation on File > Preferences > Display > Op/Tool and Machine Simulation Settings. If the Collisions/Limits is set to "Log To Display", then a report detailing any "collisions" is generated. The report, which includes when the collision occurs (Time), the XYZ value of the collision (Location), the operation, and the tool, can be saved out as a text file. Additionally, the Prim 1 section details whether a Tool or Holder was involved. The Prim 2 section reports whether the collision was with Stock or a Fixture. Using this option will slow down rendering speed.

Clash Console

Description	Time	(x, y, z)	Op #	Tool	Prim 1	Prim 2
Rapid Clash	0.0	2.540,-15.000, 30.000	1	7	Tool	Stock (1)
Rapid Clash	0.0	2.540,-15.000, 35.000	1	7	Tool	Fixture (1)
Clash	2.9	2.347,-14.028, 5.000	1	7	Tool	Fixture (1)
Rapid Clash	2.9	13.147,-19.393, 30.000	1	7	Tool	Stock (1)

Program error checking

(Available for Rapid, Operation and Machine Simulation)

This setting checks for any axis motion that is beyond the limits set in any machine component. If an axis limit is exceeded, a collision event is generated in the same way that it is generated when two components collide. The result of an axis limit exceeded event is controlled by the Collisions/Limits settings in the **Simulation Settings** dialog.

This applies to active operations only; inactive operations are unaffected. For more information about active and inactive operations, see *Common Reference*, chapter "Miscellaneous", section "Lists", subsection "Active and Inactive Operations".

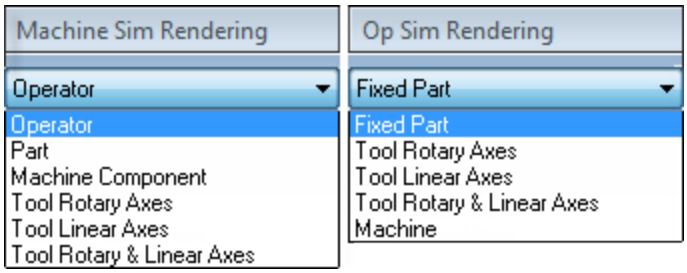
Please note that a Program error message will be displayed if axes are not properly set up or are missing in Machine Manager, even if this option is not enabled.

Point of View Lock

(Available for Operation and Machine Simulation)

This gives you control over how the virtual camera moves around the scene, defining the point of view during animation. The element chosen to be locked then becomes fixed in position.

The choices available for Point of View Lock will be different, depending on the rendering mode.



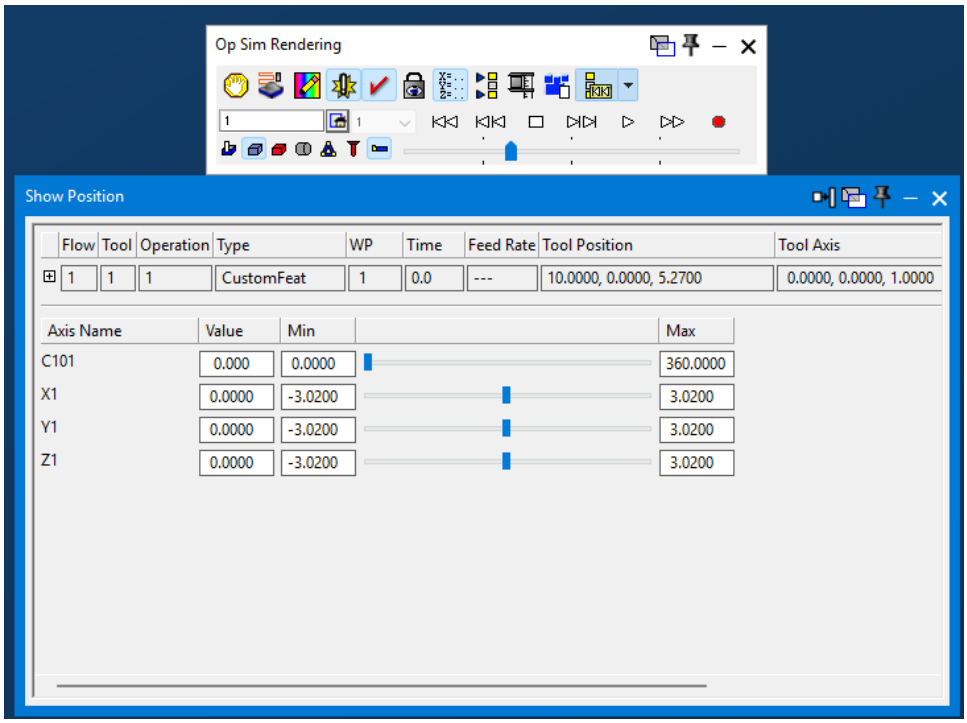
- Operator (Machine Sim): This fixes the view to outside the machine.
- Part/Fixed Part (Machine/Op Sim): This enables a pull-down menu to select the spindle to use to sync the simulation. GibbsCAM will focus on the stock, and the tools and machine will rotate around the part.
- Machine (Op Sim): This shows the viewpoint from within the machine. It is similar to Fixed Part, but shows stock moving/rotating. Tools do not move around it.
- Machine Component (Machine Sim): This locks the view to a particular machine component. Use the pull-down menu to choose which component is to remain stationary during simulation.
- Tool Rotary/Linear Axes (Machine/Op Sim) The view will lock on the Tool and its movements along the chosen axis.

Show position

(Available for Tool, Operation and Machine Simulation)

This option opens a dialog box showing continuous tracking of axis motion and current feature information. Sliders allow you to explicitly move machine axis. The **Render Tracking** dialog summoned by the Machine Sim Rendering palette might display more axes than the **Test Machine** dialog (in Machine Manager), which shows only the axes that are available in the kinematic tree. The Machine Sim **Show Position** dialog can potentially also display axis motion for custom axes defined by a VMM or by a script, such as for a Z601 axis for a bar loader.

For easy visualization, when you press Next Feature, in the **Rendering** dialog, the **Show Position** dialog shows the axes that move by changing the color of their values and sliders.



Skip Unselected Ops

(Available for Rapid, Operation, Tool and Machine Simulation)

This option reduces the rendering time.

The Skip Unselected Ops function preserves material state. This is equivalent to setting multiple Start at Op triggers. Selected ops will play in detail; unselected ops will have material removed immediately, and playback will skip ahead to the next selected op, so selected ops are always played in the correct program and material context.

If you want to skip material removal for unselected ops, deactivate the ops instead of deselecting them, and the program will simulate as if they were not present.

Analyze Cut Part

(Available for Rapid and Operation Simulation)

Note: To use Analyze Cut Part, you must select a solid before starting Simulation.

The **Analyze Cut Part** dialog provides several options to determine the results of the toolpath on a rendered part. This is an easy way to determine if there are areas on a body that are not being machined (Remaining Material Change) or if any cuts violate the part.

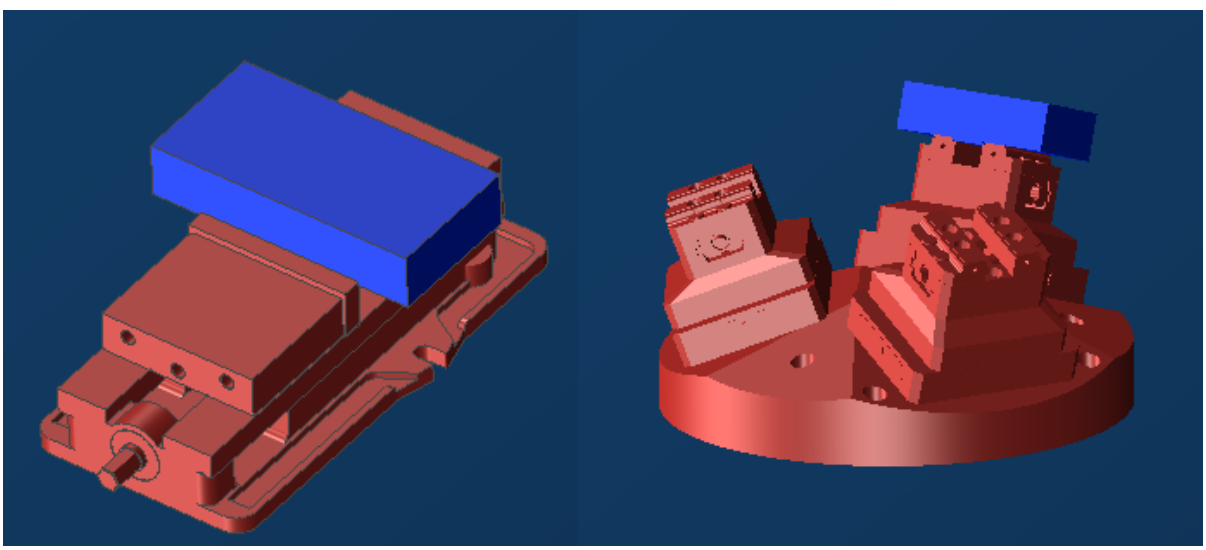
Show Multipart

(Available for Operation, Tool and Machine Simulation)

The Show Multipart icon is available on MDDs with Multi-Part capability.

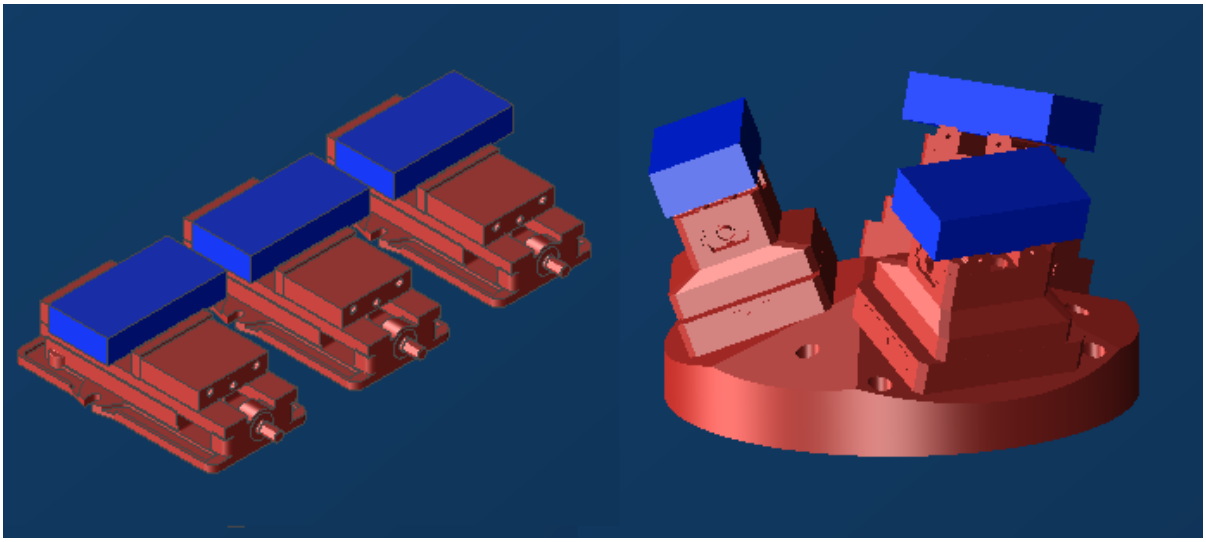
This icon appears only when Part Duplication is selected, or when the TMS with Multi Part option button is selected. For more information on the Multi-Part tab, see the *Mill* or *TMS* guide.

In Op Simulation, the Multipart icon shows as available to use but is OFF by default. In Machine Simulation, Multipart is ON by default and can be toggled off if desired.



Example of Show Multipart OFF

When Show Multipart is ON, all duplicate parts are shown. When using Part Duplication, the part and each fixture will be duplicated. When using TMS, the parts will be duplicated on the fixture.



Example of Show Multipart ON

Trace Options

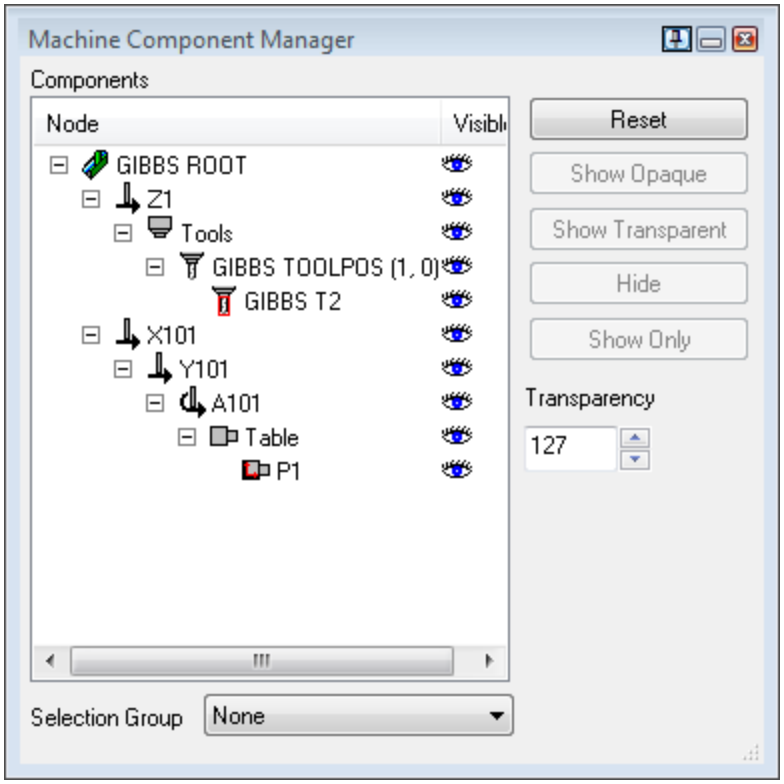
(Available for Tool Simulation)

This option let you choose how much of the toolpath will remain on the screen — 0%, 1%, 5%, or 100% of the toolpath — as the tool follows the toolpath. Trace Operation will draw one operation at a time. Trace From Run will clear the previous toolpath and begin redrawing if you stop the render midway through and then restart.

Machine Component Visibility

(Available for Machine Simulation)

This option opens a dialog box allowing control of the visibility of components in the machine assembly file. You can control the components individually and/or by the groups set in Build Machine. The eye icon lets you show or hide a component. Additionally, you can set a specific transparency level from `0` (invisible) to `255` (opaque).



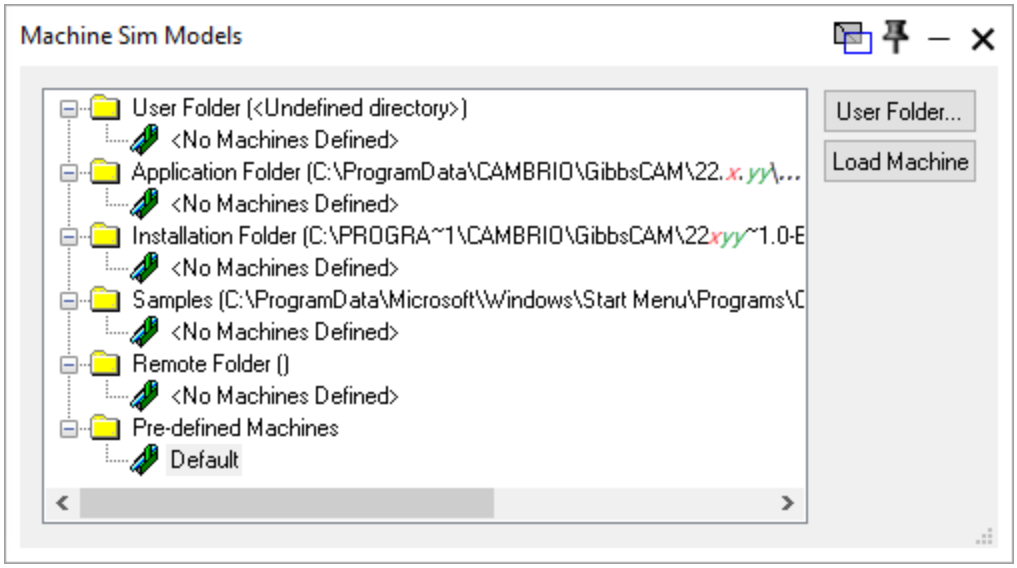
Don't Block Stock

(Available for Machine Simulation)

Don't Block Stock is a rendering mode that renders all fixture bodies that appear in front of the stock bodies as transparent. This lets you see material removal on stock bodies even when some other bodies move in front of them.

Load Machine

(Available for Machine Simulation)

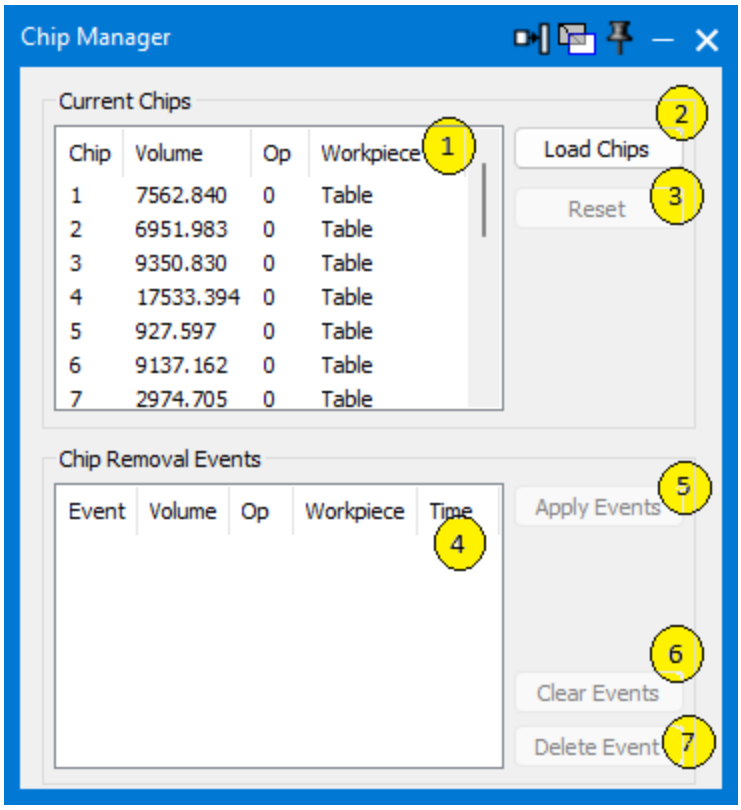


This option allows you to select which machine assembly file will be used for the current part. Once it is selected, the same machine will automatically be used for the part until a different machine is selected. Clicking User Folder allows you to select a directory that contains machine assembly files. Select the machine you wish to use and then click OK.

Chip Manager

(Available for Machine and Operation Simulation)

This option allows you to identify and remove specific chips generated during simulation. Removed chips are handled through chip removal events, ensuring they remain excluded from subsequent simulation steps and providing greater control and clarity during simulation.

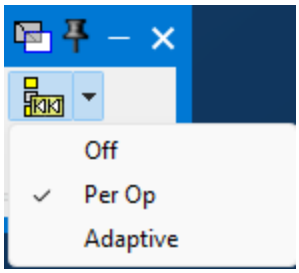


1. Displays stock body chips when loaded.
2. Detects and loads chips from the current simulation state. Chips are also detected and loaded automatically when the simulation are stopped either through manual interaction, showing the dialog, hitting a breakpoint, or program completion.
3. Clears chip removal events and reverts any chip removal done in the current simulation state. This button is only enabled when a chip has been removed.
4. Displays currently defined chip removal events.
5. Forces the execution of selected chip removal events against the current simulation state.
6. Removes all defined chip removal events.
7. Remove selected chip removal events.

Previous Step Mode

(Available for Machine and Operation Simulation)

This option allows you to save simulation states for the Active Flow by selecting a Previous Step Mode. Saved states can be revisited by clicking the Previous Step button.

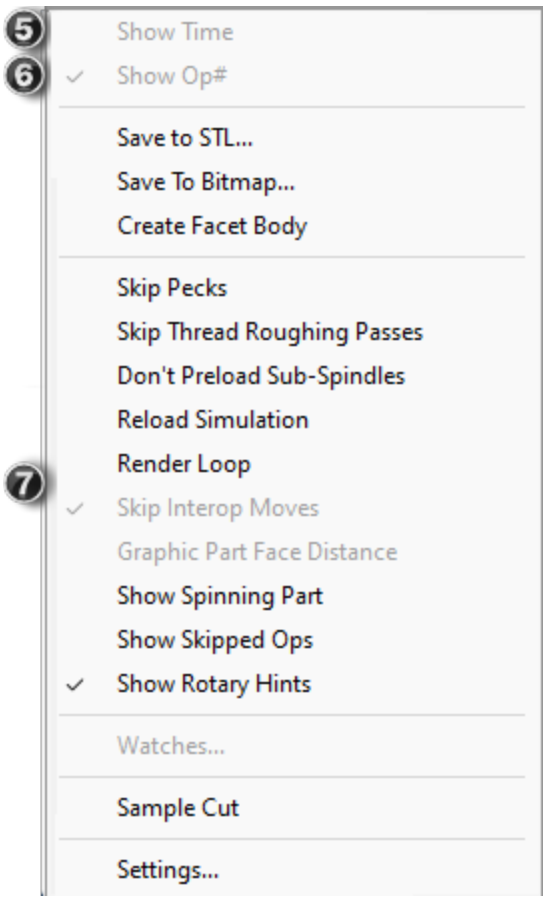


Rewind modes are:

- Off – no states are saved (default).
- Per Op – a state is saved at each operation change.
- Adaptive – a state is saved at each operation change plus each rapid/feed change plus the most recent five features of that operation.

Simulation Context Menus

To display the Simulation context menu, right-click the title bar of the Simulation Control palette. Left-click to select any of the options.

Legacy CPR	Simulation
	

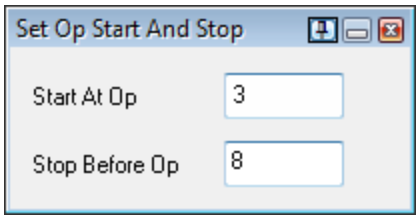
Legacy CPR Context Menu

The following options are found in the context menu of Legacy CPR.

Set Op Start/Stop #s...

This option opens a dialog to specify an operation at which to start rendering and/or set a stop point where rendering will stop and wait for you to start it again. The Start At Op option can be used when operations have already been rendered once and you would like to skip operations you have already seen. For example, you have a part with 10 operations and you know the first 7 operations are correct and render correctly but need to modify operations 8 through 10. Using this option, you can skip operations 1 through 7 and start at 8. As long as the tool or operation information for the first 7 operations does not change, the rendering will jump to operation 8. The Stop Before Op option will cease rendering before the operation you specify. If you press the Play button, then rendering will begin again.

> **Please Note:** This applies to active operations only; inactive operations are unaffected. For more information about active and inactive operations, see *Common Reference*, chapter "Miscellaneous", section "Lists", subsection "Active and Inactive Operations".



Use Op Start

Placing a check mark on this option activates the Start At Op option. Removing the check mark will disable the start point specified in the Set Op Start/Stop #s dialog.

Use Op Stop

Placing a check mark on this option activates the Stop At Op option. Removing the check mark will disable the stop point specified in the Set Op Start/Stop #s dialog.

Stop Before Load/Unload

Placing a check mark on this option activates the Stop Before Op option. Removing the check mark will disable the stop point specified in the Set Op Start/Stop #s dialog.

All Context Menus

The following options are found in context menus of both Legacy CPR and the Simulation options.

Show Time

23.4 This will set the Current Display to show the elapsed cut time.

Show Op

3 This will set the Current Display to show the current operation number.

Render Loop

Replays the simulation until the user presses the stop button.

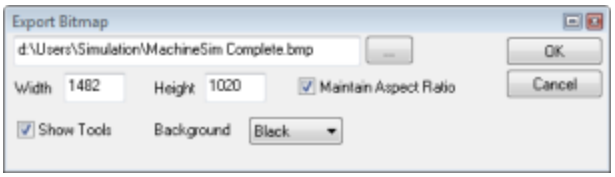
The following options are only to be found in the Simulation dropdowns:

Save to STL

Select this option to save a copy of the simulation to an .stl file for viewing later.

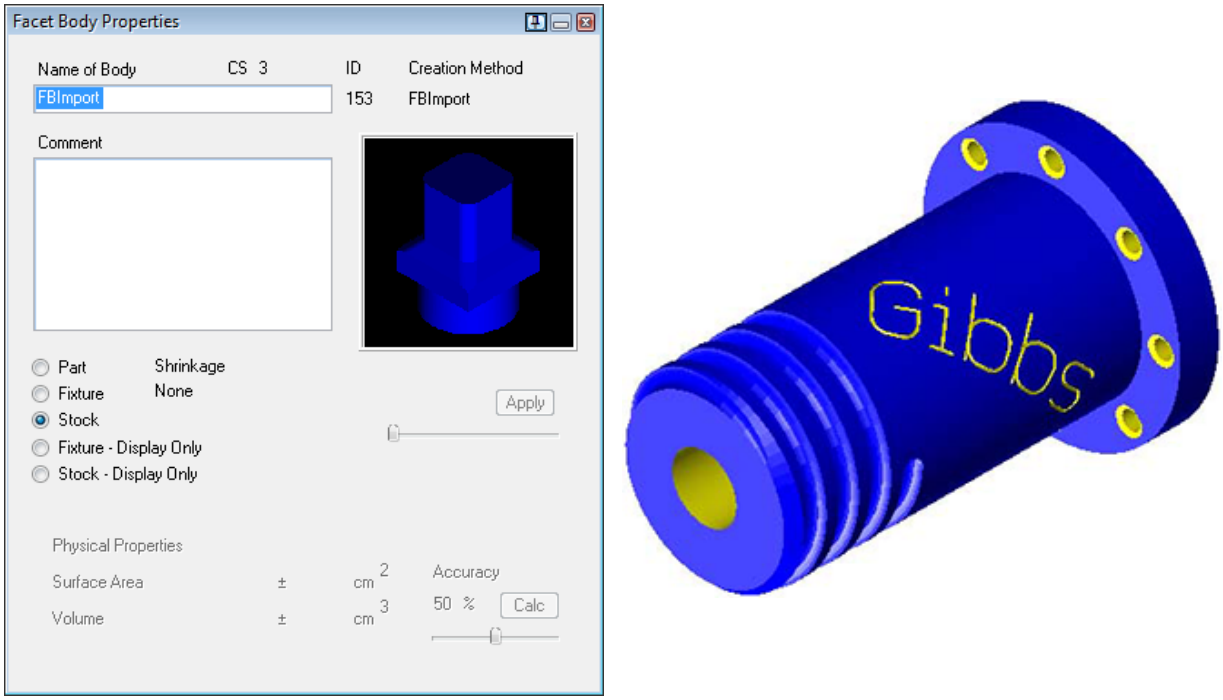
Save to Bitmap

Select this option to save a copy of the simulation to a bitmap file. This will re-render your current state at the specified resolution and save it as a picture. Please note that some video cards will not support saving a picture over 4000x4000.



Create Facet Body

This function turns the current cut rendered condition into a facet body. The facet body will appear in the workspace as a transparent body. Facet bodies can be regarded as other solid bodies: They can be queried, profiled, sliced, and machined. One of the uses of facet bodies is that they can be set as stock for “display only” purposes; i.e., they are not used as a stock condition for creating toolpath but they can be shown in rendering. Setting a facet body as stock can be quite useful for saving a rendered condition so you can instantly get to later operations.



Example of a Facet Body used as stock.

Skip pecks

The Skip Pecks option does not render any pecking moves used in drilling operations. Pecking is still generated. This option simply reduces the rendering time.

Don't preload Subspindles

Disables sub-spindle stock generation. This improves performance with the tradeoff of not having accurate stock on the sub-spindle during the first run. Not available for Tool Sim.

Reload Simulation

This option reloads the simulation.

Render Loop

This option causes the simulation to loop continuously.

Skip Interop Moves

Switches off the display of inter-op moves. Available only for Op Sim.

Graphic Part Face Distance

For use with parts containing multiple spindles. When enabled, this will render simultaneous display of multiple spindles. Available only for Op Sim, and standard for Flash CPR.

Show Spinning Part

Displays a transparent solid that represents the stock spinning at high speed, projected over the subspindle stock. This transparent solid image is called a *spun outline*. This option is useful for visualizing stock that is asymmetrical or has deep concavities. It is not useful if the stock is simply a revolved profile, because the spinning stock is identical to the stock at rest.

Show Skipped Ops

> **Please Note:** Use in conjunction with Skip Unselected Ops enabled.

Shows current stock condition of skipped ops. This condition includes the work performed by the ops that have been skipped, up to the point of the unskipped op.

This is useful for showing only a certain few ops, without having the unmachined material of the skipped ops in the way of the render.

Show Rotary Hints

Displays a green arc arrow that indicates the spindle direction.

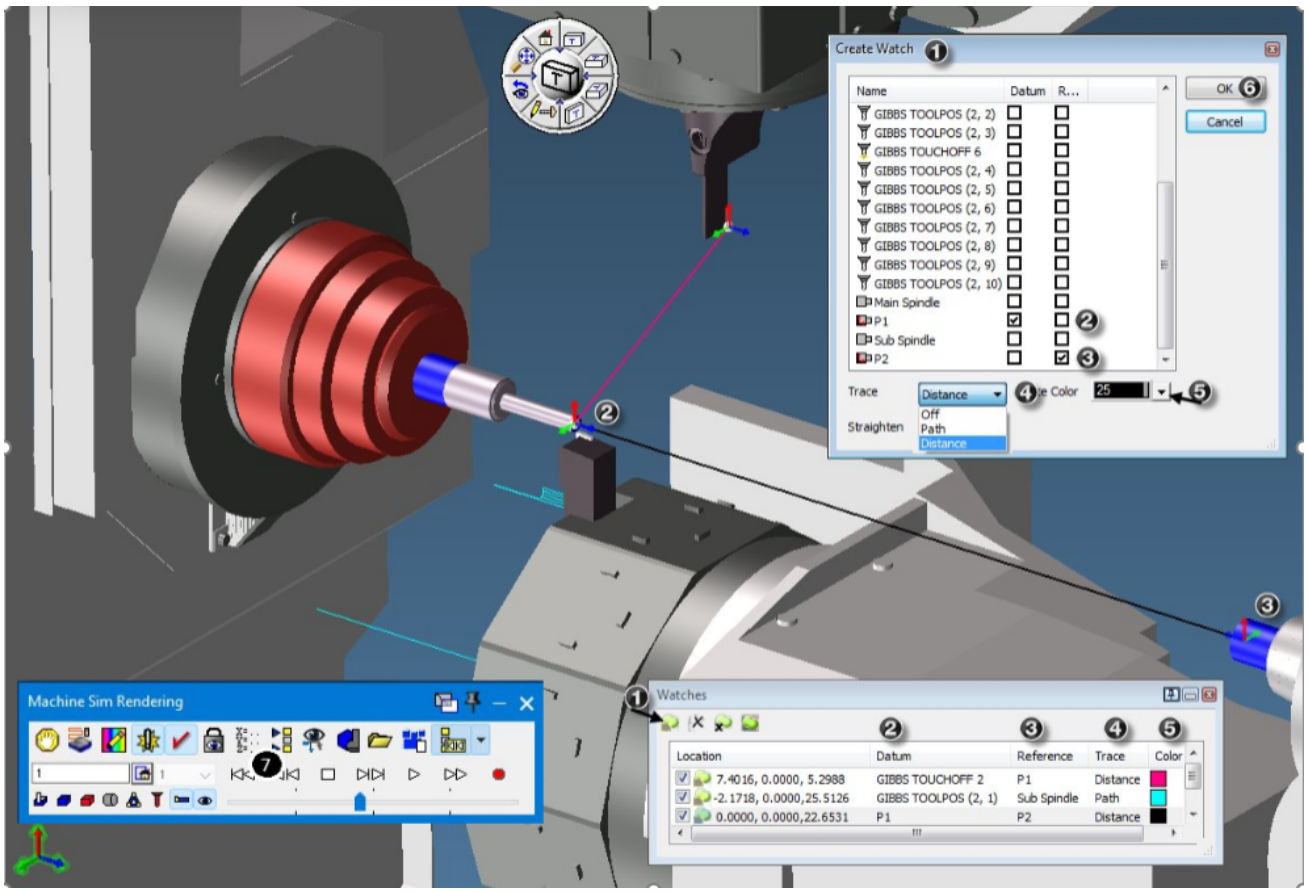
Watches

The purpose of the Watches function is to graphically display, trace, measure, and analyze the relationship and the location between chosen point A (Datum) and point B (Reference). These points can be set to almost any item defined in the current Machine Model. You can choose from the Trace dropdown either to track Distance between points or to trace the Path they take.

Multiple watch combinations can be created and displayed at the same time. You have the ability to differentiate them by Trace Color. To enable or disable watches, check or uncheck the checkbox. To edit a watch item, double-click it. You can import and export watches as `*.smd` files.

The location measurements are updated continually during rendering, and are displayed on the **Watches** dialog. Path Trace lines are shown in the workspace in their chosen color. Distance lines also display an axis block at each end. Available only for Machine Sim.

Watch example



Steps to creating a Watch

1. Click the Create watch Icon in the Watches dialog. Result: The Create Watch dialog opens.
2. Set Datum
3. Set Reference
4. Set watch to either Path or Distance (or turn Off)
5. Set color
6. Click OK. The Create Watch dialog closes.
7. Rewind Rendering and play. Path and Distance markers display in the Workspace. Location data displays in the Watches dialog.

Sample Cut

Enables/Disables MachineWorks SampleCut technology. Sample Cut is a MachineWorks technology feature that minimizes memory used by simulating material removal at the expense of accuracy. (Material removal is accomplished using a 3D grid space instead of the standard facet-body booleans). This means that an application will only use a fixed amount of memory to render an arbitrarily complex toolpath. This can be of benefit in particularly dense surfacing toolpath where users have sometimes run up to the memory limits of their computer.

Settings

This option opens the relevant Sim Settings dialog. There are separate dialogs for Op/Tool settings and Machine Sim settings. (Rapid cut uses the Op/Tool Simulation settings dialog.) Although both dialogs are essentially the same, they save separate data files. For details on these

preference settings, see the documentation on File > Preferences > Display > Op/Tool and Machine Simulation Settings.

Printing the Rendered Image

After the rendering is complete, the rendered image can be printed, either in either black and white or in color. When the rendered image is on the screen, summon the system **Print** dialog using either the standard keyboard short (`Ctrl+P`) or by choosing File > .

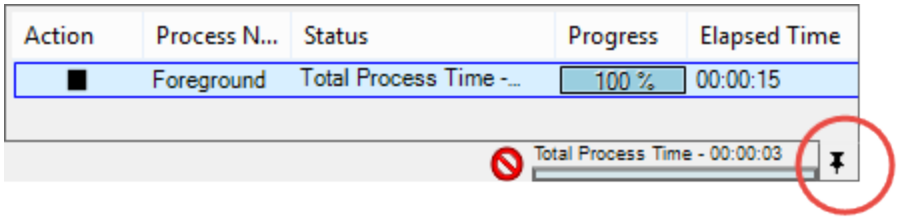
Depending on your printer, you might have very fine control over each printout using the options in the system **Print** dialog. To adjust default printing preferences, using File > , Display tab, Printing section. For more information, see “ Display Preferences ” on page 16.

Status Bar

At the bottom of the workspace is a Status Bar which includes a message display. To the left of a progress bar, the following items are displayed: current Coordinate System, current Workgroup, current unit of measurement, and current TMS status (such as number of parts) are displayed.

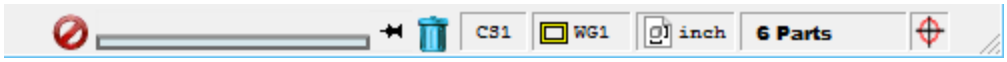
Double-clicking on the Workgroup and CS indicators will open their respective dialogs. If present, double-clicking the TMS status indicator will open the DCD dialog to the Multi-part tab.

Hovering over or clicking the progress bar reveals the Progress dialog, which will display real-time feedback of toolpath calculations and timings. Use the pin symbol indicated below to keep the dialog open.



The messages displayed indicate the current function the system is performing, and the progress bar gives the percent-completed status of the function. Click the Stop button to stop the current function, if required.

Some Plug-ins, notably Show Position, can be added to the status bar. To do this, click and open the Show Position plug-in, then right-click the Show Position title bar and check the option to Install to Task Bar.



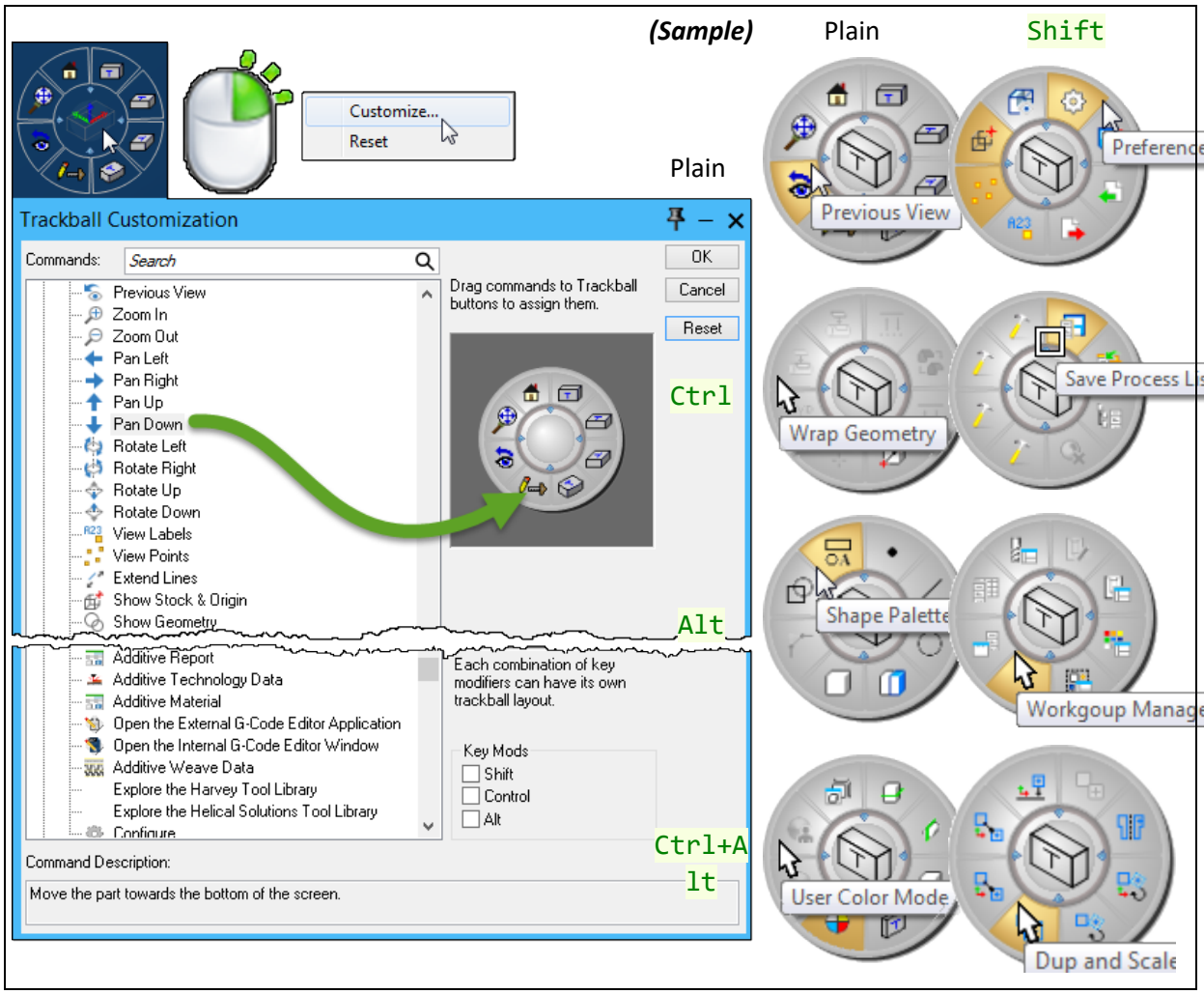
To uninstall the plug-in, right-click the icon on the Status bar and select the option Uninstall Task.

Customizing the User Interface

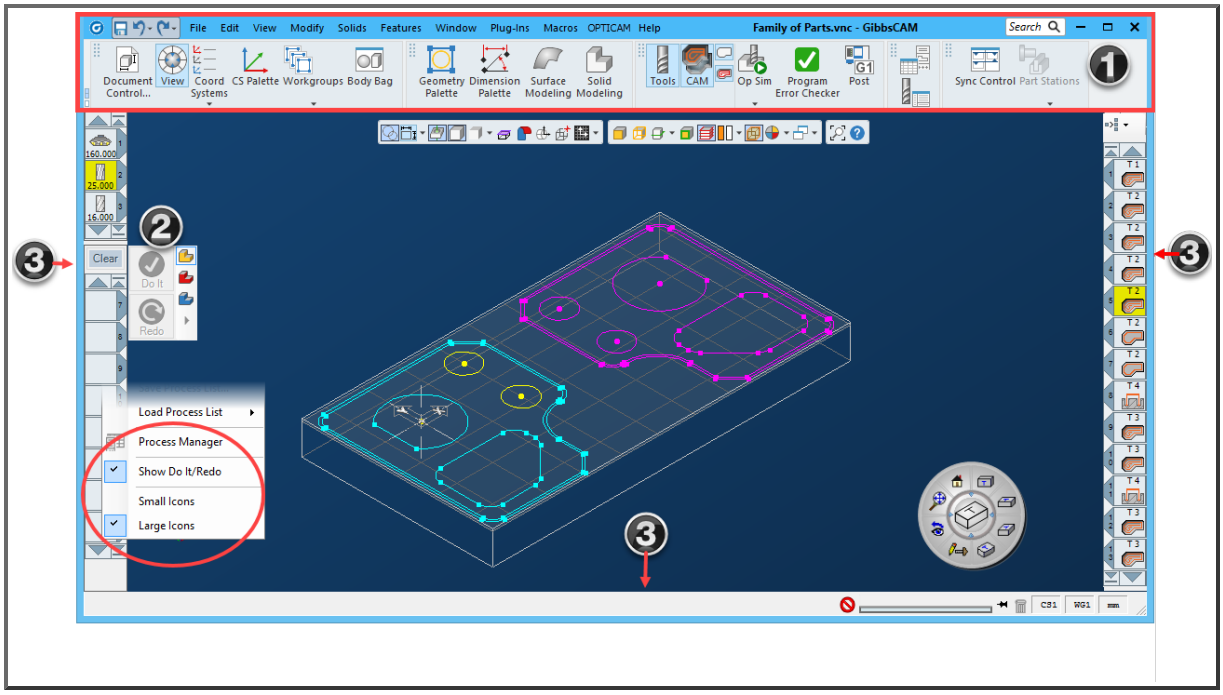
Command Shortcuts in Trackball

When you right-click the trackball, its context menu now offers a powerful choice: Customize.

The **Trackball Customization** dialog allows you to drag any command to any of the eight slots of the trackball, using any of eight possible combinations of key modifiers (`Shift`, `Ctrl`, and/or `Alt`). This puts up to 64 commands at your fingertips: Press `F5` to summon the trackball, press up to three modifier keys (the trackball display changes accordingly), and click a sector of the trackball.



Customizing the Workspace

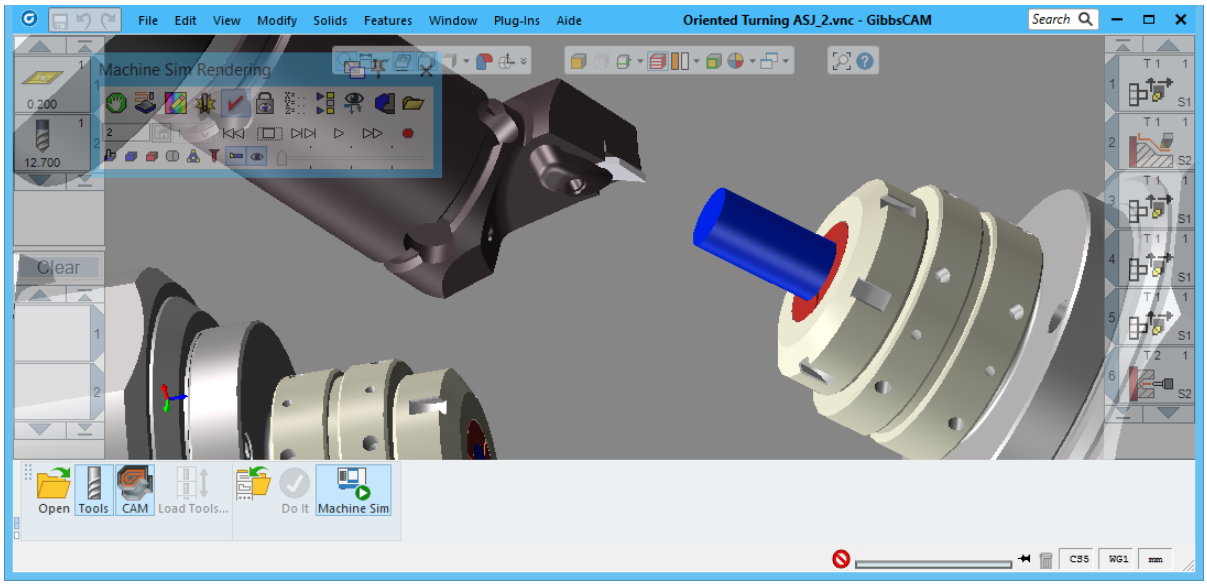


1. Right-click here to Customize
2. Do it/Redo Command.
3. Alternative command docking areas

Some customization is available within the workspace. For instance, from the tile right-click menu you can select large or small tile size. Right-clicking a process tile also enables the Do it/Redo controls that are attached to the process tiles to be hidden or shown. (The Do it/Redo commands are still available at any time by right-clicking in the workspace.)

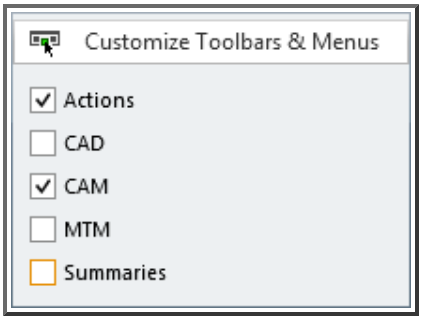
The File>Preferences>Interface tab has options to control the Startup window size, the Transparency of the Floating Toolbar, the dialogs and the tile lists and also various other controls.

Other more sophisticated customization is possible by right-clicking within the header area highlighted within a red box in the screenshot above. You can totally rearrange, or duplicate command icons and place them into existing or custom Toolbars and Menus. For instance this would enable you to create a command menu specifically for your own oft-used workflows.

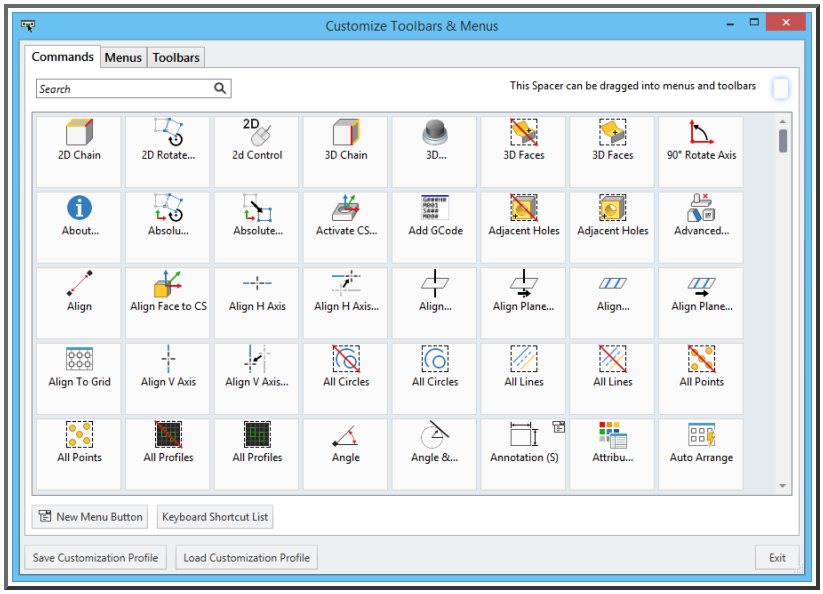


Customizing Toolbars and Menus

Access the Toolbar and Menu customization function by right-clicking anywhere in the Title or Commands Toolbar. When you are in customization mode, functionality that affects the workspace is temporary disabled. The following dialog will appear:



The checkboxes in the Customization menu can be used to quickly show/hide the GibbsCAM Toolbar groups. Click the option Customize Toolbars & Menus to open the main customization dialog.



Multi-Select Mode and Range-Select Mode options

These commands can be added through Customize Toolbars & Menus and are disabled by default.

Multi-Select provides selection behavior equivalent to using the CTRL key, while Range-Select behaves like the SHIFT key, including support for drag selection. Selection mode remains active even after clicking within the graphical window, allowing continuous selection of operations without repeatedly reactivating the command. Only one selection mode can be active at a time, and active modes are visually highlighted for clear identification.

Save/Load Customization Profile

These options enable you to save and load your customizations to and from a `*.cui` file. This way it is possible to set up customizations for specific tasks.

You can also load a copy of the as-shipped default UI at any time.

> For your convenience, three preset interface `*.cui` files are available.
>
> A Level 2 Legacy Interface is provided. This allows you to use Level 2 as it was before GibbsCAM 12. Or, you can choose from the standard GibbsCAM 12 Level 1 and Level 2 Interfaces.
>
> To load an interface, from the bottom of the Customization main screen, click the option Load Customization Profile. Navigate to the following folder:
>
> C:\Program Files\Gibbs\GibbsCAM\<*version*>\User Interface Layouts
>
> Double-click the required option.

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

New Menu Button

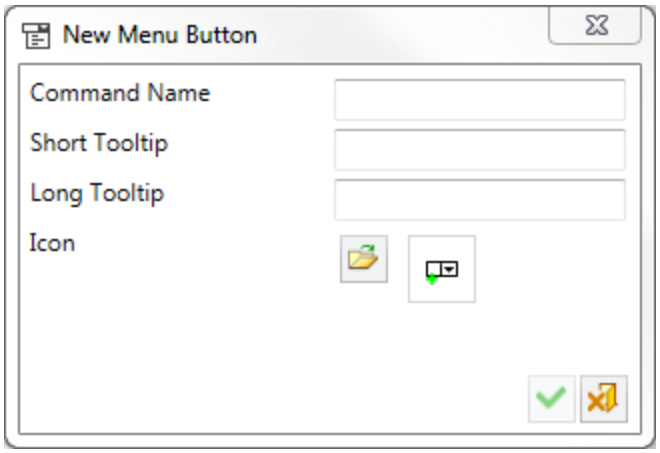
The New Menu Button dialog lets you name a new command container and provide tooltip text strings and an icon for it. After a command container has been dragged onto a toolbar or into a menu, you can drag one or more commands into it.

Command Name
The text string that will label the command container.

Short Tooltip
The text string that will appear when the mouse hovers over the command container.

Long Tooltip
The text string that will appear on mouseover if Balloons are turned on.

Icon
If you do not use the (Browse) button to navigate to and choose an `*.icn` file, then default icon will be .

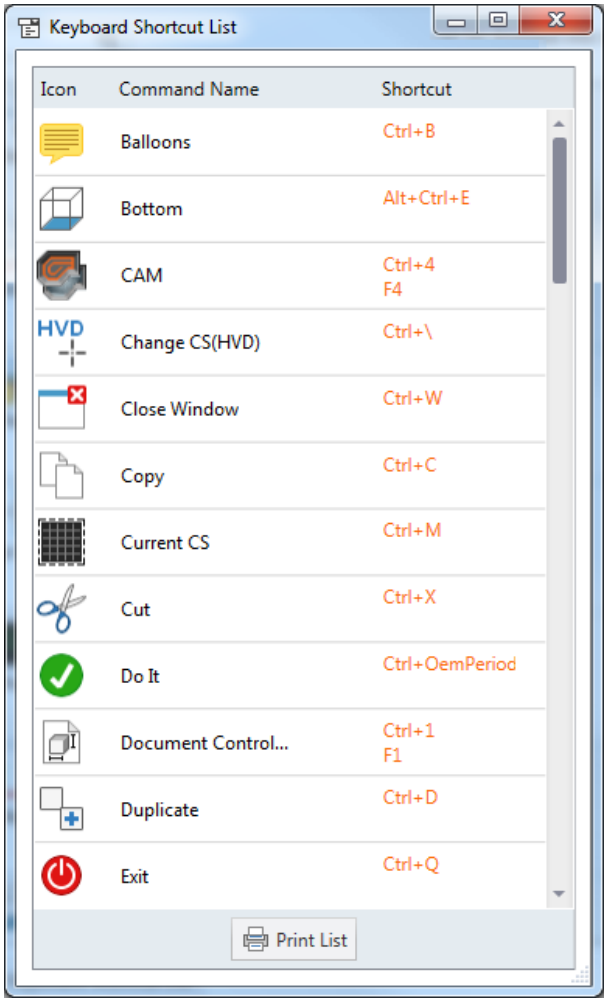


OK: Accept changes and close the dialog.

Cancel: Discard changes and close the dialog.

Keyboard Shortcut List

The Keyboard Shortcut List dialog lets you display and print a list of all commands that have keyboard shortcuts assigned.



Keyboard Shortcut List

Icon	Command Name	Shortcut(s)
	Balloons	Ctrl+B
	Bottom	Alt+Ctrl+E
	CAM	Ctrl+4 F4
...	...	...
	Zoom In	Ctrl+OemPlus Alt+DOWN
	Zoom Out	Ctrl+OemMinus Alt+UP

Commands tab

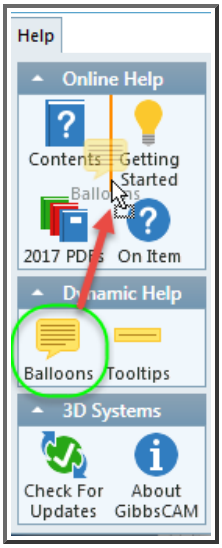
This tab contains all the GibbsCAM command icons available for UI customization.

Search

A search function is provided at the top of the screen. This searches not only the command titles, but also the Tooltip descriptions to enable you to find the correct command.

Shortcuts

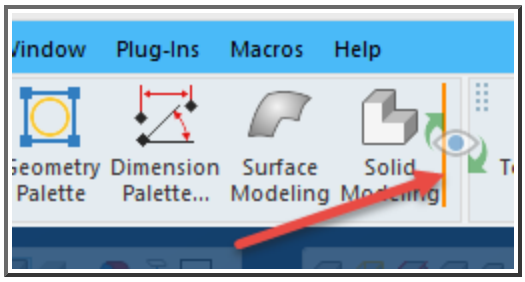
The commands tab is also used to set up new keyboard shortcuts. Click to highlight an icon. A box will appear if it is available for shortcuts. Simply hold the Ctrl or Alt key, together with the character you wish to use for the shortcut. If the shortcut is not already in use, it will appear in the box. Click any other icon to enter, or click the x next to the box to delete. If a shortcut is already in use, a warning will be displayed. You then have the opportunity to reassign the existing shortcut, or choose another.

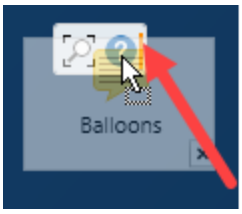


To insert a command

Click, hold and drag the required command icon either into a new or existing command group or main menu dropdown. If you hover over a Main Menu option (File, Edit, View etc.) the dropdown will automatically open. Move the icon around and you will see an orange bar. Once this bar is positioned just to the left of where you wish to insert the command, let go the mouse button and the icon will drop into place.

When in Customization mode. a command can be inserted into either a Menu dropdown, the Floating toolbar or an existing Toolbar Command group. Items placed anywhere else will disappear into the trashcan. It is especially important that you are aware of this if you are moving existing icons. If inadvertently deleted, they must then be reinserted.



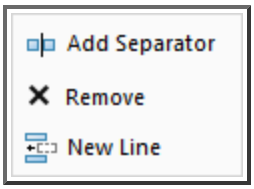


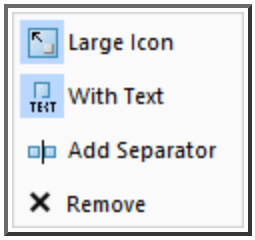
Reposition an icon

If your command has been place incorrectly, simply click, hold and drag it into the correct position. All existing menu commands can also be moved around in this manner.

Right-click menus

Right-clicking a command icon in the UI while in Customization mode gives additional options to arrange the commands by adding separators or extra lines. Please note that only one separator can be added in a particular space. Clicking Add separator a second time will remove it.

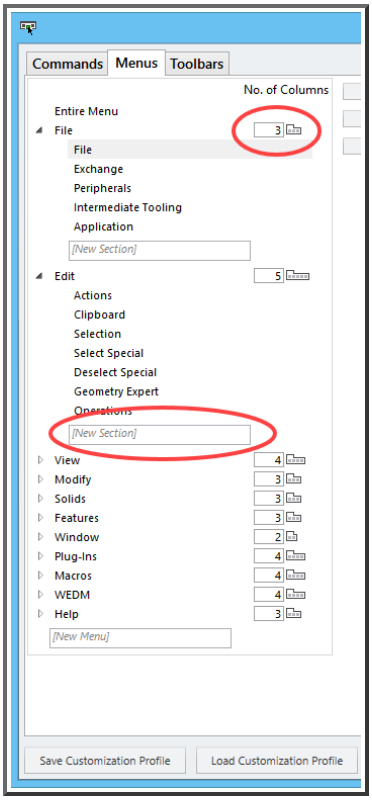




Right-clicking an icon in the Main-menu dropdown

Right-clicking an icon in the toolbar

Menus tab



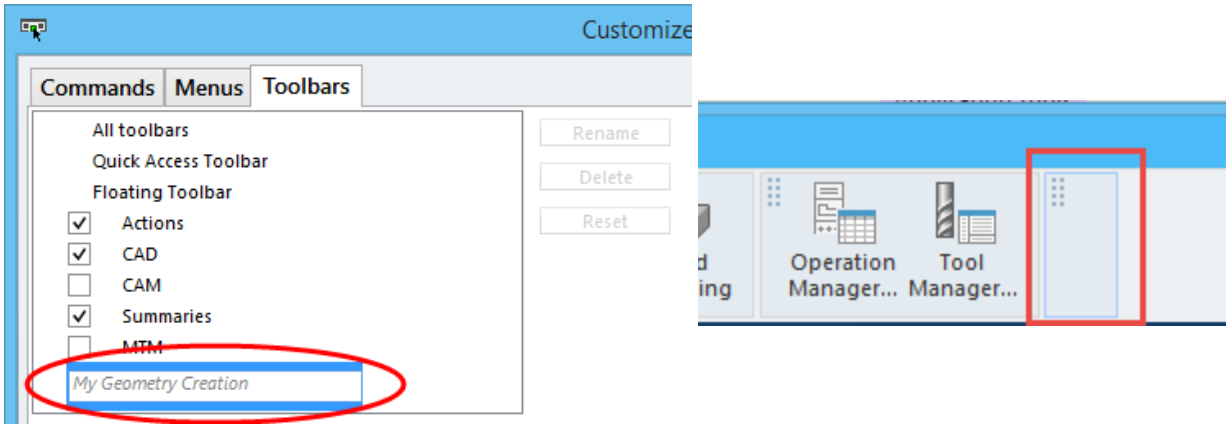
Use the Menu tab to create a new Main Menu option or to add a sub-section to an existing menu. You can also change the width of the Main menu dropdown - just type in the required number of columns and click the column icon next to it. Menu options can be renamed, or if need be, deleted.

Rename/Delete/Reset Buttons

You can rename or delete any menu or toolbar. The Reset option will reset sub-menus back to the default condition.

Note: Take care not to delete top level menu items (Main Menu options like File, Edit. View etc that appear in the header bar), or the actual command Toolbar groups, unless this is what you intended. The Reset option will not work for these. The only way to recover these deletions is to reload from a saved Profile.

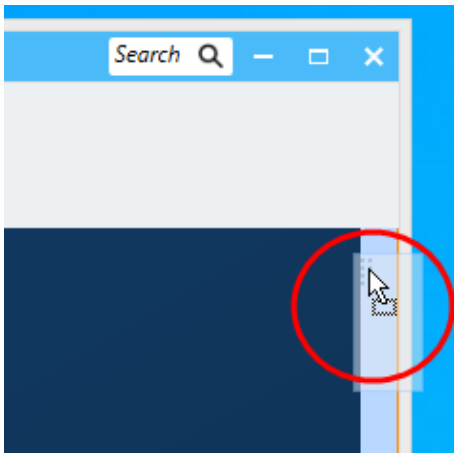
Toolbars Tab



Enter a title into the New Toolbar box and hit return. A new Toolbar Command Box is created. You can then drag command icons into this new box from the commands tab. The new menu will also appear on the Customize screen and can be switched on/off using a checkbox, just as with the default groups.

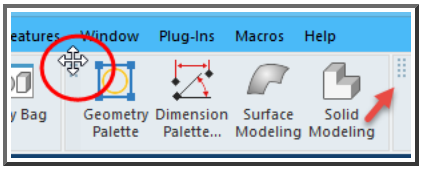
To add items to the Quick Access or Floating Toolbars, just drag the required command into the toolbar.

Positioning Command Toolbar Groups



Command Toolbar groups can be pinned to the outer margins of the workspace (Left, right and bottom).

In Customization mode, click and drag the Toolbar Command Box grip point (the group of 8 dots in the top left corner of each toolbar).



The cursor will change to a white arrow and an orange line appears when the group is in a location where it can be inserted.

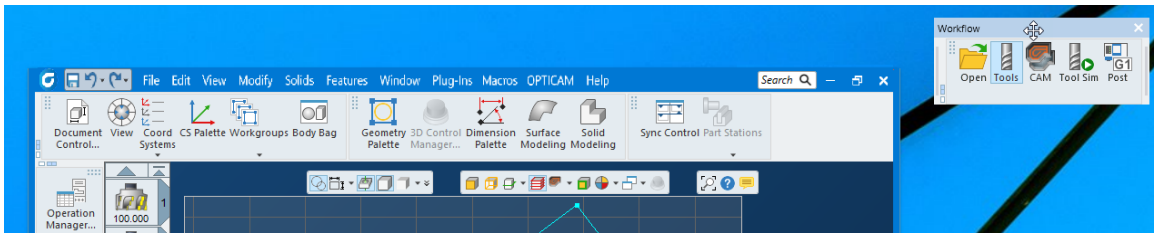
The orientation automatically adjusts to fit the available space.

Exit

Once you have finished customizing the UI, click the close or Exit button (situated in the bottom right corner of the customization dialog). This will save the changes and return to normal GibbsCAM functionality.

Moving Command Groups around the workspace.

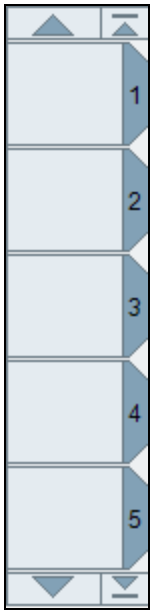
Once you have exited customization mode, you can click and drag command groups to any location on your screen, or even onto a different screen. When you close and reopen GibbsCAM the group will appear where you last left it. To return a group to a docked position, drag the group using the grip point, this will activate the orange guideline and the group will then snap into place. You do not need to be in Customization mode to do this.



Miscellaneous

- Lists
- “Other Workspace Items” on page 218
- “Shortcuts” on page 219
- “Cursors” on page 220
- “Eyeball icon” on page 222
- “Freehand Actions” on page 222
- “Right-Click” on page 224
- “Colors” on page 225
- “Math Functions” on page 226
- “Interrogate” on page 226
- “About Clearance Volume ” on page 228

Lists

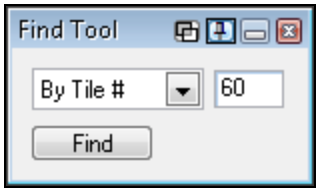


There are three main lists in the system. These are the Tool List, the Process List and the Operations List. These lists hold tiles and are not moveable. Each tile is used to describe an individual tool, process or operation. A list can hold many items.
To scroll through a list, click on the large blue arrow at the top or bottom of a list. This will move the list up or down one item at a time. Lists may be scrolled through quickly by `clicking` on the scroll arrow and `dragging` the mouse past the arrow. This is called Speed Scrolling. When Speed Scrolling, the list advances six tiles at a time. The arrow will turn red when Speed Scrolling.

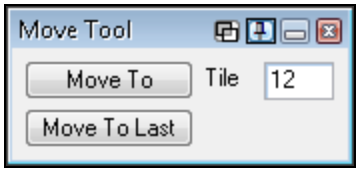
You can also scroll to the first or last non-empty tile by clicking the smaller button next to the single-scroll button.

Hold down the `Shift` key and double-click on the triangular space between process tiles to create an empty space between processes. Hold down the `Shift` key and double-click on an empty space between process tiles to delete the space between processes. To navigate through the lists faster, use a context menu command.

`Right-clicking` on either of the arrows in a list will bring up a menu with the Find command. `Right-clicking` on a tile will bring up a menu to select the Find or Move dialogs.

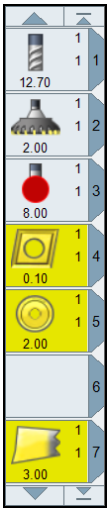


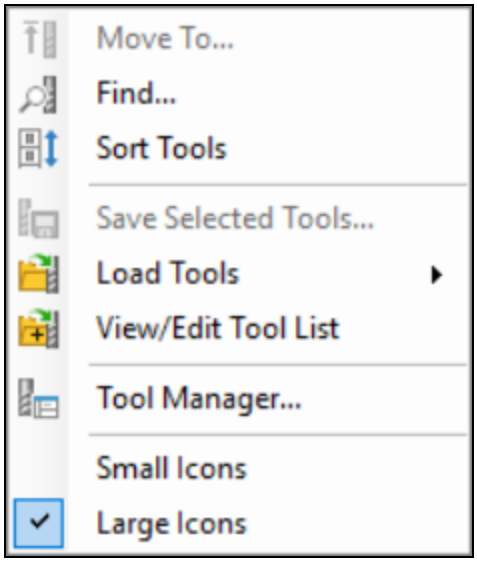
Find Tile dialog



Move Tile dialog

Tool List





The Tool List and its (right-click) context menu

The Tool List is used to define the tools used to cut a part. When you `double-click` an empty tile, a new tool is created and the Tool dialog opens. A tool is then fully defined by the parameters presented in the Tool dialog. This includes, but is not limited to, the type of tool and its size and material. Only one Tool dialog can be open at a time.
Tool tiles display a graphic of the tool type and the size of the tool. This instantly changes with any modifications to the tool. Tool Lists may contain a mixture of milling, turning, broaching, and probing tools at the same time. Each tool has a unique graphic to help you quickly determine what each tool is.

The Tool List is toggled on and off by the Tool button in the Commands palette. There can be a maximum of 999 tools in a part. Tool tiles do not have to be contiguous; there can be empty spaces between tiles.

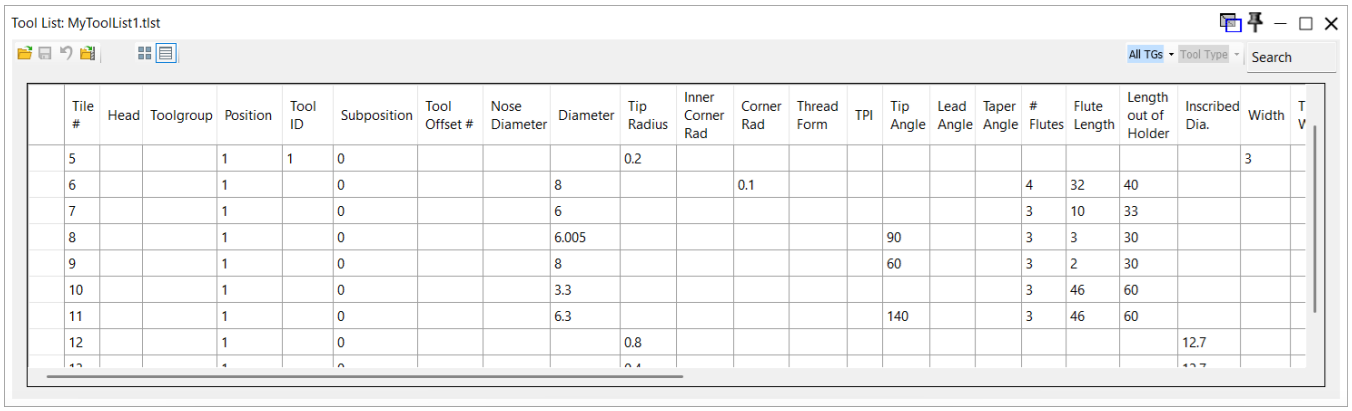
To navigate through the Tool tiles, click on any part of the Tool List to select it. Then use the arrows above the tool list to scroll up one tile ( ) or to the top ( ), or the arrows below the list to scroll down one tile ( ) or to the bottom ( ), Or you can use the mouse wheel to scroll through the list. `Ctrl+`mouse wheel scrolls through multiple tiles at a time. The Process List and Operations List, described elsewhere in this section, can also be navigated in this manner.

Tool lists can be saved and even automatically generated by using default part files. For more information on default parts, see “ Type Default Part ” on page 13.

Using Tool List (*.tlst) Files

When you right-click a tool tile, the context menu offers three items that affect tool list (*.tlst) files. Each (*.tlst) file contains all information for tools in a particular tool list.

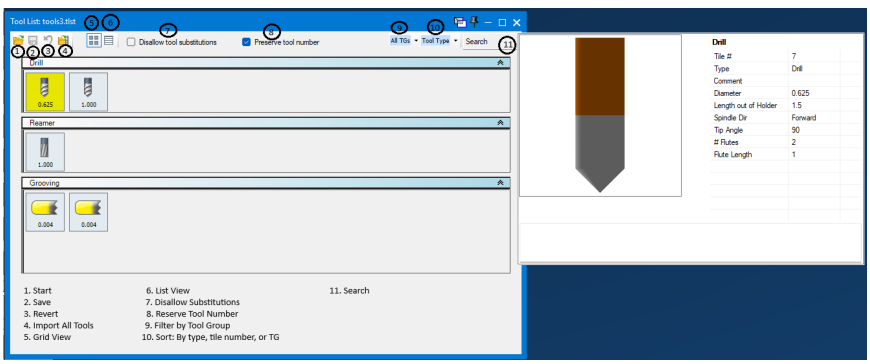
- Save Selected Tools: To create a new *.tlst file from the tool tiles you have selected. To create and modify selections, you can use `Ctrl+A` to select all tiles, and/or `Shift+click` to select a range of tiles, and/or `Ctrl+click` to select/deselect individual tiles. This menu item opens a dialog box that lets you specify the location and name of the *.tlst file.
- Load Tools: To add tool tiles from an existing *.tlst file.
- View/Edit Tool List: Opens a dialog that lets you work with the contents of an existing *.tlst file.



Tile #	Head	Toolgroup	Position	Tool ID	Subposition	Tool Offset #	Nose Diameter	Diameter	Tip Radius	Inner Corner Rad	Corner Rad	Thread Form	TPI	Tip Angle	Lead Angle	Taper Angle	# Flutes	Flute Length	Length out of Holder	Inscribed Dia.	Width	T V
5			1	1	0				0.2												3	
6			1		0			8			0.1						4	32	40			
7			1		0			6									3	10	33			
8			1		0			6.005						90			3	3	30			
9			1		0			8						60			3	2	30			
10			1		0			3.3									3	46	60			
11			1		0			6.3						140			3	46	60			
12			1		0				0.8											12.7		

This dialog can be toggled between spreadsheet form, shown above, and list form, shown below. It lets you:

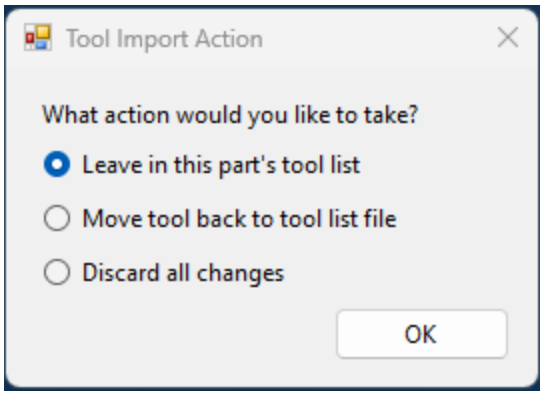
- Hover over a tile in the dialog to show the Tool Preview in the hover window.



- Order tools by tool type (default), by tile number, or by toolgroup.

- Use the search bar to find a tool by text and/or by its most important dimension, optionally filtered by toolgroup.
- Delete a tool from the tool list.
- Drag a tool from the tool list to the part or vice versa.

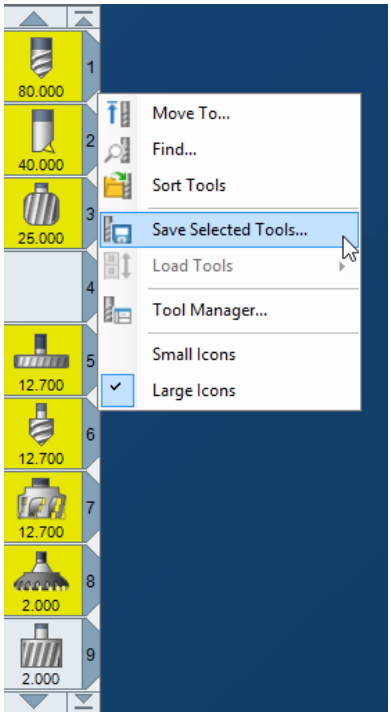
Double-clicking any required tool in the list opens the Tool dialog, allowing tool parameters to be reviewed and updated as required. After completing the modifications and closing the Tool dialog, a confirmation dialog is displayed :



Save or Load Selected Tools

To save a set of tools:
Select the tool tiles, right-click any non-blank tile in the Tool List, and choose Save Selected Tools. *To load a saved tool list file:* Right-click a blank tool tile, choose Load Tools from the context menu, and then navigate to and select the tool list file (`*.tlst`) that you want to load.

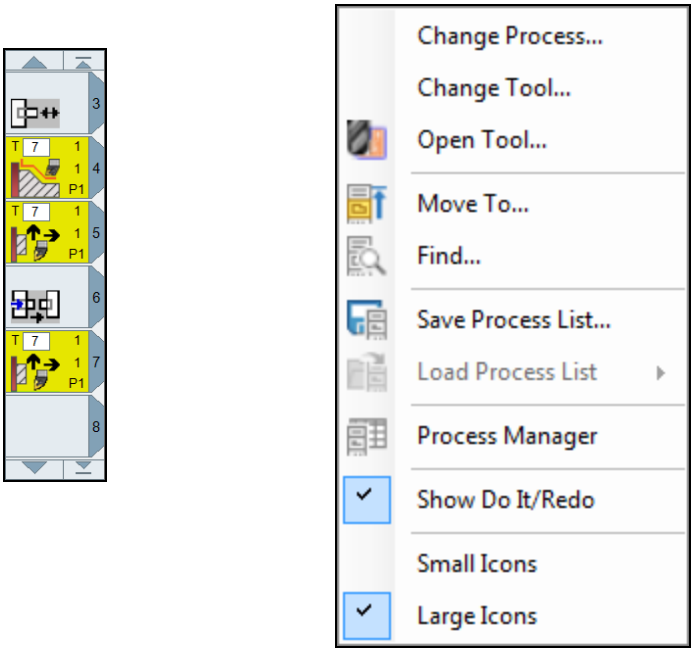
Tool List



To load: **Right-click a blank tile and choose** Load...**.**

To save: **Select one or more tiles, right-click any non-blank tile, and choose** Save...**.**

Process List



The Processes List and its (right-click) context menu

The Process List is used to define toolpath and create operations. A process consists of a single tool from the Tool List and a machining function. When a Process tile, a process is created.

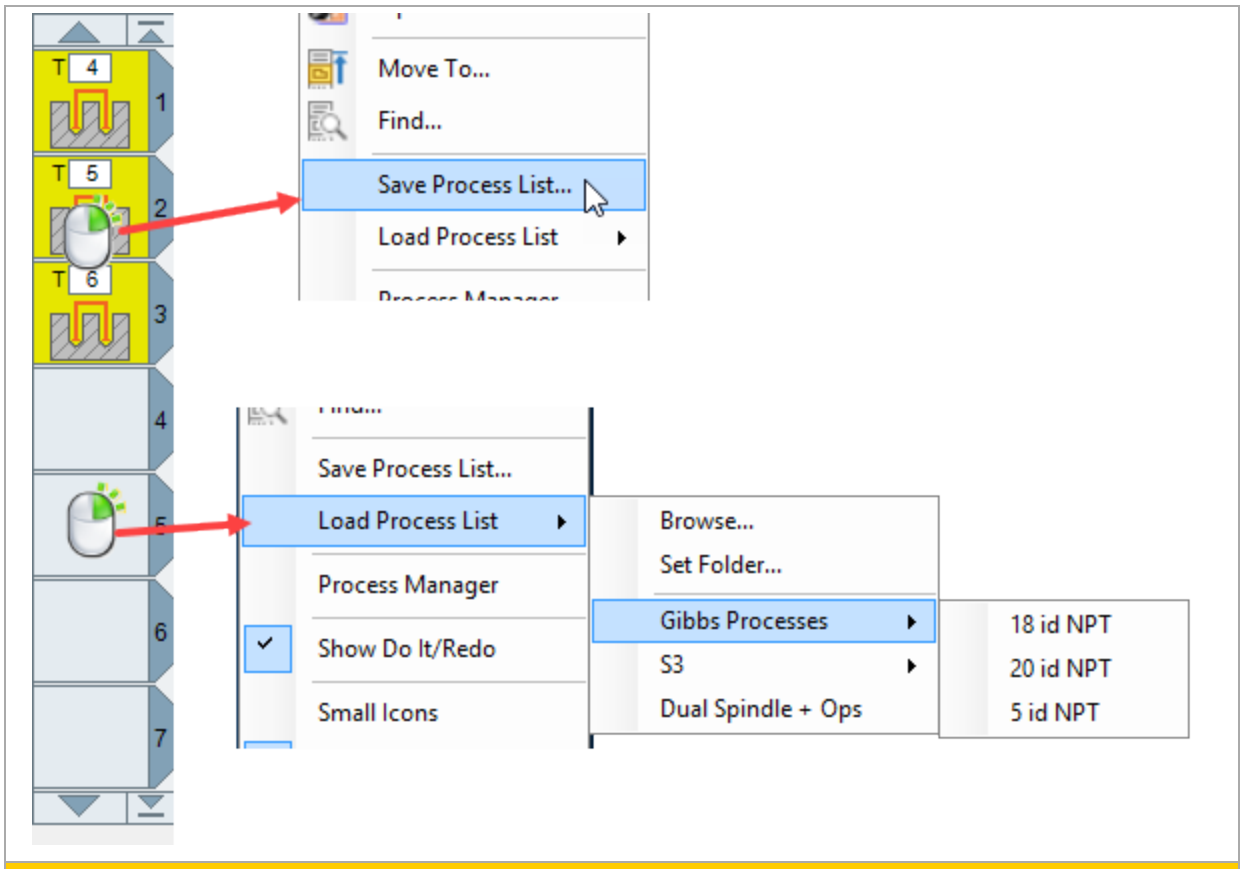
Clearance values, cut depth, speeds and other items are specified in the Process dialog. In most cases, geometry or a solid is then selected to generate toolpath by clicking the Do It button. Once toolpath is generated, one or more operations are created for each process. After operations are created, the processes can be thrown away, because the information from the process is stored in the Operation.

Please note that multiple processes can be created in the same list. This will cause multiple grouped operations to be performed on the same geometry or surface. This function can be used to drill, bore, and tap — all in one set of operations. It can also be used to create a set of operations that drill, rough, and contour the same geometry or solid. Any combination of machining functions can be grouped as needed.

The Process List contains a maximum of 99 Process tiles. The Process tiles do not have to be contiguous; there can be empty spaces between tiles.

Save or Load Process Lists

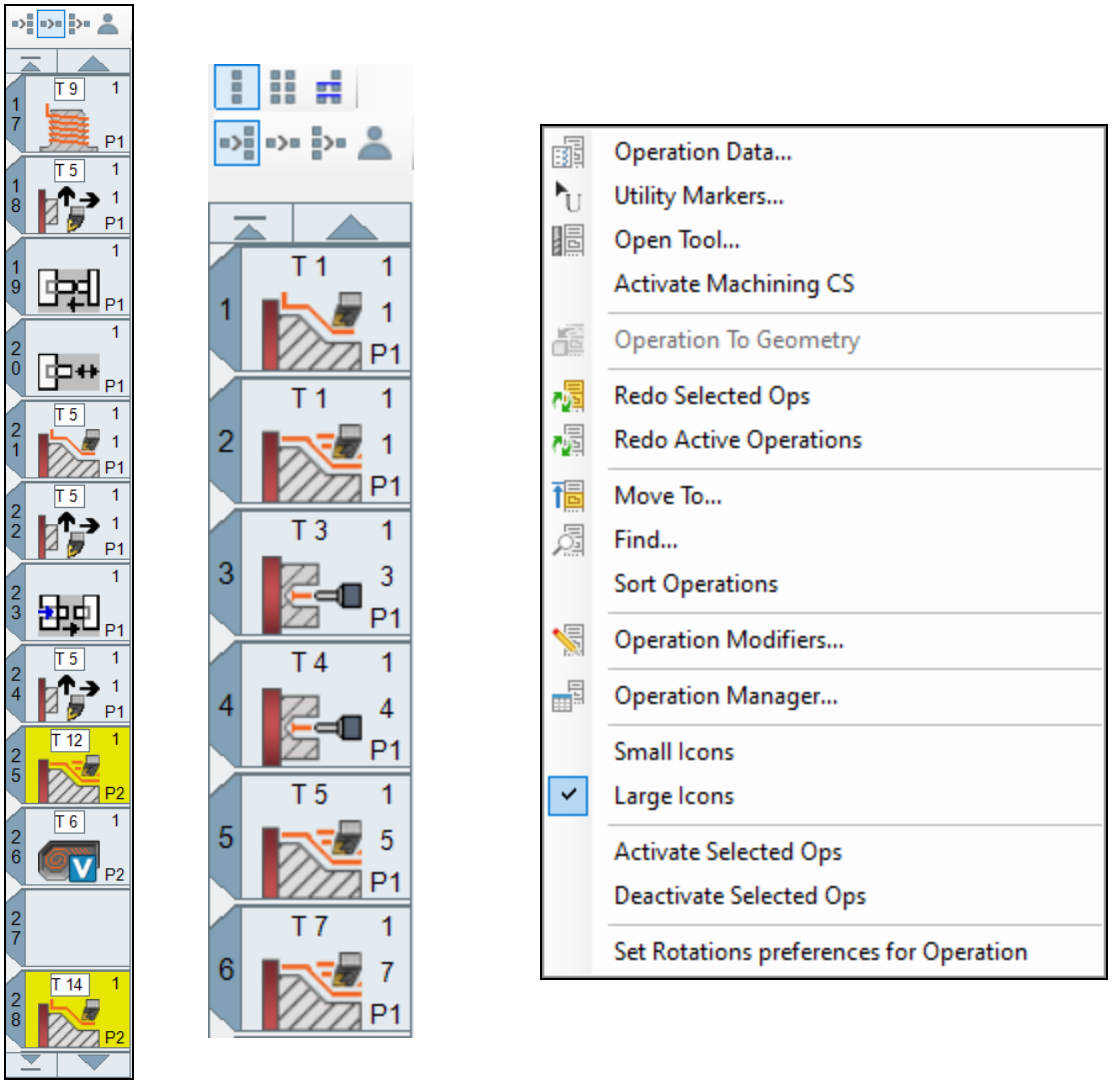
Process List



To save: **With one or more process tiles selected, right-click a non-blank tile and choose** Save Process List...**.**

To load: **Right-click a blank process tile and choose** Load Process List**.**

Operations List



Two examples of the Operations List and its (right-click) context menu

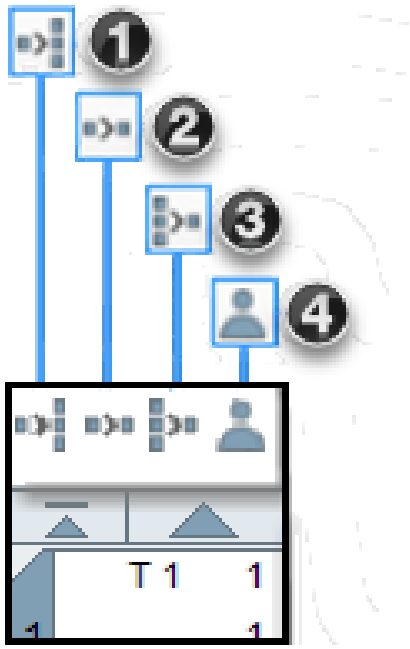
Operations are created from processes and tools. Operations store the toolpath (the cutting motion of the tool) and the information defined by a process. A process generates at least one operation, some processes will result in more than one operation. The software will use the list of operations to generate the final program that is sent to the CNC machine.

A part can have a maximum of 16,000 operations. The Operation tiles do not have to be contiguous; there may be empty spaces between tiles.

The Operations List has a larger context menu than the other lists. The menu contains a number of commands that can be applied to an operation or its toolpath. The normal Find and Move options are also included. For detailed information, see the *Mill* and *Turning* guides.

Operation Tile Stacking

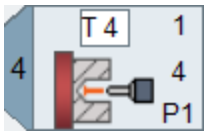
The Operations list offers four choices for organizing Op tiles by stacking them:



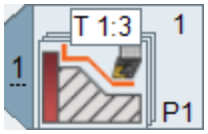
1. Individual: Unstacked.
2. Process Mode: Consecutive operations are stacked together if they originated from a single process and all belong to the same flow.
3. Process List: Consecutive operations are stacked together if they originated from a group of processes to machine the same geometry—and all belong to the same flow.
4. Manual: Allows you to create stacks of consecutive operations that all belong to the same flow. Can include blank tiles and operations on different spindles.

A blue box enclosing a stacking icon indicates the currently selected choice.

Icons that are grayed out (e.g.,) are unavailable; this occurs in Multi-Flow parts when the Operations list is in Sync Flow mode.



Unstacked op tile has no dashed line under the number in the tab.

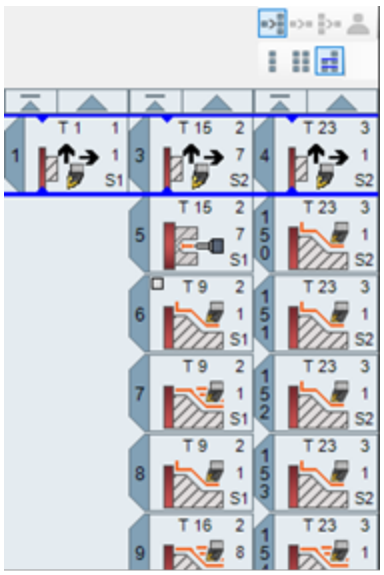


Stacked op tile has a dash-underscored tab number and a visual depiction of multiple stacked items.

Icons for Organizing Operation Tiles in Multi-Flow Parts

For MTM parts that have two or more flows, the Operations list offers additional choices for organizing Op tiles:

Icon	*Name*	*Function*
	Show All Flows	Displays op tiles in a single column.
	Show Each Flow	Multiple columns, one for each flow. Gaps in the column indicate where a flow waits for another flow.
	Sync Flow Mode	Multiple columns, one for each flow. Gaps are suppressed; instead, blue bars indicate syncs between flows.



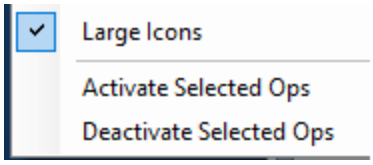
Show Each Flow | Sync Flow Mode

Active and Inactive Ops

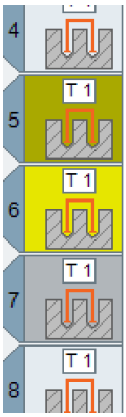
The Inactive Operation feature can be used to work with parts as multiple programs in sequence, to experiment with alternative programming strategies, or for any other purpose where an op needs to be retained in the file but excluded from the program. Active and inactive ops functionality adds a more reliable solution to disclude ops, improving consistency and posted code accuracy.

Op active/ Op inactive status persists when op selection is changed and even when the part is saved and reloaded. You can also view or set active/inactive status using Operation Manager.

To activate or deactivate ops, do one of the following:

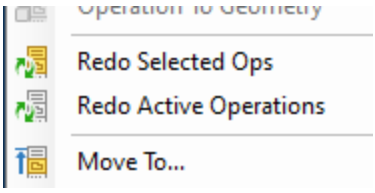


- Right-click an op tile selection. Context-menu choices now include Activate Selected Ops and Deactivate Selected Ops. Only valid ops can be deactivated.
- Use Operation Manager to batch-activate/deactivate operations.
- Use Edit > Operation and click Activate All Ops or Deactivate Unselected Ops.
- Use Plug-Ins > Main; the Find Operations dialog box has a Look In checkbox for Inactive Ops.



Colors. Op tiles for inactive ops have a dark gray background when unselected, dark yellow when selected. Toolpath for inactive ops is drawn in gray rather than orange (when using default colors). Note that inter-operation connection moves are not drawn for inactive ops, since they are not participating in the program.

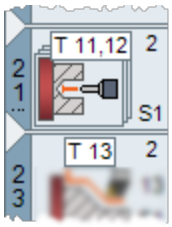
Affected functions. Inactive ops do not participate in any of the following GibbsCAM functions: Post Processing, Simulation, Material State Calculation (for Material Only or Start at Op), Program Error Checking, Sync Control, WFO assignment, Reporter output, and Multipart (TMS and Part Instancing).



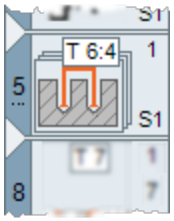
Redo and **Op Modifiers.** If you perform a manual Redo on an inactive op, or if you add an op modifier to it, then its status changes to Activated. However, **batch**Redo actions skip inactive ops by default, and their names have been changed to Redo Active Ops and Redo Selected Active Ops.

Appearance and Behavior of Op Tile Stacks

Tile stacks differ from single op tiles as follows:



- **Stack of frames.** The process icon is framed to suggest a stack of icons.
- **Tile number.** The stack is numbered according to the first op tile in the stack, followed by an ellipsis ... indicating a break in the numeric sequence between it. In the illustration, 21... is above op tile 23; 5... is above op tile 8.
- **Tool numbers.** A single tool number indicates that all ops stack share the same tool. If the ops use exactly two tools, then the first-used is separated from the second-used by a comma(T 11,12 in the illustration). If the ops use three or more tools, the first-used is separated from the last-used by a colon (T 6:4 in the illustration).



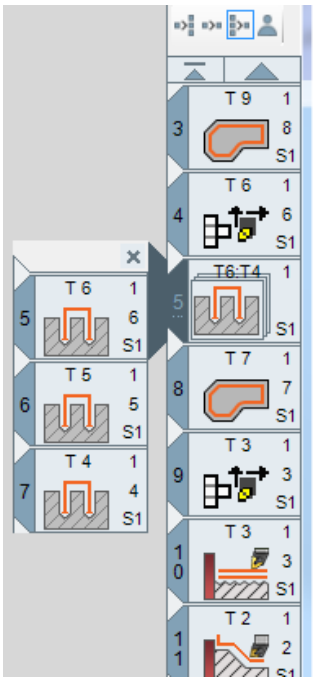
- **Extended tile numbers.** Numbers for toolgroup, tool ID, and spindle number are displayed only if all ops share the same value. In the illustrations, toolgroup and spindle are shared by all ops in each stack; tool ID is not.

Which Process Icon Is Displayed?

If all operations share the same process type, the stack displays that icon.

For Process List, the stack icon indicates the highest-priority process type in the process list. For example, if a process list consists of **Holes**, **Pocket**, and **Contour** , stack displays a **Pocket** icon. When two or more processes of equal priority exist (such as Pocket and VoluMill), the stack displays the icon for the earliest process.

For Manual, the stack icon shows the process type of the earliest operation.



Opening, Closing, and Scrolling a Stack of Op Tiles

Double-clicking a stack opens a flyout that shows a list of op tiles it contains:

To close the flyout: click its close icon or move/resize the GibbsCAM window.

Just as in the main op list, a sufficiently long flyout list provides a scrollbar that lets you navigate up and down.

About Manual Stacks

Choosing Manual op tile stack display creates the following special conditions:

- The context-sensitive (right-mouse-click) menu offers two additional items:
  - Create Stack Allows you to create a manual stack. For Manual only, a stack can consist of one tile.
  - Unstack Undoes the manual stack of the right-clicked item.
- The part "remembers" all manual stacks even when it is in a different mode. Reselecting Manual will restore the previous Manual display state, unless manual stacks have been broken by other op tile additions, moves, or deletions.

Dragging and Dropping

You can drag an entire stack and drop it anywhere in the op list or into another stack. When a stack flyout is open, you can drag tiles into or out of the stack. However:

- Dragging a tile into or out of the stack may cause the stack to break into two stacks if the rules for that stack cannot all be kept.
- Dragging a stack into the flyout of another stack might result in two or three stacks if the rules for the target stack cannot all be kept.
- Dropping a stack onto a stack does nothing.

About Linked Operations (Parent/Child)

In some circumstances, such as using the TpTrans plug-in on a selected operation, one or more operations are created that are linked to the operation that was selected. Each new operation created in this way is called a *child op*, and the original from which a child op is created is called a *parent op*.

A child op can itself become a parent of other ops, each of which is a grandchild of the original op; likewise with great-[great-[...]]grandchildren.

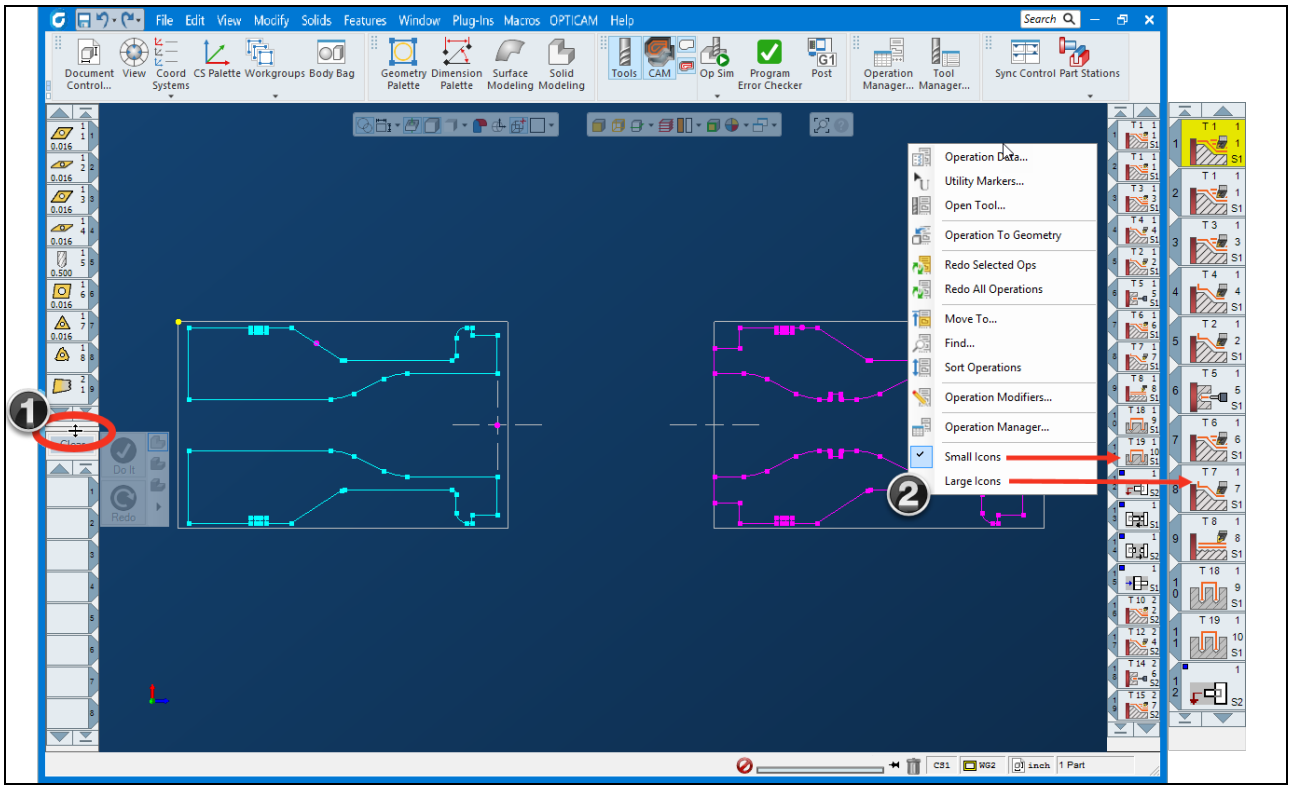
A child op has no process of its own: its processes derive from the process (or process group) of its parent op. Therefore, deleting a parent op deletes any child/grandchild/... ops linked to it, and redoing the parent after changing its process propagates the same change to all its child/grandchild/... ops.

Op Tile Stacking. Since an op shares the same process or processes with all its children, such tile stack views as Process Mode and Process List group the parent together with all its children.

Breaking Links. To break the link from a child op to its parent, use the Break Parent-Child Link plug-in.

Extended Tile Lists

If the GibbsCAM window is sized large enough, additional slots will be visible in the tool and operation lists. You can increase the size of the GibbsCAM window by dragging the lower right hand corner or by clicking the Maximize button to make the window grow to the full size of the screen.

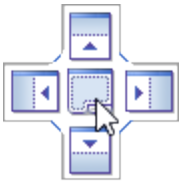


❶ You can view more tool icons by dragging down the bottom margin of the Tool Tile list. Drag it up to see more Process Tiles.

❷To increase the number of tiles on view you can also switch to Small icons.

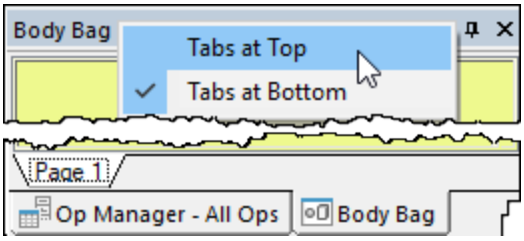
Dockable Panes

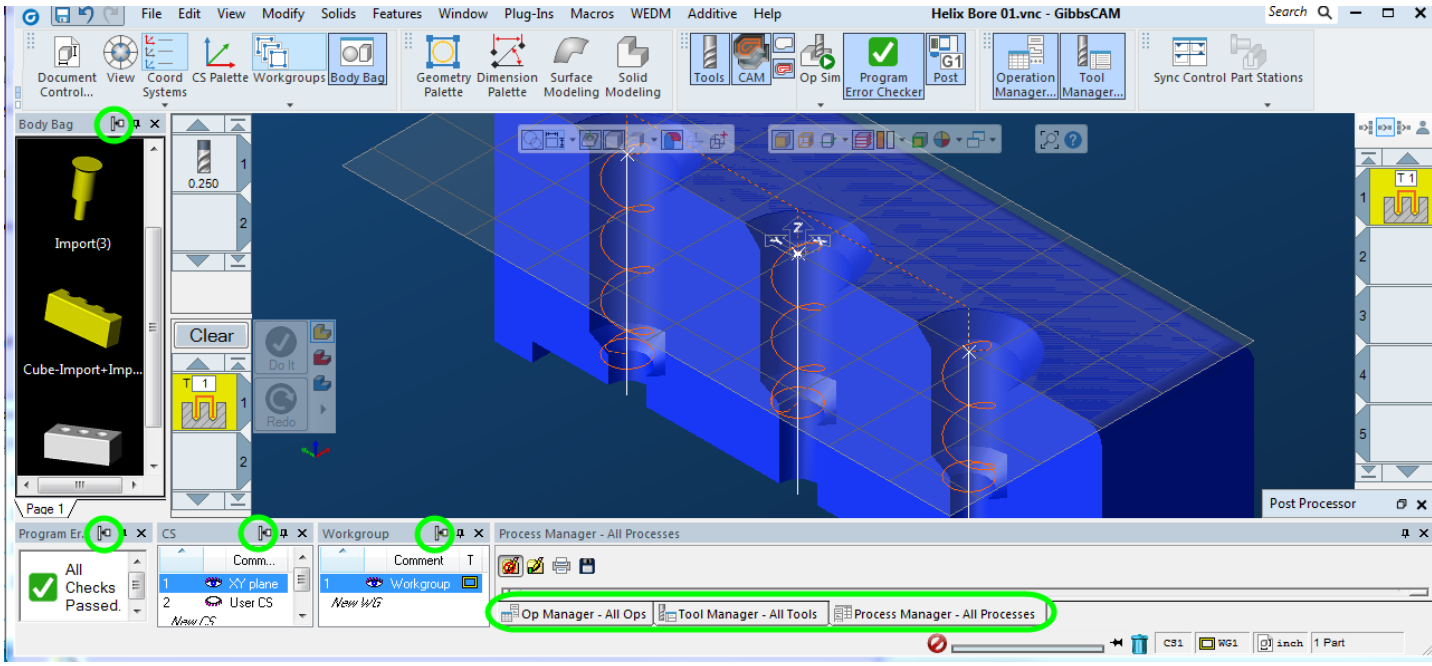
Many dialogs can now be **docked** — that is, attached to the left, bottom, right, or top side of the workspace or to each other (creating a tabbed window). These dialogs include: Body Bag, Program Error Check, CS List, Workgroup List, Operation Manager, Tool Manager, Process Manager, and a few others.



Clicking in a dialog's titlebar activates the Docking Control in the workspace and in the center of each docked pane. Dragging the dialog onto an outward-pointing arrow (up=, left=, right=, down=) docks it to that side, and dragging it onto the center () adds a tab.

Right-clicking the titlebar of a dialog in a tabbed pane lets you set tabs at top or bottom.

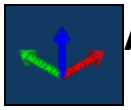




Other Workspace Items

Workspace

The workspace occupies the entire window space. It can be moved and repositioned using the View Control palette or keyboard shortcuts. All drawings of geometry, toolpath, and rendered images appear in the drawing window. The overall size of the workspace is defined by the Document Control dialog settings and represents the default stock condition. To change the size of the window, move the mouse pointer to the outer border and when a double arrow appears, click and drag to the desired size. Use the Viewport manager in the floating toolbar to view in GibbsCAM Multi view mode.

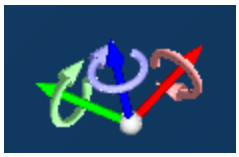


Axis Block

The axis block shown above is, for most users, permanently displayed on the bottom left corner of the screen and indicates the orientation of the part displayed. However, if your computer does not have OpenGL installed, or if OpenGL is disabled, then the axis block is not displayed.

If you want to suppress the axis block display, or display it on another corner of the workspace, go to Preferences: File > Preferences > Display tab, Edit Appearance Settings button > Background tab. See “Appearance Settings Dialog” on page 18.

If you hover over the axis block you will see a view palette that you can use in a similar way to the View palette on the Command Group menu.



Click on geometry and press `Ctrl+Alt` to display a temporary axis block at that location. For more information, see the section on Freehand Actions.

Trash

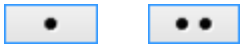
The Trash is a graphical method for deleting on-screen objects such as tiles in a list, geometry or solids. There are two ways to use the Trash. The first method is to click the Trash button which will delete any active and selected item. Multiple elements may be selected, such as one or more tools, operations and geometry. Please note that only the currently active items will be deleted. Active items are usually found in the last selected window. The second method is to drag tiles to the Trash. The last deletion is undoable by selecting Undo from the Edit menu or by pressing `Ctrl+Z`.

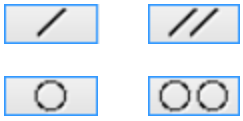
Shortcuts

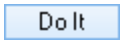
Button Shortcuts

In some cases, buttons may be depressed by actions other than clicking on them. When a button has an outline around it or is highlighted, it can be selected by pressing the `Enter` or `Return` key.

Geometry Dialogs give the user a choice of single or multiple feature creation. To create only one feature (a circle, for example), click the single circle button. To create more than one circle, click the multiple circles button. One of the buttons will always be highlighted. The highlighted button can be depressed by clicking on it, pressing the space bar, the `Enter` key or the `Return` key. Pressing `Shift+Enter` will select the unhighlighted button.





The Do It button is found in some dialogs, primarily those found in the Modify menu. 

It can be depressed by clicking on it or by pressing the `Enter` or `Return` keys.

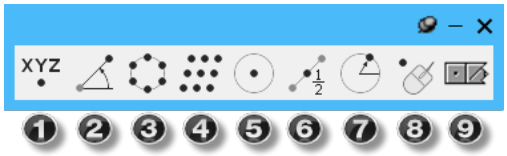
Command Shortcuts

Keyboard shortcuts can be set up for often-used commands. See the section on Customizing the User Interface.

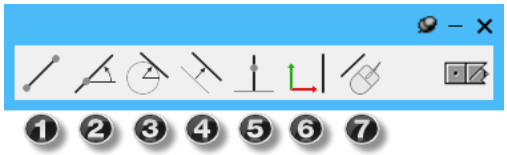
Palette Shortcuts

There are some keyboard shortcuts for functions other than menu choices in the system. Simply hit the appropriate number to activate the button.

Geometry Creation palette



Point sub-palette



Line sub-palette

Circle sub-palette

Auto-Shape sub-palette

Geometry From Solids sub-palette

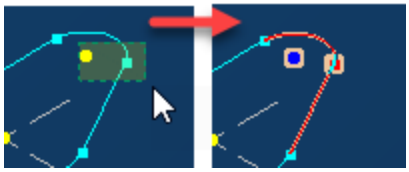
Cursors

The cursor is the screen object moved with the mouse. Its appearance changes depending on its location and will dictate its usage.

Pointer

This cursor is used to select objects and geometry by clicking on them. You can zoom by dragging a rectangle around an area on the screen as shown right.

Area Select



This mode is accessed by holding down the mouse key and dragging the mouse or using the Edit > Select > Mouse Drag command. Geometry with any of its area appearing within the rectangle will be selected in its entirety.

Pointing Finger

This indicates that the cursor is holding an object, such as a tile, that is being moved. It is also seen when moving the workspace using Ctrl-leftmousebutton drag.

Edit/Input

This indicates that the cursor is over a text or value input location. `Clicking` in the text box will produce a flashing text cursor to enter or edit the field.

Interrogation Cursor

This cursor appears when the `Alt` key or `Shift+Alt` keys are held down. The Interrogation cursor is used to automatically enter values from geometry, profiler arcs, circles, and solids into text boxes.

Spot

The cursor changes to this while rolling the trackball in the View palette or while a handle of the pop-up axis block is being dragged for freehand rotation or translation.

Small Pointer

The small pointer appears when the `Ctrl` key is held down. It allows for more than one item to be selected at a time.

Hand Pointer

This appears when selecting commands from sub-palettes

or Resize: Horizontal

This cursor signals that the window or pane can be dragged horizontally to increase or decrease its width.

or Resize: Vertical

This cursor signals that the window or list can be dragged vertically to increase or decrease its height.

Help on item Cursor

This cursor is activated from the Floating Toolbar Help button or when you select the same command from the Help menu. This cursor is active until you click on an item in the GibbsCAM interface.

The Help cursor activates context-sensitive online help for the item you click on. Online Help will open and will attempt to load the section regarding the item you clicked. Some items go to a very specific topic; others will lead to a general section of the help.

Move

This cursor can be found in conjunction with command toolbar groups. Hovering over the drag point (the group of 8 dots in the top left corner of a group) the cursor changes to Move and allows a group to be dragged to another position. Care must be taken with this however a group could become hidden if inadvertently dragged out of the visible area. For more information see Customizing the User Interface.

Eyeball icon

The eyeball icon is used in Workgroup, CS and Hole Manager Group list dialogs. An open eyeball means that the item is displayed in the workspace, a closed eyeball means that it is hidden.

A single mouse click places a box around the eyeball. This is a selection marker box. You can select multiple eyeballs (either one at a time using (`Ctrl-click`) or by specifying a range (`Click, Shift-click`). Double-clicking inside any one of the selected eyeballs will then either deselect or select all items.

Please note that you cannot close the eyeball if a Workgroup or CS row is highlighted in blue, as this is the active Workgroup/CS.

Freehand Actions

Do not confuse actions that scale / pan / rotate the *view* from actions that affect *elements* such as 2D geometry or bodies.

For example, the following actions affect only the current view:

- All mouse actions in the View Control Palette, such as:
  - `Drag` and `right-drag` (cursor) of the trackball axis handles, for constrained rotation or of the trackball itself, for unconstrained rotation.
- `Ctrl-drag`(cursor) to move the workspace.
- `Mousewheel-turn` to zoom and `mousewheel-drag` to rotate the workspace.

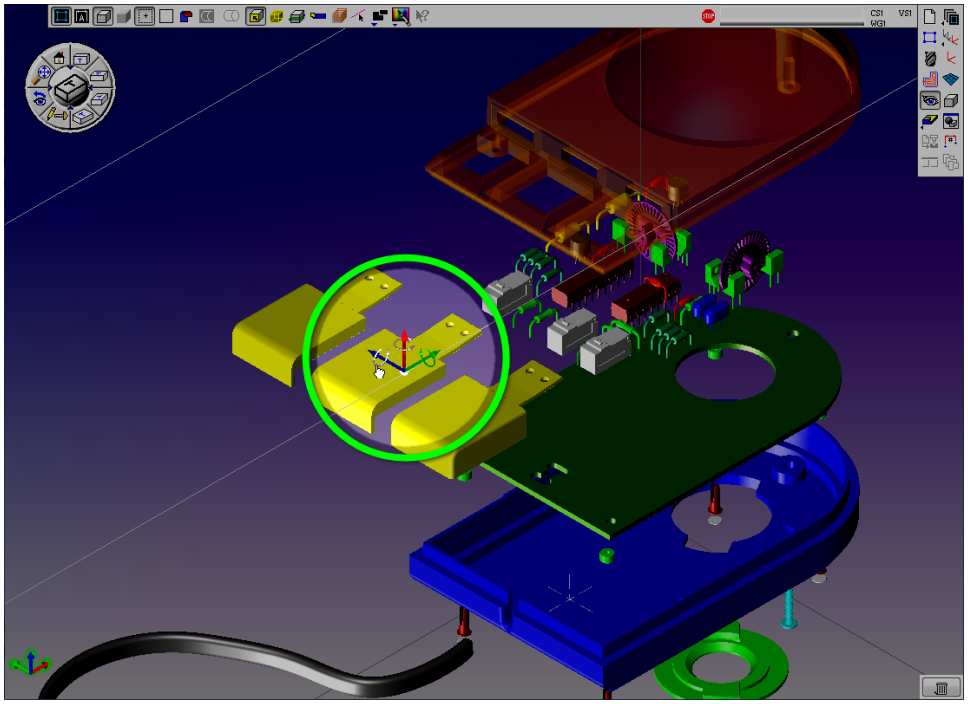
- Hardware controls on 3D motion controllers such as spaceballs
- Gestures on touch screens

In contrast, as you would expect, items on the Modify menu affect elements selected in the workspace and Body Bag, and have no effect on the view.

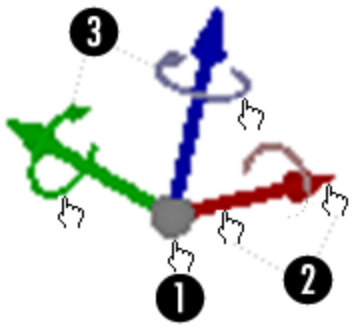
An entirely different class of actions, freehand move and rotate, affects selected elements in the workspace by creating and modifying coordinate systems on the fly.

Freehand Move and Rotate

You can move or rotate elements using the mouse: Select one or more elements, press `Ctrl+Alt` to pop up a temporary axis block, and then drag either the origin (for freehand translation) or one of the six axes (for axis-constrained translation or rotation).



The pop-up axis block remains displayed for as long as the `Ctrl+Alt` keys are held down. If you add or remove bodies in the selection set, the pop-up axis block moves to the center of the bounding box (the minimum box containing all currently selected elements). To zoom in or out on the view, use the `Down` (↓) and `Up` (↑) keys.



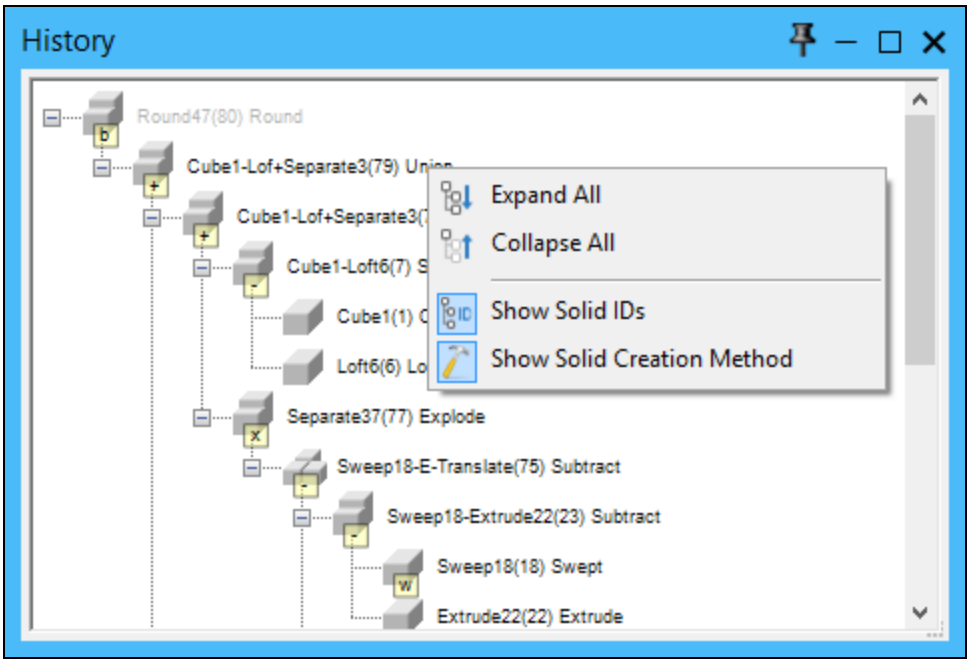
Pop-up axis block showing move (1 or 2) and rotation (3) operations

1. For a freehand move operation of selected elements, click and drag the origin of the pop-up axis block.
2. For a freehand move operation along one axis of the pop-up, click the axis arrow's shaft or arrowhead and drag along the axis.
3. For a freehand rotation operation around one axis of the pop-up, click the spinner around the corresponding axis arrow and drag. The closer to the axis origin, the more extreme the rotation.

When hovered over a handle, the cursor becomes a pointing finger ( ). As the mouse button is clicked and held, the cursor becomes a fist ( ).

When a freehand move or rotation operation is performed, a new or modified CS is associated with each body and profile that has been shifted. The CS is labeled with the comment Freehand CS. Subsequent translations and rotations of the body or profile will update the associated CS.

Right-Click



The mouse has (at least) two buttons that are used in different manners. The left button selects/deselects and accesses items such as geometry, bodies, text boxes, and buttons. The right button gives access to context menus that contain items specific to the selected object or objects. You can `right-click` a title bar of some dialogs to open a menu containing items specific to that dialog.

An example of a context menu is shown above. When you `right-click` the title bar of the History list, its context menu appears. The context menu only contains options that are applicable to the History list.

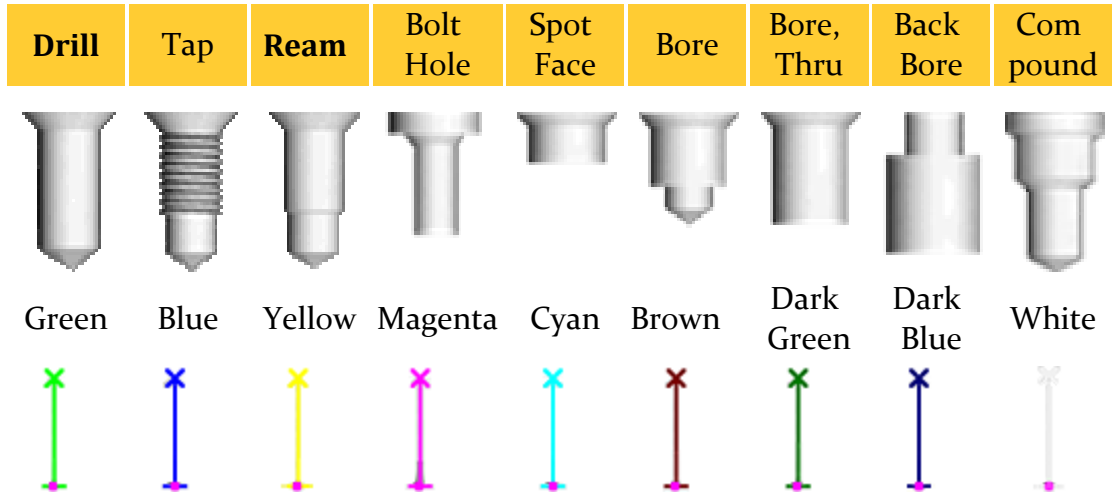
Colors

The system uses colors to graphically display different items drawn on the screen. All system colors may be customized in File > Preferences > Display tab > Colors tab. The default color scheme for geometry, toolpaths, and rendering is listed below.

Item	Color	Meaning
Workspace	White	Wire frame view of the Part Stock and coordinate system markers
	Green	Profile Grid
Geometry	Yellow	Unconnected Geometry or a terminating Point
	Dashed Yellow	Unconnected Rapid Geometry
	Light Blue	Connected Geometry
	Dashed Light Blue	Connected Rapid Geometry
	Red	Geometry designated as “Air” geometry for machining
	Gray	Geometry viewed as a background workgroup
	Magenta	Geometry viewed in a background coordinate system
Toolpaths	Dark Blue	Cut Shape; selected area of a shape to be machined
	Solid Orange	Feed Move in Toolpath
	Dashed Orange	Rapid Move in Toolpath
	Dashed Gray	Rapid Move used for positioning moves and tool changes
	Solid Gray	Approach Feed Move
Solids/Sheets	Gray	Solid Body
	Yellow	Selected Body (solid or sheet)
	Blue	Stock; Selected Profile
	Red	Fixture or negative depth side of a sheet
	Light Blue	Sheet or plane
	Green	Deselected Profiles
Rendering	Blue	Stock
	Yellow	Rendering of selected operations and/or selected tools
	Gray	Rendering of unselected operations and/or unselected tools
	Red	Tool Interference

Hole Manager

The Hole Manager draws an “X” at the top of the hole and a line leading to the drill point (the final depth), for each hole. The color of the line and “X” will vary depending on the hole type.



Math Functions

All numeric input boxes will accept the four standard math function symbols (+ - x /) as well as a number of special functions (either * and x can be used for multiplication). Pressing the `Equal` key or `Tab` key on the keyboard will display the final value. The following list provides all of the math functions recognized by the system.

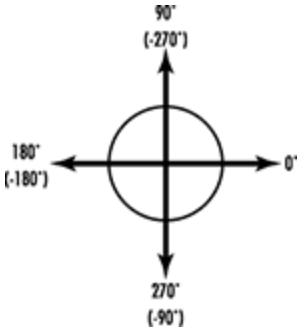
Key	Function	Key	Function
+	addition	–	subtraction
*	multiplication	x	multiplication
/	division	=	equals
m	convert inch to metric (x25.4)	i	convert metric to inch (/25.4)
s	sine	c	cosine
t	tangent	a	arc tangent
r	square root	e	scientific notation (41e-2 = 0.41)

Interrogate

When a text box contains a cursor, you can `Alt+click` a point, virtual point (such a line segment's midpoint, a circle's centerpoint, a solid's vertices, the corner and edge points of cubic stock, and many others), including toolpath, to interrogate its value and copy the value to the text box. In some circumstances, you can `Alt+Shift+click` to copy the value to multiple text boxes.

The contents of any text boxes can be cut, copied and pasted to and from the Clipboard by choosing items from the Edit menu or by using `Ctrl+C`, `Ctrl+X`, or `Ctrl+V`.

Angular values entered in text boxes follow the standard Cartesian coordinate system, shown below. Negative values are acceptable as input.



Appendix

- A Discussion About OpenGL
- Recommendation
- Known Issues and Fixes
- FAQ (Frequently Asked Questions)

About Clearance Volume

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

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

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

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

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

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

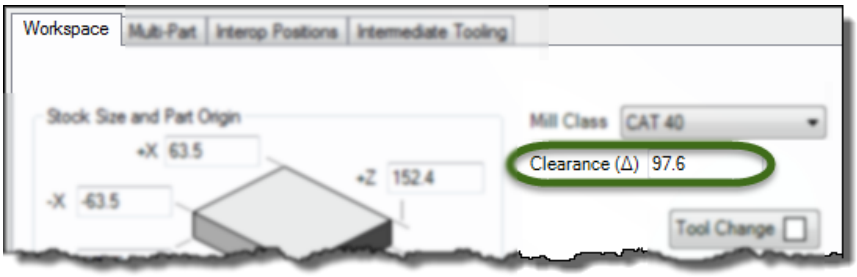
Caveats: Interop moves generated by Clearance Volume contain 5-axis simultaneous moves; thus it is best if the control has TCP capabilities, and it is unsuitable for machines that have indexing rotary axes or rotary axes that must be clamped between moves.

User Interface

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

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

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



Machine Space and Part Space

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

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

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

G-Code

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

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

A Discussion About OpenGL

OpenGL enables several graphic features and provides enhanced rendering of solids. If you have a fast video card with ample memory it can improve rendering performance, because rendering

is done by hardware and software (the driver) on the video card and not by your machine's CPU. Every make and model of video card is different. Performance, display quality and display behavior will vary from card to card. In general, if you have a graphics problem when using OpenGL, you have three options on how to proceed.

1. Use the software driver setting, eliminating the video card hardware and driver use. This will use the generic Microsoft OpenGL implementation.
2. Disable OpenGL, which uses the GibbsCAM display software entirely for rendering. This may be your only option when all else fails. However, the axis block will not display.
3. Leave OpenGL enabled and try a different combination of OpenGL and/or driver settings to see if the display issues can be resolved.

Recommendation

- Modifying some of the Appearance options can adversely affect the display of GibbsCAM. In particular, changing the title bar size and the associated font can make items illegible. We recommend that you not change these items.

Known Issues and Fixes

- When dragging dialogs or windows there can be a "smearing" or after-images of the dialog. This is a normal effect of Windows® settings and OpenGL. If you find this effect undesirable it is easy to fix by turning off the "Show window contents while dragging" option. We recommend this to all users.
  - Right-click on the desktop and select Properties.
  - Select the Appearance tab and click on the Effects button.
  - Deselect the "Show window contents while dragging" option and click OK.
- If your system has an ATI video card moving the display trackball can cause the display to flicker. This is an issue with ATI drivers and the operating system.

FAQ (Frequently Asked Questions)

- **How do I eliminate empty spaces in a list?**

  Using a `Shift+double-click` on the empty space will clear the space.

- **My entire screen just went black and I can't see any geometry. What happened and how do I fix this?**

  In all likelihood, you accidentally zoomed in on the part using a mouse drag. There are three ways to fix this.

1. On the View menu, select Unzoom.

2. On the View Control palette, select Unzoom.

3. Press `Ctrl+U`.

- **Why can't I access posting?**

  Operations must be created before accessing Post dialogs. If you need to get to the Post dialog before creating your part, make a dummy operation (such as drilling a point). This can be done very quickly and can be deleted later.

- **I can't drop my Operation tiles on my Process List. Where are they?**

  Operation tiles can not be moved away from the Operation List. They can be sorted and reordered. To edit an operations process information, double-click the operation to load its process to the Process List.

- **The Chamfer Radius button is not working!**

  Turn on labels (`Ctrl+L`) to identify possible overlapping points. You may also have input an edgebreak value that is too large.

- **When I draw an element, it does not appear**

  Elements drawn outside of the stock boundary described in the Document Control dialog may not always be visible. Try zooming out or increasing stock size. Using the Shrink Wrap command (`Ctrl+‘`) will automatically adjust the window to fit all geometry (including geometry in hidden workgroups or the Body Bag). Shrink Wrap Visible (`Ctrl+]`) adjusts the window to fit visible geometry only.

- GibbsCAM **has become corrupted and will not restart!**

  Installed in the same folder as the GibbsCAM software is an application named Recovery.exe. Instead of having to reinstall GibbsCAM, try running the Recovery.exe application. This should repair most damage done to GibbsCAM. It is also possible that Windows caused a corrupted process that was not fully able to close. Using the Windows Task Manager (Ctrl+Alt+Delete) you can verify that GibbsCAM (virtual.exe) is not still running. If the process is still in the list you may try to End Process and if this fails you may need to reboot your system.

- **Where do I install my hardware key?**

  Keys can be inserted into any USB port in the computer or connected USB hub.

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

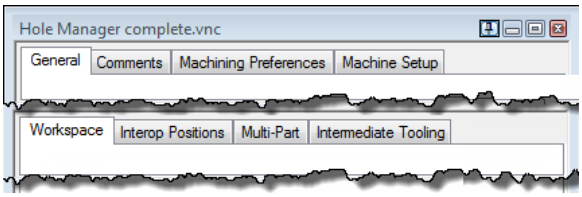
Screen text. Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

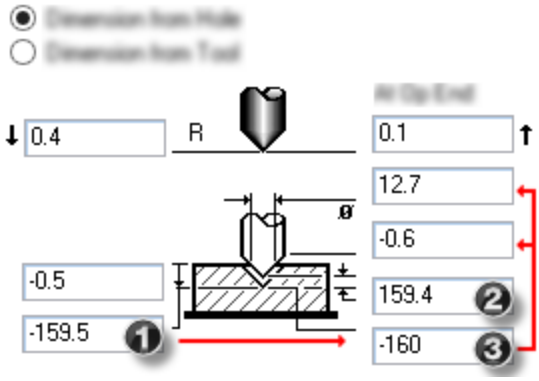
`Keystroke/Mouse`. Text with `this appearance` indicates a keystroke or mouse action, such as `Ctrl+C` or `right-click`.

`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.

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