



Version 14 : September 2020

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## Common Reference



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# Contents

---

<b>INTRODUCTION</b> .....	<b>8</b>
<b>MAIN SCREEN COMPONENTS</b> .....	<b>9</b>
<b>MAIN MENU</b> .....	<b>10</b>
<b>File Menu</b> .....	<b>11</b>
Preferences .....	14
Display Preferences .....	15
Interface Preferences .....	29
Machining Preferences .....	35
File Preferences .....	39
Import/Export Preferences .....	41
Auto Save Preferences .....	42
Coordinate Systems Preferences .....	42
Post Processor Comments .....	44
Communication Setup Preferences .....	47
Intermediate Tooling Preferences .....	48
G-Code Editor Settings .....	49
Additive Preferences .....	51
Materials .....	51
Importing a Material into the Database .....	55
Selecting Material from within a process dialog .....	57
<b>The Edit Menu</b> .....	<b>58</b>
<b>Clipboard</b> .....	<b>59</b>
Select and deselect .....	59
Geometry Expert .....	63
<b>The View Menu</b> .....	<b>65</b>
View Orientation and Control .....	66
Explode/Coordinates .....	67
Visibility .....	67
<b>The Modify Menu</b> .....	<b>69</b>
Transform .....	71
Other Modify Commands .....	74
Wrap/Stock .....	78
<b>The Solids Menu</b> .....	<b>80</b>

---

Tools sub-menu .....	80
Validity Checks .....	80
<b>The Features Menu .....</b>	<b>83</b>
Hole Wizard .....	84
Hole Manager .....	84
Attribute Manager .....	85
Feature Manager .....	85
<b>The Window Menu .....</b>	<b>86</b>
Arrangement .....	86
Managing a Large Number of Processes .....	87
Tool, Process , Operation and Workgroup Managers .....	88
Viewing Data .....	88
Editing Data .....	89
Multi-Selection .....	89
Multiple views .....	90
Manager Context Menus .....	90
Title Bar Context Menu .....	91
Header Customization .....	92
Report Row Context Menu .....	92
<b>The Plug-Ins Menu .....</b>	<b>94</b>
<b>The Wire EDM Menu .....</b>	<b>95</b>
<b>The Macros Menu .....</b>	<b>95</b>
Diamond Insert .....	95
Configuring the Macros Menu .....	96
<b>The Help Menu .....</b>	<b>98</b>
Online Help .....	98
Dynamic Help .....	100
CAMBRIO .....	101
About the Online Help .....	102
Troubleshooting the Online Help System .....	105

---

## **FLOATING TOOLBAR .....**

**107**

---

## **COMMANDS TOOLBAR .....**

**123**

Actions Command Toolbar Group .....	124
Document Control Dialog .....	124
View Control Palette .....	125
CS List .....	127
CS Palette .....	127

Workgroup .....	127
Body Bag .....	128
<b>CAD Command Toolbar .....</b>	<b>128</b>
Geometry Palette .....	128
Dimension Palette .....	129
Surface Modeling .....	129
Solid Modeling .....	129
<b>CAM Command Toolbar .....</b>	<b>130</b>
Tools .....	130
CAM .....	130
Tile Lists .....	131
Cut Part Rendering and Simulation .....	131
Program Error Checker .....	131
<b>Post Processor dialog .....</b>	<b>131</b>
Options tab .....	133
Workfixtures tab .....	134
<b>Summaries Command Toolbar .....</b>	<b>135</b>
<b>MTM Command Toolbar .....</b>	<b>135</b>
Sync Control .....	135
Part Stations .....	136
<b>Cut Part Rendering and Simulation .....</b>	<b>137</b>
Cut Part Rendering and Simulation Options .....	138
Playback Controls .....	139
Record Video .....	140
CPR/Rendering Visibility Controls .....	141
Simulation Control Icons .....	144
Simulation Context Menus .....	151
Legacy CPR Context Menu .....	152
All Context Menus .....	152
Printing the Rendered Image .....	156

---

## **STATUS BAR .....**

**157**

---

## **CUSTOMIZING THE USER INTERFACE**

**159**

<b>Command Shortcuts in Trackball .....</b>	<b>159</b>
<b>Customizing the Workspace .....</b>	<b>160</b>
Customizing Toolbars and Menus .....	161
Save/Load Customization Profile .....	162
New Menu Button .....	162
Keyboard Shortcut List .....	163
Commands tab .....	164
Menus tab .....	166

Rename/Delete/Reset Buttons .....	166
Toolbars Tab .....	166
Positioning Command Toolbar Groups .....	167
Exit .....	167
Moving Command Groups around the workspace .....	167

---

## MISCELLANEOUS ..... 169

<b>Lists .....</b>	<b>169</b>
Tool List .....	170
View/Edit Tool List .....	171
Save or Load Selected Tools .....	172
Process List .....	173
Save or Load Process Lists .....	173
Operations List .....	175
Appearance and Behavior of Op Tile Stacks .....	176
Opening, Closing, and Scrolling a Stack of Op Tiles .....	177
About Manual Stacks .....	177
Dragging and Dropping .....	177
Extended Tile Lists .....	177
<b>Dockable Panes .....</b>	<b>178</b>
<b>Other Workspace Items .....</b>	<b>179</b>
Workspace .....	179
Axis Block .....	179
Trash .....	180
<b>Shortcuts .....</b>	<b>180</b>
<b>Cursors .....</b>	<b>181</b>
<b>Eyeball icon .....</b>	<b>183</b>
<b>Freehand Actions .....</b>	<b>183</b>
Freehand Move and Rotate .....	184
Right-Click .....	185
<b>Colors .....</b>	<b>185</b>
Hole Manager .....	186
Math Functions .....	187
Interrogate .....	187

---

## APPENDIX ..... 189

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About Clearance Volume .....	189
A Discussion About OpenGL .....	190
Recommendation .....	191
Known Issues and Fixes .....	191
FAQ (Frequently Asked Questions) .....	191

---

<b>CONVENTIONS</b> .....	<b>193</b>
Text .....	193
Graphics .....	193

---

<b>LINKS TO ONLINE RESOURCES</b> .....	<b>194</b>
--	------------

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<b>INDEX</b> .....	<b>195</b>
--------------------	------------

# Introduction

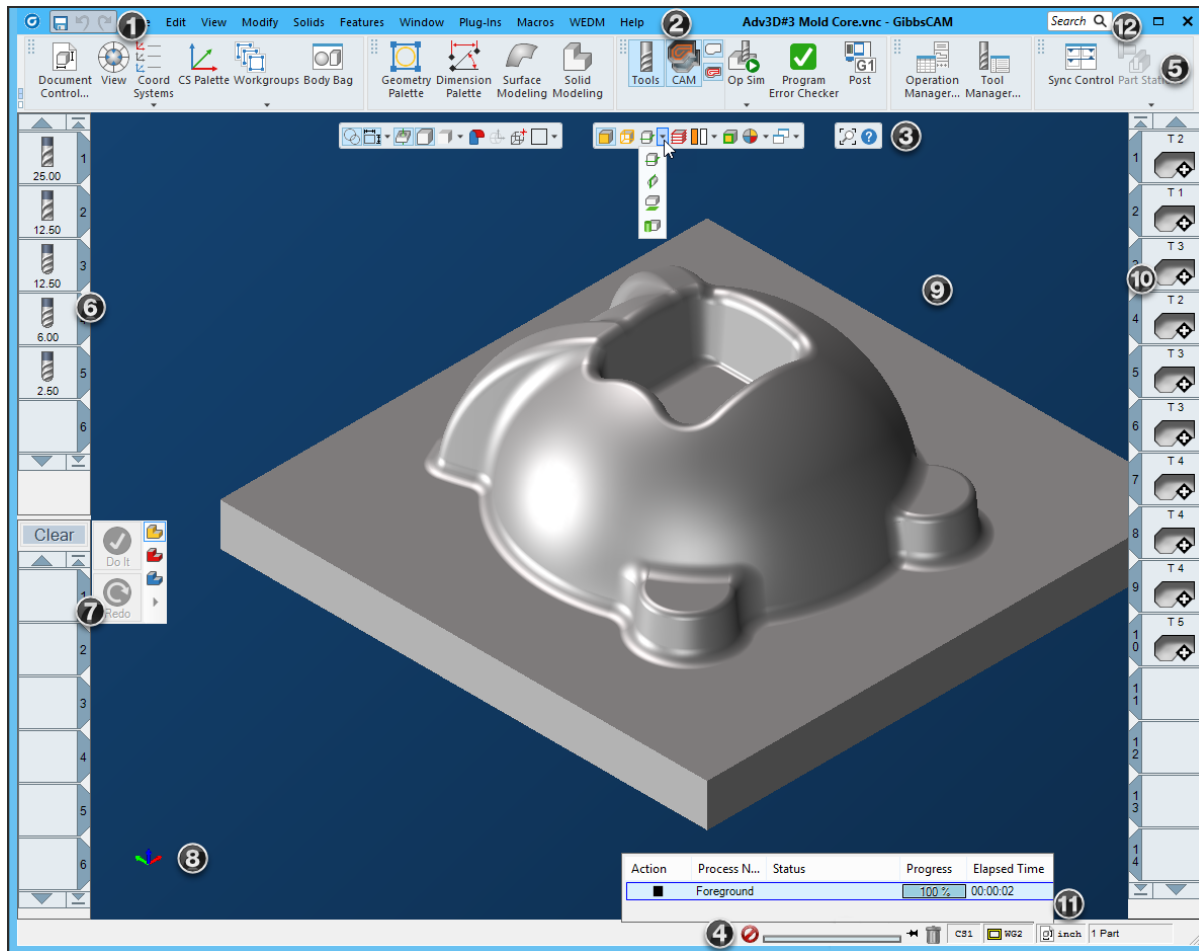
Welcome to the GibbsCAM [Common Reference](#) guide. GibbsCAM is a modular application, meaning that different customers will have different functions depending on their need. In other words, not everybody needs Mill/Turn functionality or the SolidSurfacer module.

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.



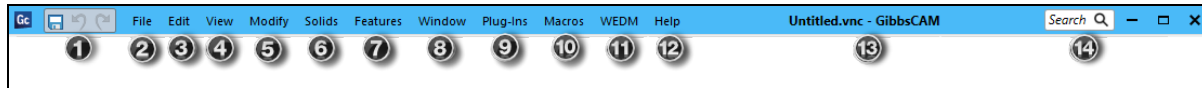
# Main Screen Components



- |                                     |                                     |                                 |                                     |
|-------------------------------------|-------------------------------------|---------------------------------|-------------------------------------|
| 1. <a href="#">Quick Launch Bar</a> | 4. <a href="#">Status Bar</a>       | 7. <a href="#">Process List</a> | 10. <a href="#">Operations List</a> |
| 2. <a href="#">Main Menu</a>        | 5. <a href="#">Commands toolbar</a> | 8. <a href="#">Axis Block</a>   | 11. <a href="#">Trash</a>           |
| 3. <a href="#">Floating Toolbar</a> | 6. <a href="#">Tool List</a>        | 9. <a href="#">Workspace</a>    | 12. <a href="#">Command Search</a>  |

# Main Menu

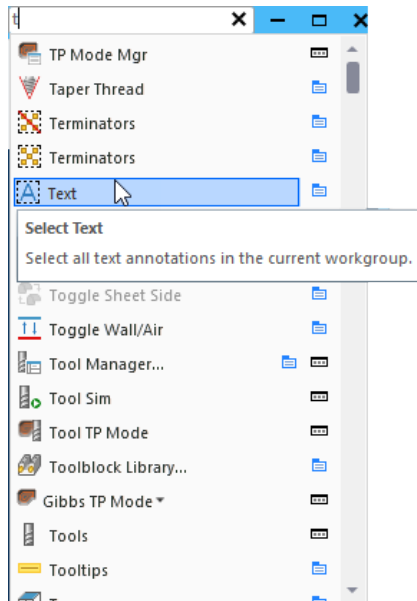
The Main Menu is available to the user at all times. The Menu provides access to many of the system's functions through individual sub-menus. Some items may or may not appear depending on the installed options.





1. Quick Access Toolbar  
This can be customized and extended if required. See [Customizing the UI.htm](#)
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. The Wire EDM Menu
12. The Help Menu
13. Current filename display
14. Command search.

## Command Search

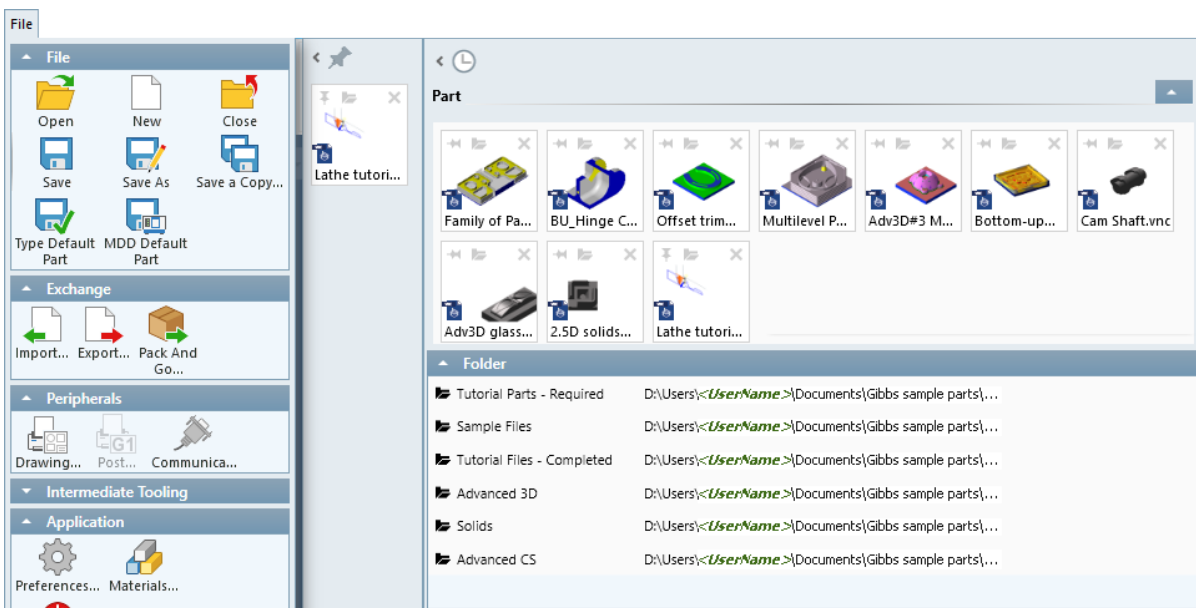
Command Search lets you find a command just by typing in a portion of its name.



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. Selecting geometry or some other appropriate item on the screen may 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 – the original file will remain open.

Previous system versions are available to save the type to an older version. Please note that saving a file to an older version may cause the part to lose capabilities or functions not available in the older

version. In particular, for parts with toolpath created using Advanced 3D, 5-Axis, VoluMill, or VoluTurn, we recommend against saving parts backwards to GibbsCAM 12 or earlier.

### 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\CAMBRIO\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 "**Printing**" on page 24.



### 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 48. 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 14.



### 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 51.



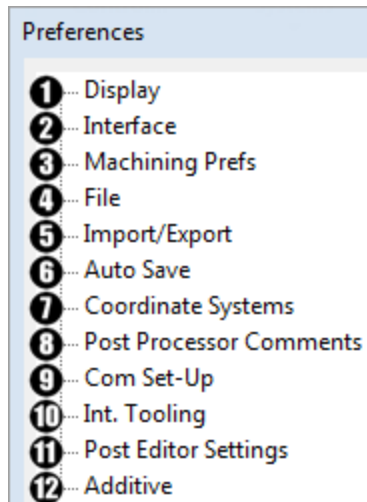
### Exit

This closes GibbsCAM.



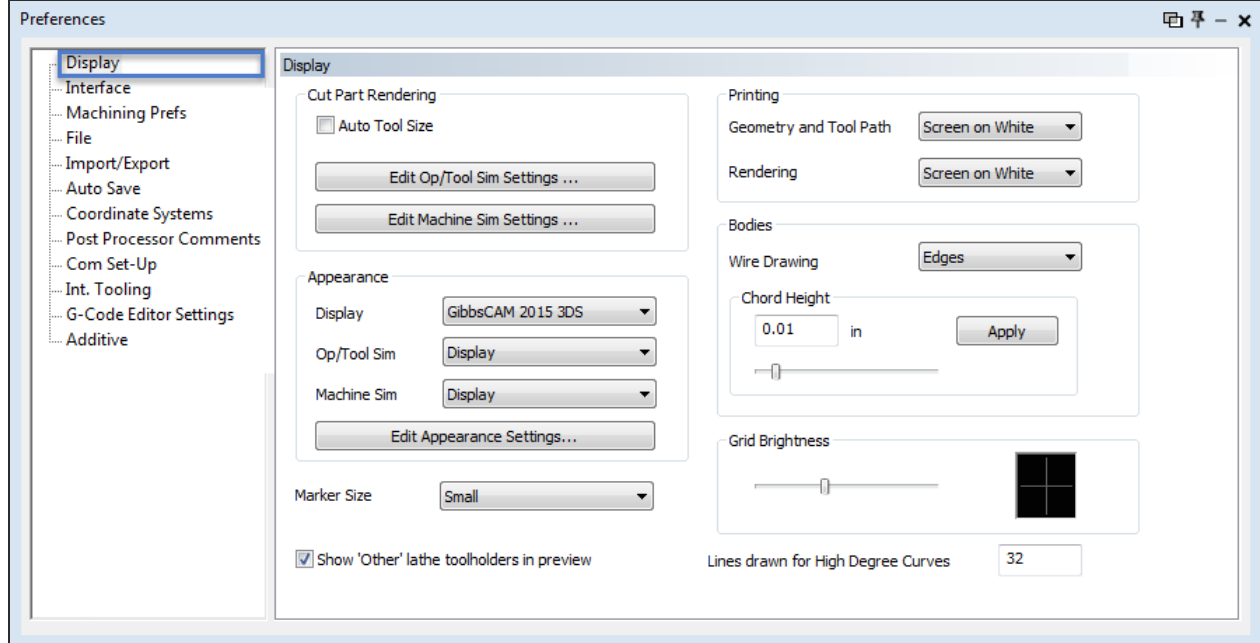
## Preferences

The Preferences dialog presents tabs containing controls that affect different parts of the system.



1. [Display Preferences](#) , next
2. “[Interface Preferences](#)” on page 29
3. “[Machining Preferences](#)” on page 35
4. “[File Preferences](#)” on page 39
5. “[Import/Export Preferences](#)” on page 41
6. “[Auto Save Preferences](#)” on page 42
7. [Coordinate Systems Preferences](#)
8. “[Post Processor Comments](#)” on page 44
9. “[Communication Setup Preferences](#)” on page 47
10. “[Intermediate Tooling Preferences](#)” on page 48
11. “[G-Code Editor Settings](#)” on page 49
12. “[Additive Preferences](#)” on page 51

## Display Preferences



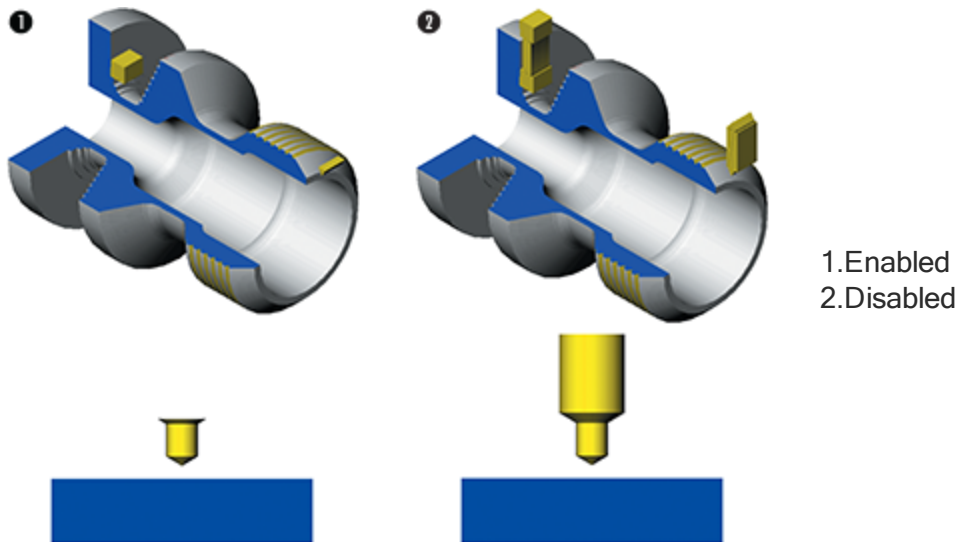
The Display preferences affect what you see on screen or what you print out.

### 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.



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 189](#) 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

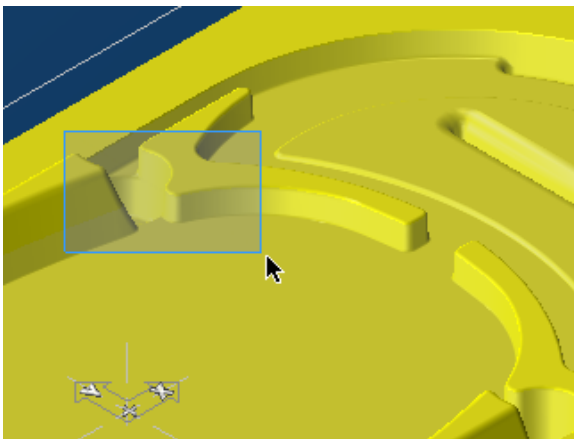
When this option is active, you can bring up the Appearance preferences dialog by pressing and holding the right mouse button when the mouse is positioned over an empty area of workspace – in other words, not over solids, toolpath, or geometry.

#### Unzoom on View Change

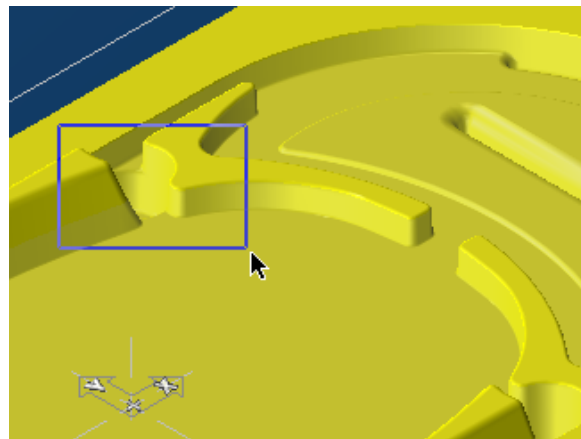
When this item is active, the system will automatically unzoom the part when you change views using any of the trackball buttons or keyboard shortcuts for the standard views. The system will not unzoom if you perform a pan or rotate.

#### 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.

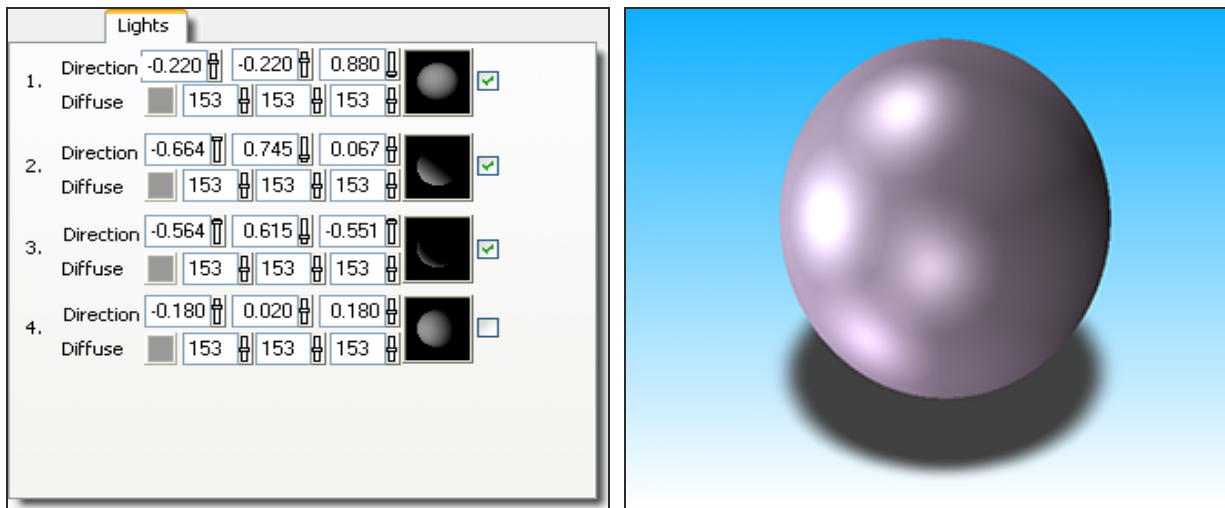
### Plane Opacity

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.

### 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 through 2551. 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 button and select a light color from the color picker. The color picker is described in “Colors ” on page 23.

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 and transparency of solids, surfaces, and many other items. The color picker is described in “Colors ” on page 23.

### 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.

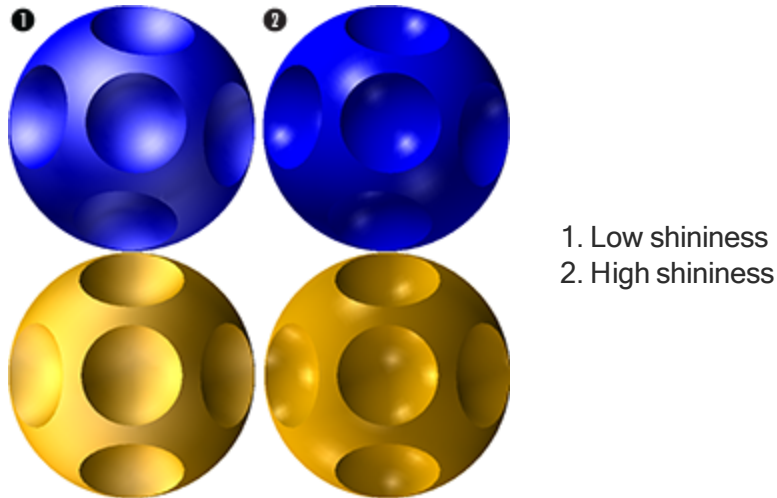
#### Ambient

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.



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.

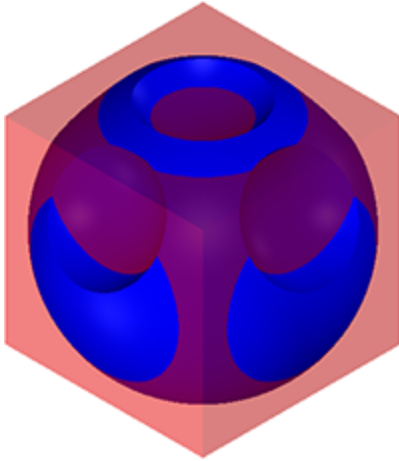
**Halo Transparency** Sets the transparency of the pre-selection. A value of less than 20 will be very subtle, whereas the maximum value, 255, will make the preselection opaque.

**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 CPR. You can change the default color of rendered objects by setting the RGB values or by clicking on the appropriate color button and selecting a color. Separate colors can be applied to the Stock, selected bodies (Target), Fixtures, toolpath (Op), interference (Clash), the primary color of the cutting tool (Tool), the Shank of the cutting tool, and the color of wireframe drawing (Edge). The color picker is described in “Colors” on page 23.

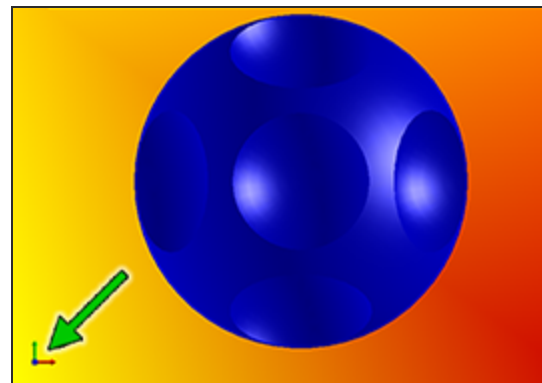
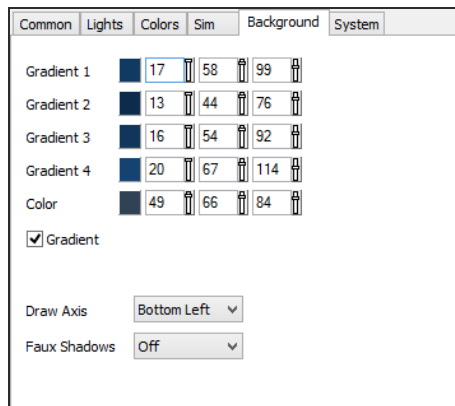
You can set the transparency level for the stock, fixtures and tools. A value of 0 turns the transparency off (the item is solid); the maximum value of 255 makes the item completely transparent. Whether the stock and/or fixture is transparent is controlled by the right-mouse menu of the Render Control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

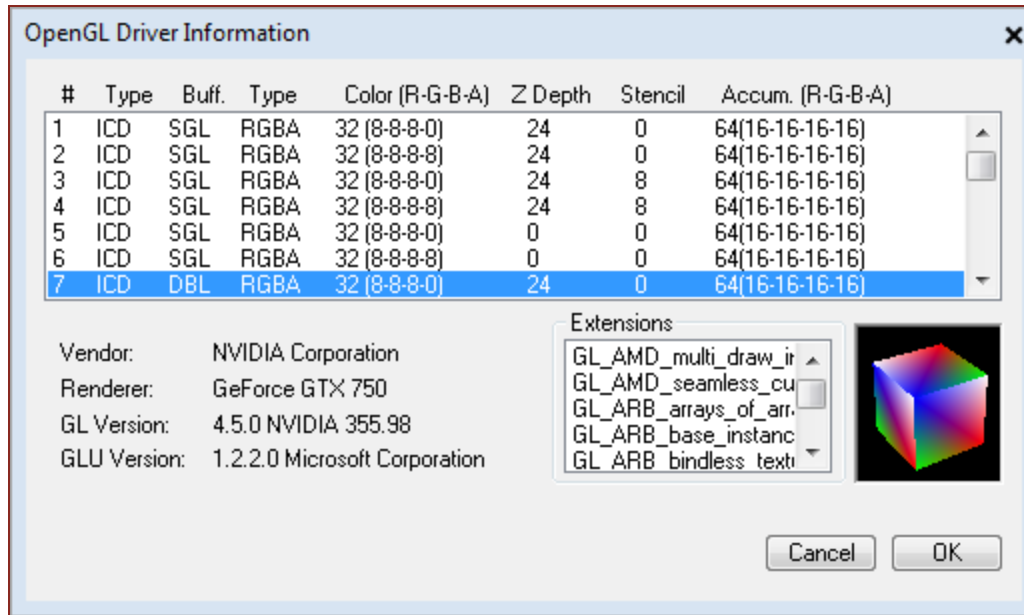
**P-Buffer**

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

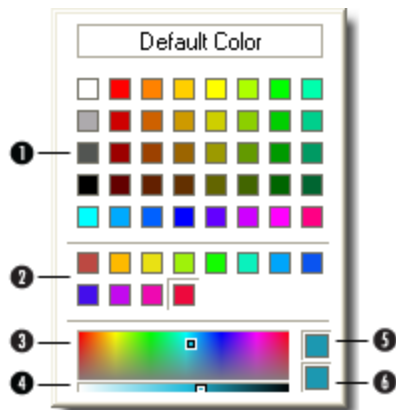


This item opens a dialog that displays the currently available OpenGL drivers on the machine, their supported extensions, and the amount of currently used and available graphics memory.

## Test Frame Rate

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

## Colors



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

The Rendering Preferences 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.

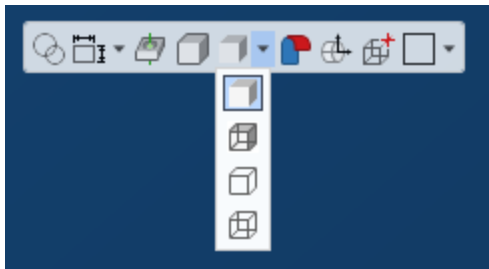
## Printing

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.

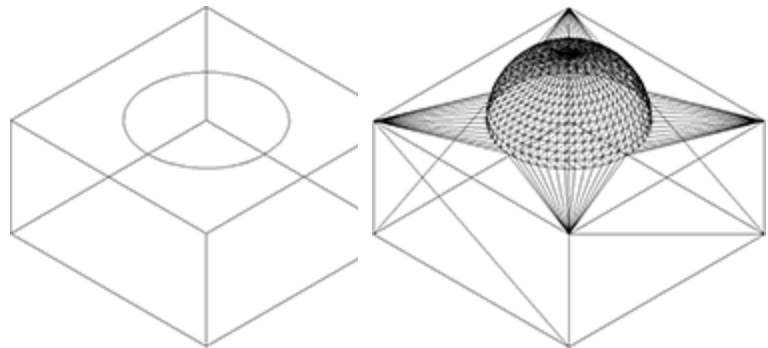
Separate options are available for printing the geometry and toolpath and for the rendered part. Selecting **Screen** will print the drawing exactly as it appears on the screen, with a black background and the colors as they appear on the screen. Selecting **Screen on White** will print the image in 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

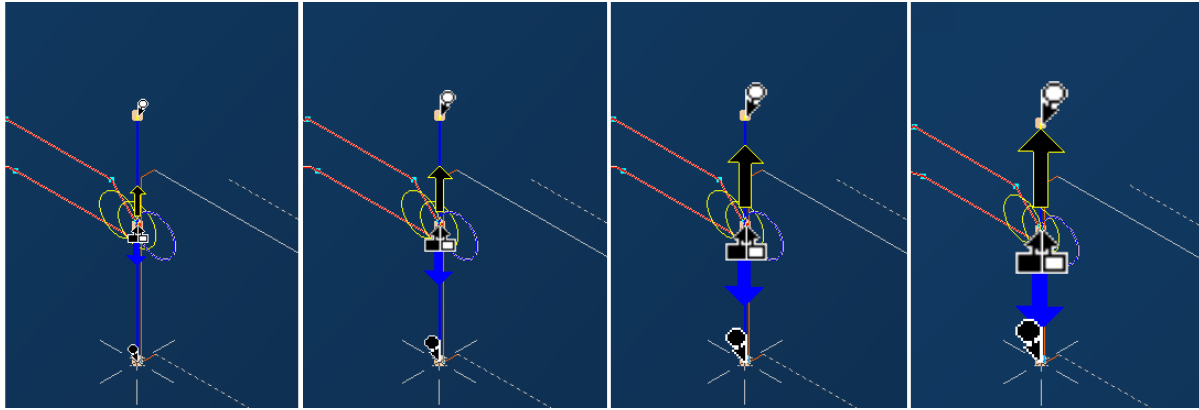
The **Grid Brightness** option is for the Level 2 interface only. This slider controls the brightness of the coordinate system grid in the stock diagram. The brightness can be adjusted by moving the slider.



## Other Items

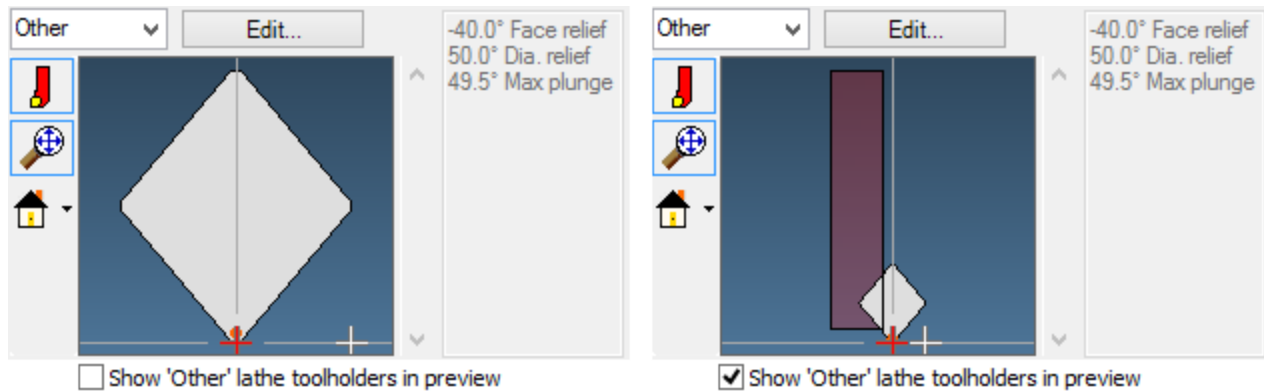
### Marker size

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



### Show 'Other' turning toolholders in preview

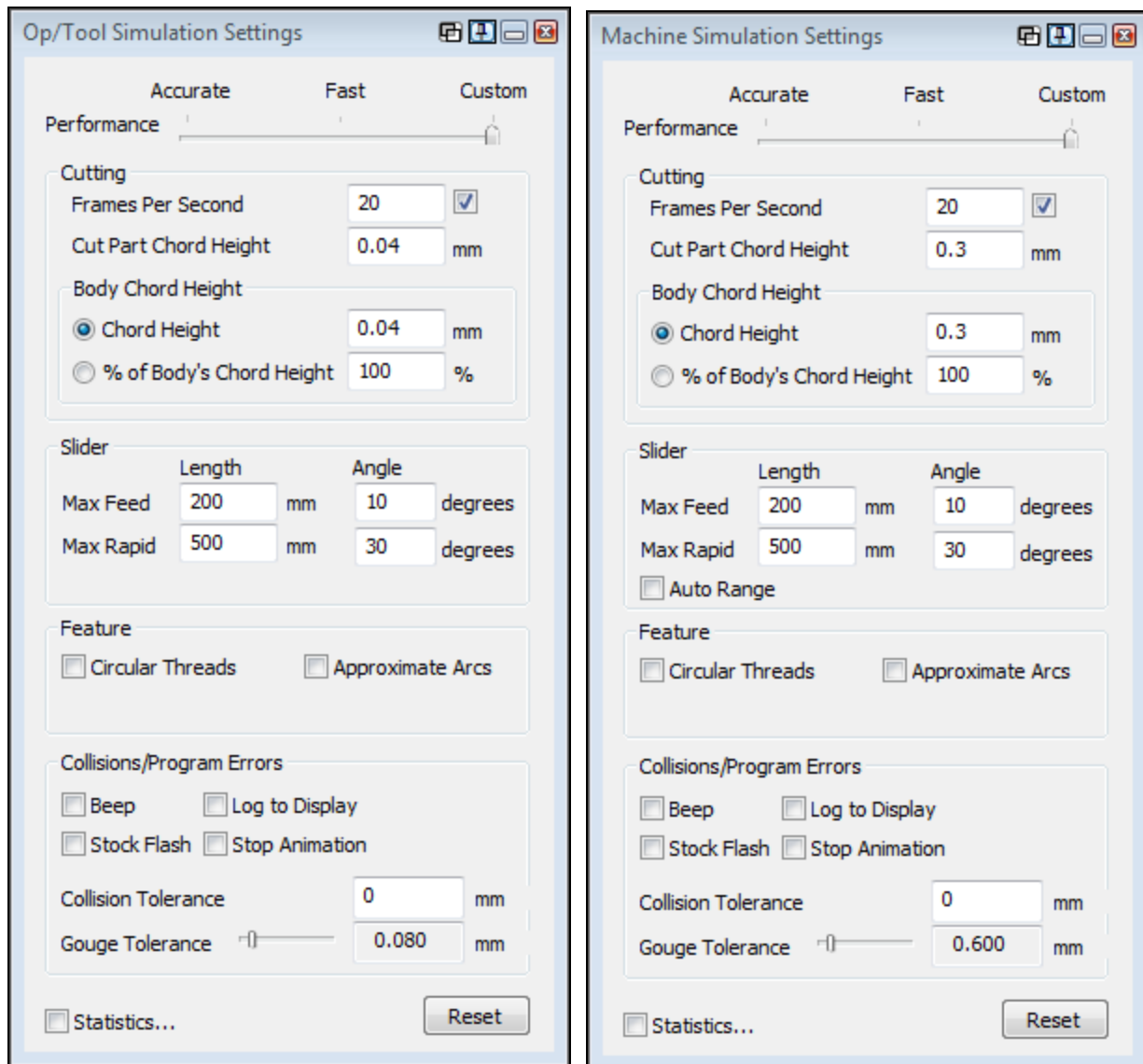
For Turning inserts when no toolholder has been specified, select this checkbox to display a "pseudoholder" (rectangle drawing) in the tool preview window.



### 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.

### Performance

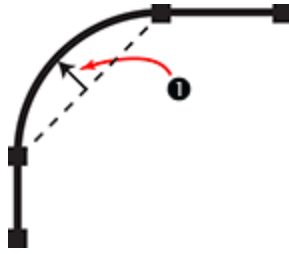
Drag the performance slider to choose predefined settings to set greater accuracy or greater speed, or to choose custom settings.

### Cutting

#### Frames per Second

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

### Cut Part Chord Height



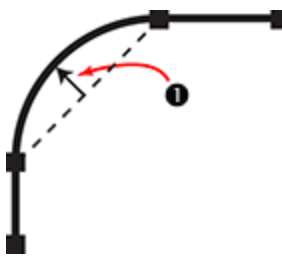
1. Chord Height

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

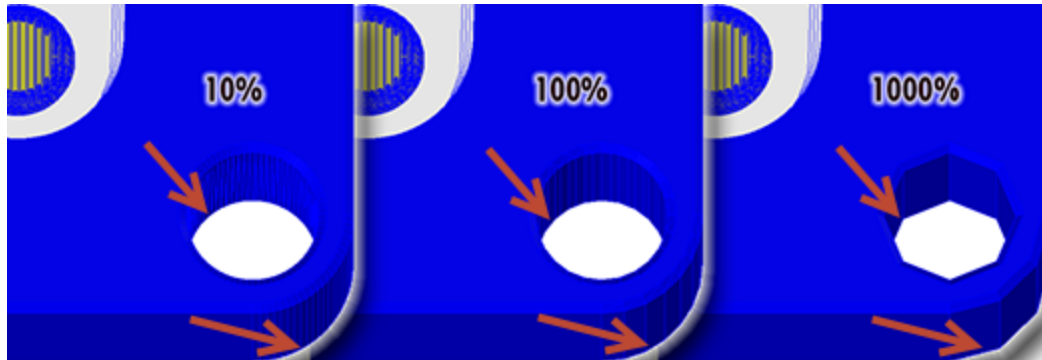
Determines the resolution of bodies (part, stock, and fixtures) in rendering.

#### 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

#### Circular Threads

This options renders "circular" threads rather than proper spiraled threads. Enabling this option will render threads much more quickly.

### Approximate Arcs

This option renders arc features as a series of lines. Approximated circles may provide greater rendering stability.

### 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.

### Collision Tolerance

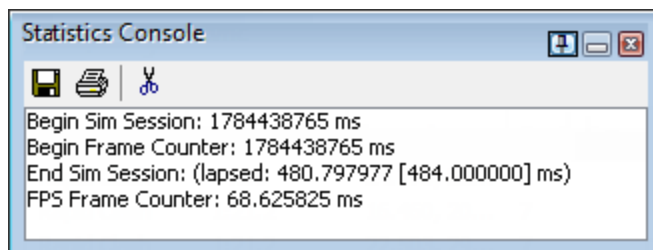
The Collision Tolerance setting allows a different value for parts in metric and inches. Any collision within the specified tolerance generates a collision alert.

### 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

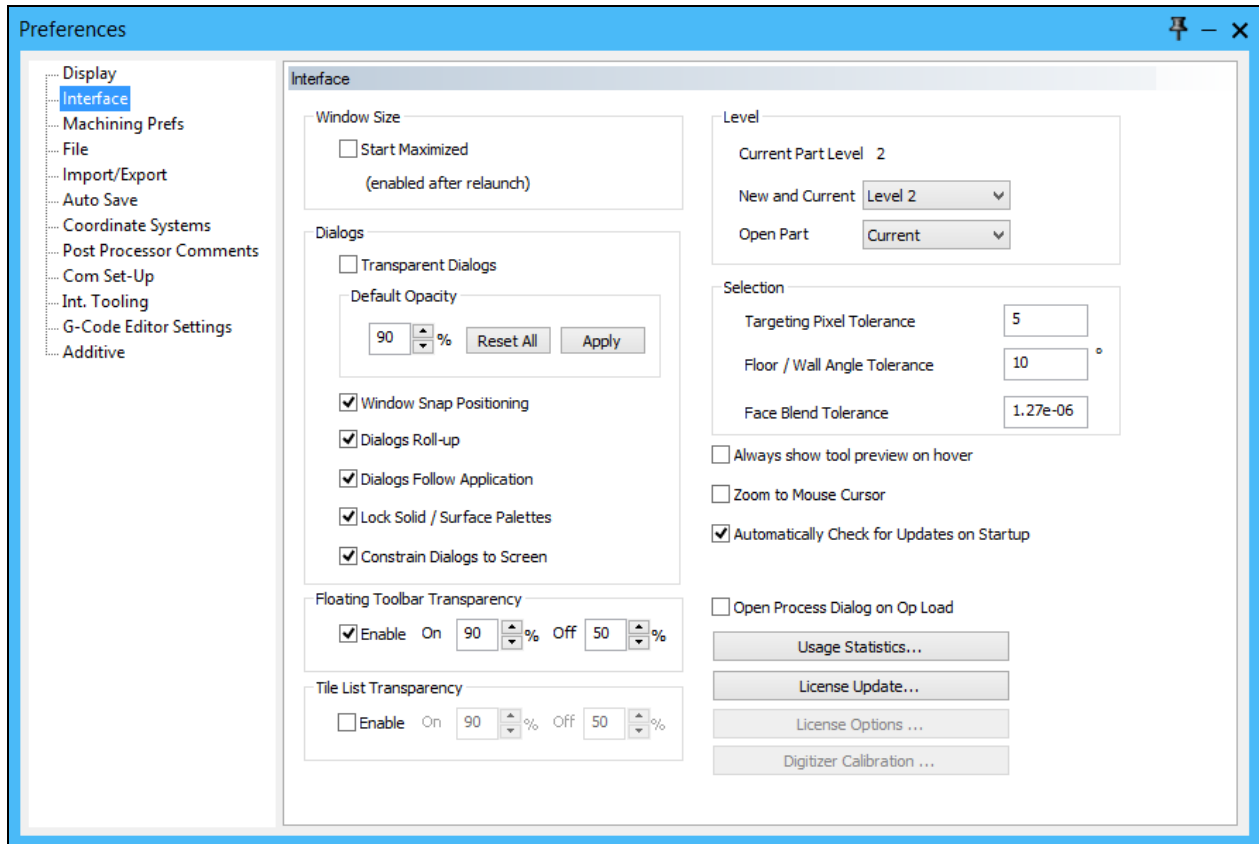
When the Statistics option is enabled, a window will open when you activate Simulation. The window logs the current frame-rate for your machine as well as any logged errors for the CPR session.



### 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.

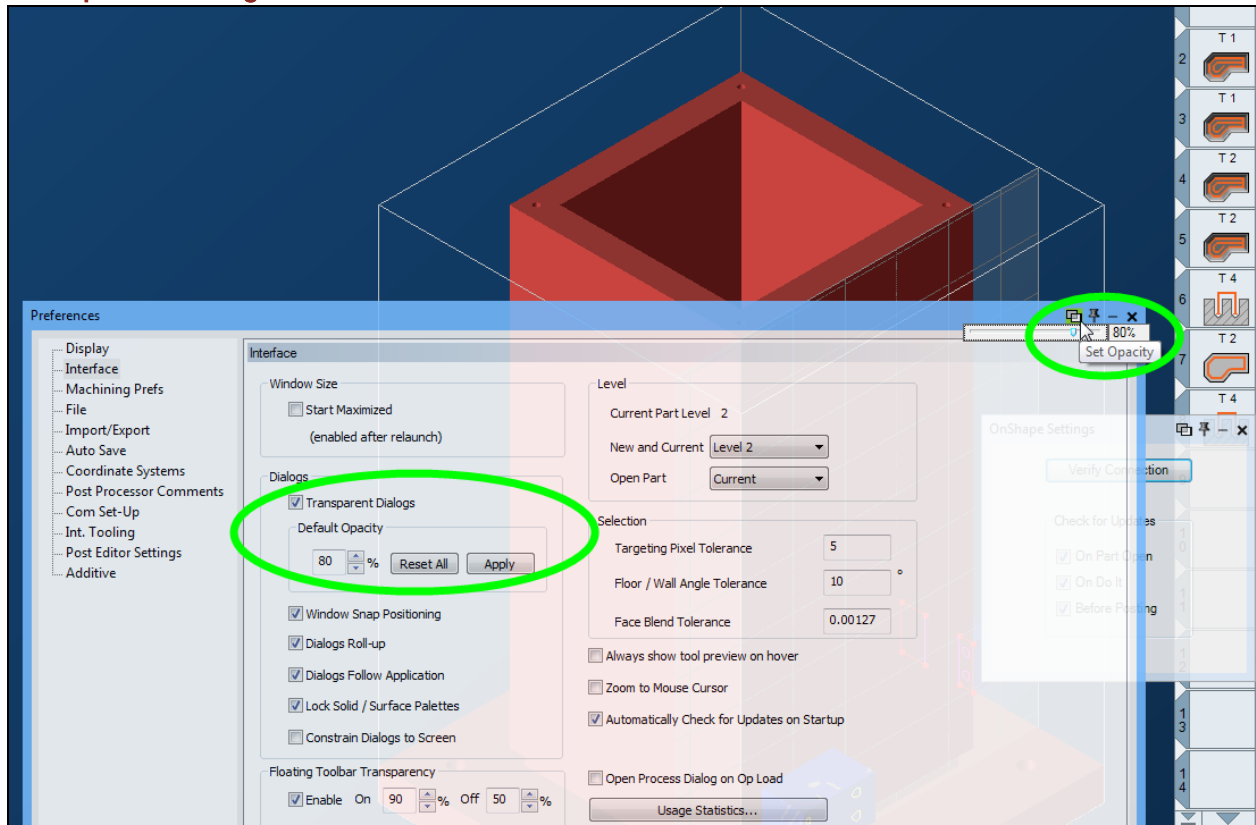
### Window Size

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 allow users to toggle between the Level 1 and 2 interfaces. Although users should use the interface that best suits their 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 for the user. **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 gives users 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.

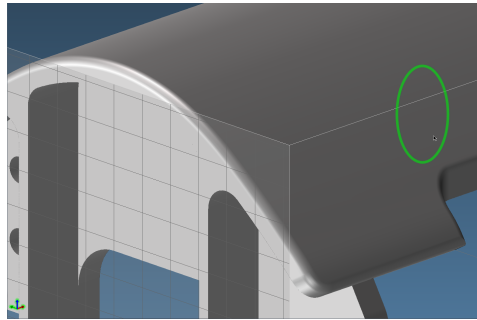
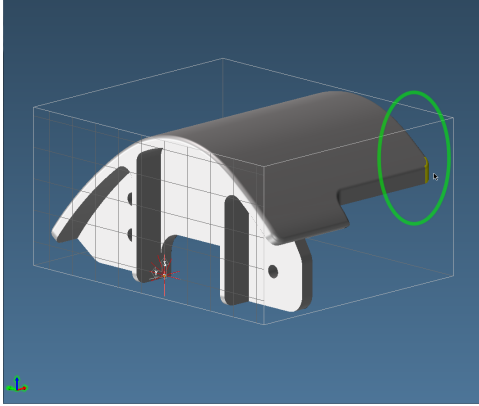
## Selection

### Targeting Pixel Tolerance

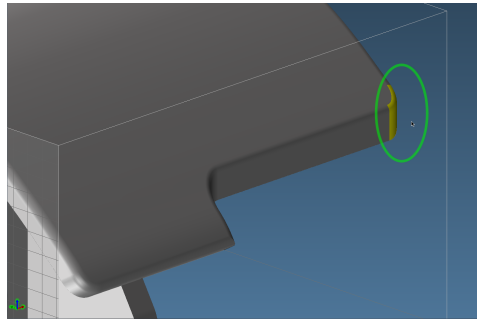
This setting specifies the accuracy you need to have when you select an item in the workspace. The default setting, 5 pixels, means that the workspace item is selected if it lies within 5 pixels of the cursor tip when you click the mouse, and not selected if it lies 6 or pixels from the cursor tip. If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







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



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

### 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. Uncheck to disable this function.

### Open Process Dialog on Op Load

When this checkbox is selected, the process dialog will open whenever the user double-clicks an operation.

### 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 a license onto the dialog for application installation, browse the computer for a license file, or run the Registration Tool to obtain a license file through the Gibbs Activation Service.

- If you already have a license file for v13 or earlier, however, you can simply drag and drop it onto this dialog, or click **Browse for your license file on this computer...** to navigate to it and select it.



**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 to users of the Network Licensing option. This preference allows users to 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.

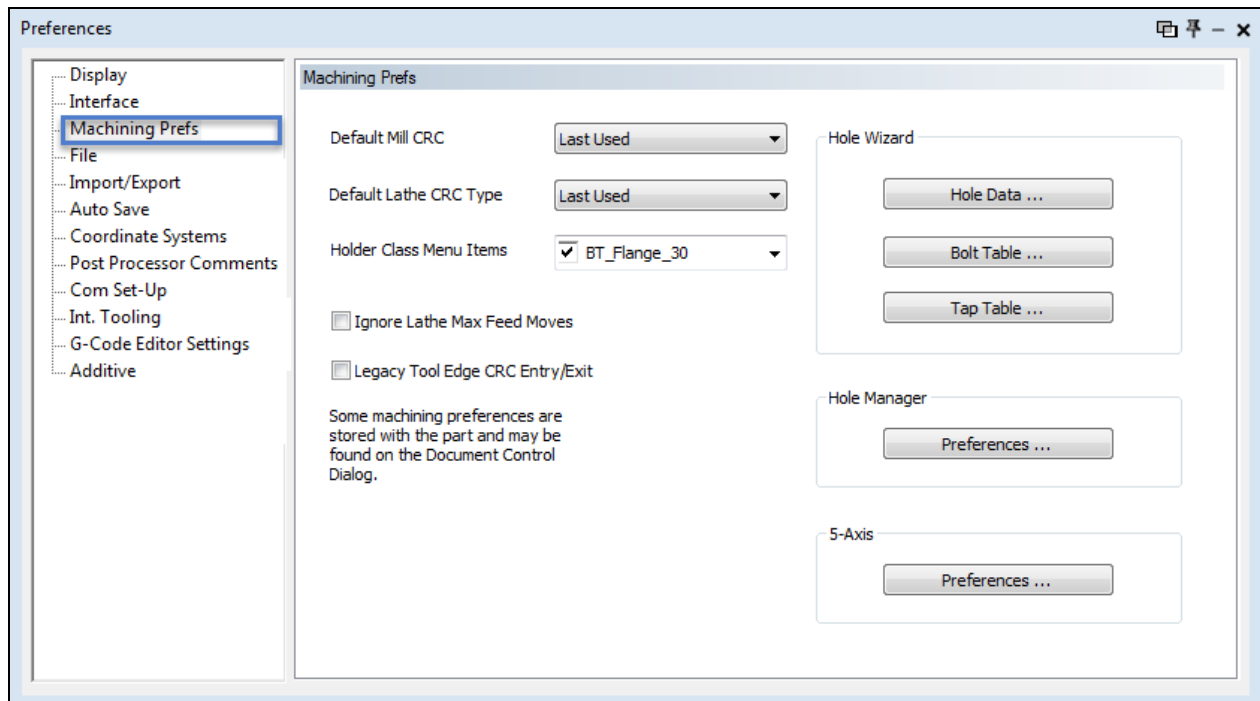
## Digitizer Calibration

This preference allows a user 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.

## 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, unless in **Tool Edge** mode, in which case, (because we apply CRC to the last pass only) the last pass will be calculated from the tool edge.

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



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.

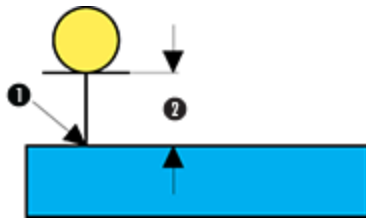
### Last Used

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.

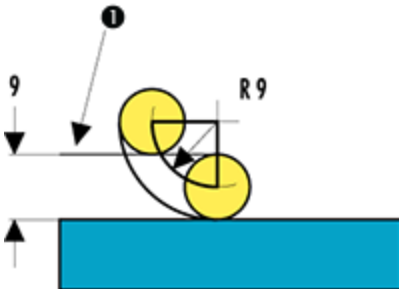


1. Machining Marker
2. Process dialog Line value

### 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.

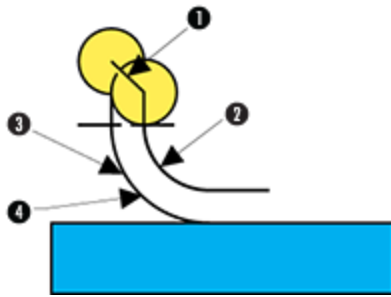
Tool Edge CRC defines the line length and arc radius in terms of edge position. If you specify “0” for the line length, the tool edge starts on the geometry, and a tool radius-long line will be output in the G-code for CRC to turn on. This results in “no move.” If a line-arc entry is used, you have to 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.

#### 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

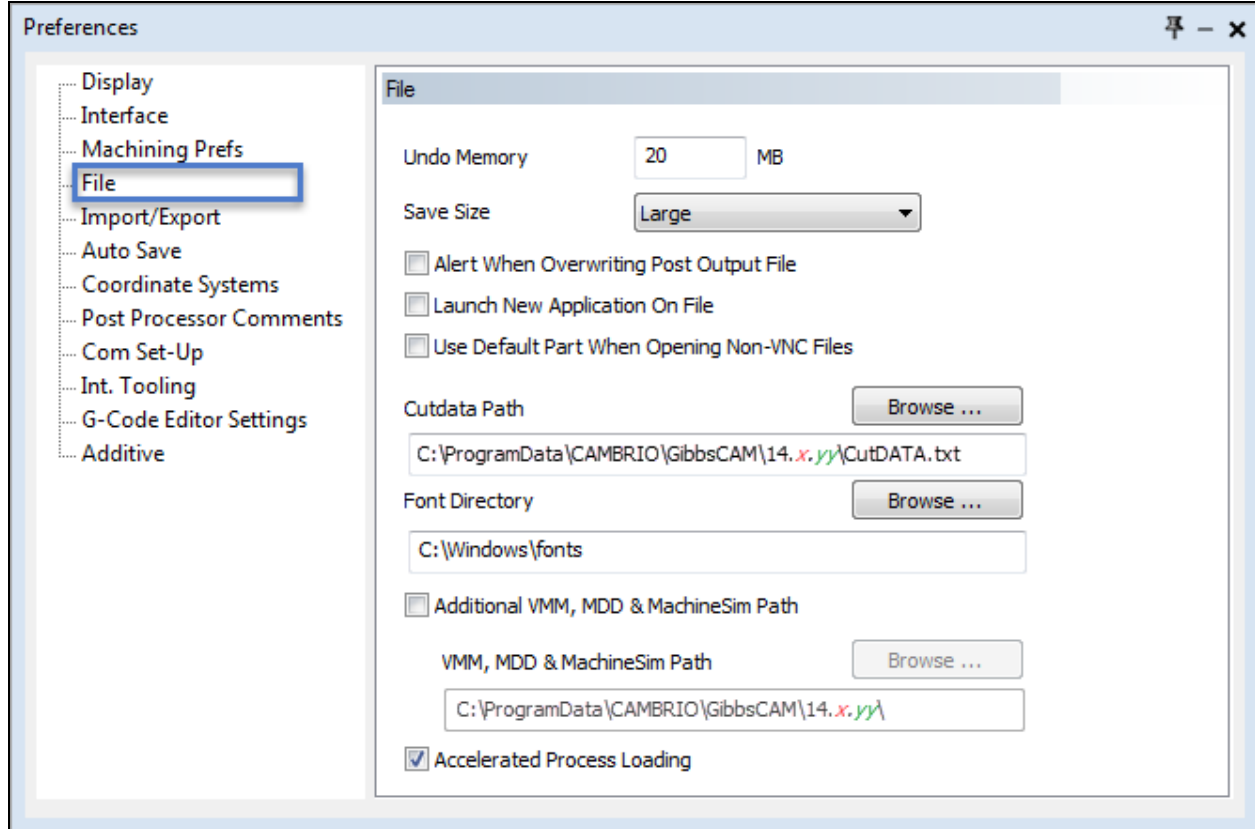
Opens a dialog which enables setting of preferences for Hole Manager. This is fully described in the [Features](#) guide.

#### 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 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\CAMBRIO\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

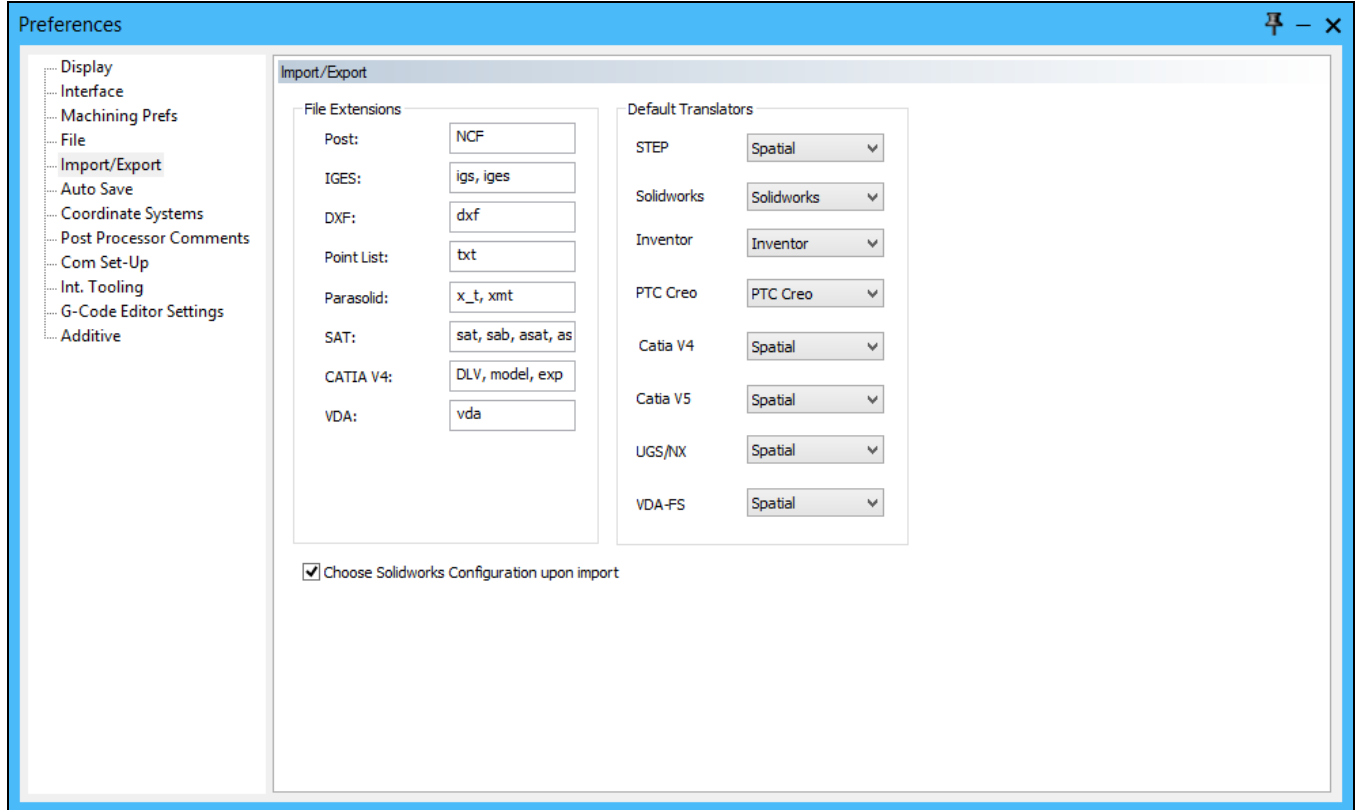
This preference allows you to manually set the directory where the system looks for VMMs, MDDs, and Machine Sim models.

### 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 Extension

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.

### Choose Solidworks Configuration upon import

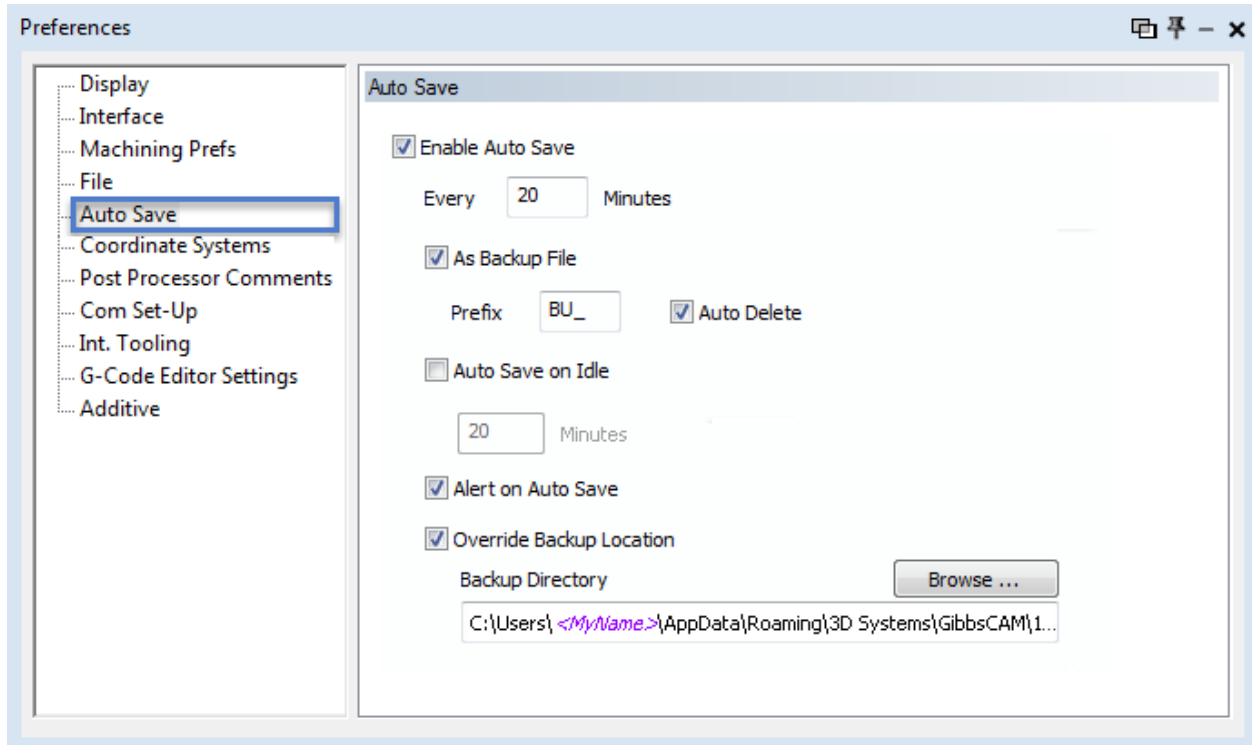
SolidWorks configuration allows the creation of multiple variations of a part or assembly within a SolidWorks file. If checked, a dialog will appear during import to enable selection of the configuration.

Unchecked, the active configuration will be used.

### 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. For more information, see the [Data Exchange](#) guide.

## Auto Save Preferences

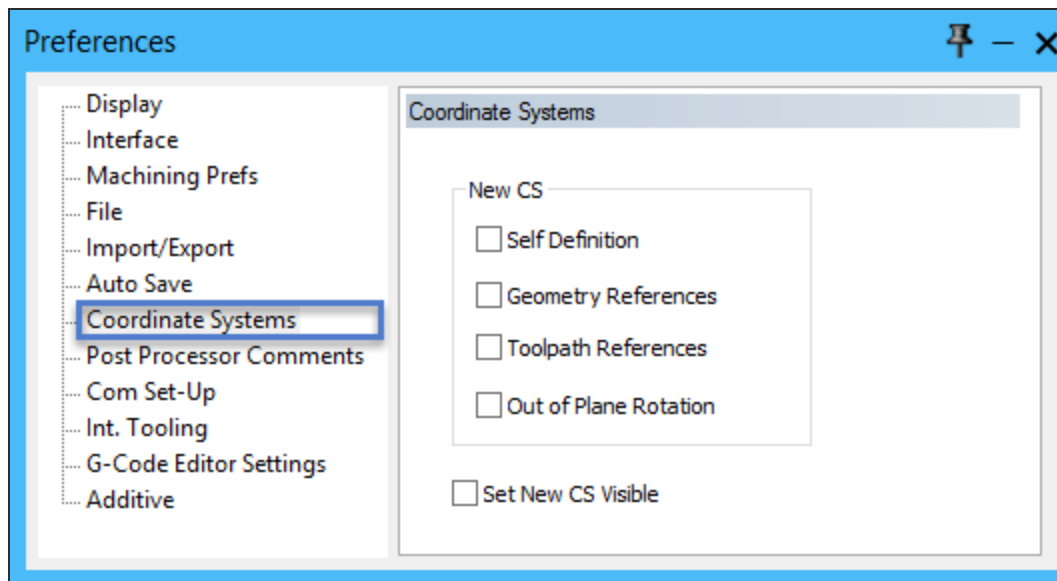


GibbsCAM has the option of automatically saving the file you are working on. You should be aware that saving will purge any available Undos and Redos.

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

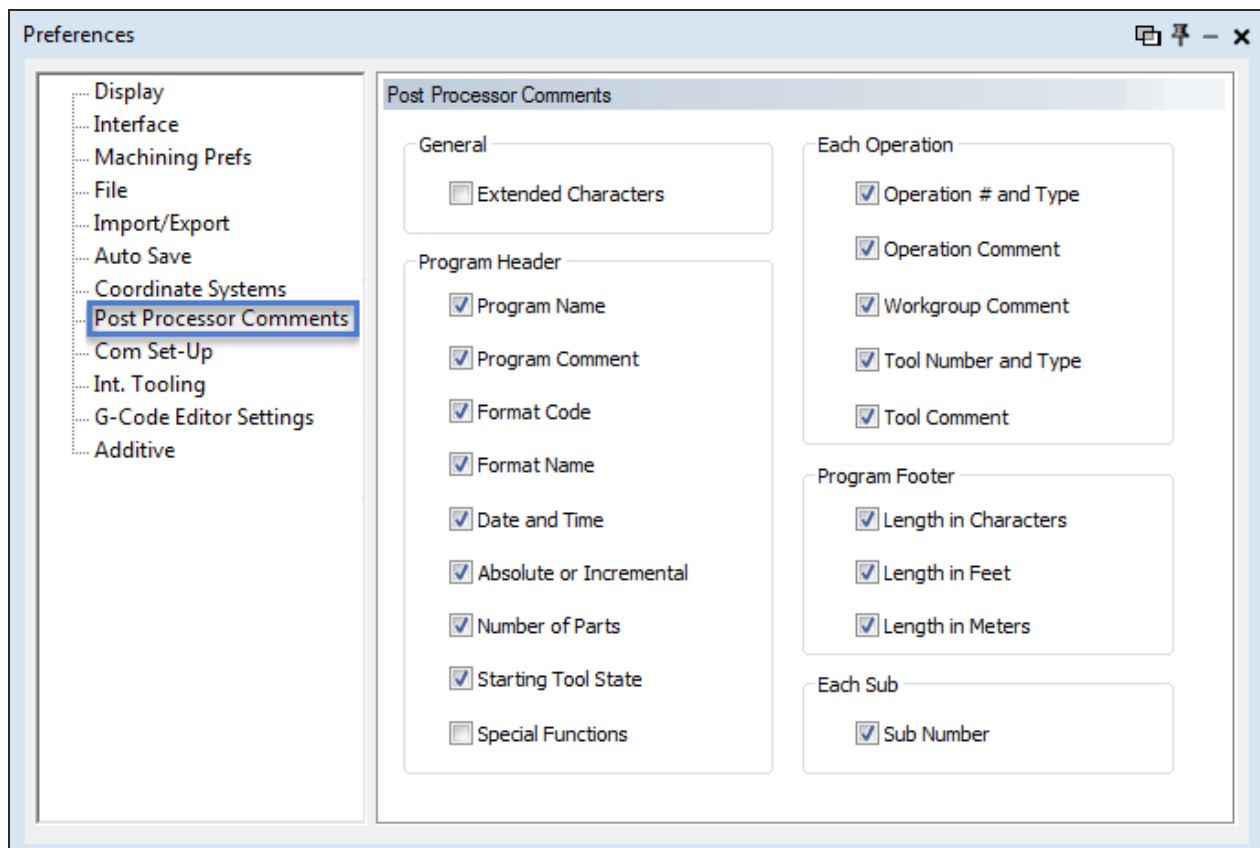
This has an effect when modifying a CS that was used for machining.

- 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



The Post Processing preference allows 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

The screenshot shows the 'Preferences' dialog box with the 'Com Set-Up' tab selected. The left sidebar lists various preference categories, with 'Com Set-Up' highlighted. The main area contains the following settings:

- Protocol Name:** Fanuc 6M (selected in a dropdown), with an 'Add' button and a 'Remove' button.
- Baud Rate:** 9600 (dropdown)
- Parity:** Even (dropdown)
- Data Bits:** 7 Bits (dropdown)
- Stop Bits:** 1 Bit (dropdown)
- Handshake:** XON/XOFF (dropdown)
- Port:** Com 1 (dropdown)
- End Of Block:** LF (dropdown)
- User's EOB:** (empty text field)
- Special Beginning and End Of File:**
  - BOF: (empty text field)
  - EOF: (empty text field)
- Line and Char Delay Specification:**
  - Line: 0 (text field) with a multiplier of 1/60
  - Char: 0 (text field) with a multiplier of 1/60

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:

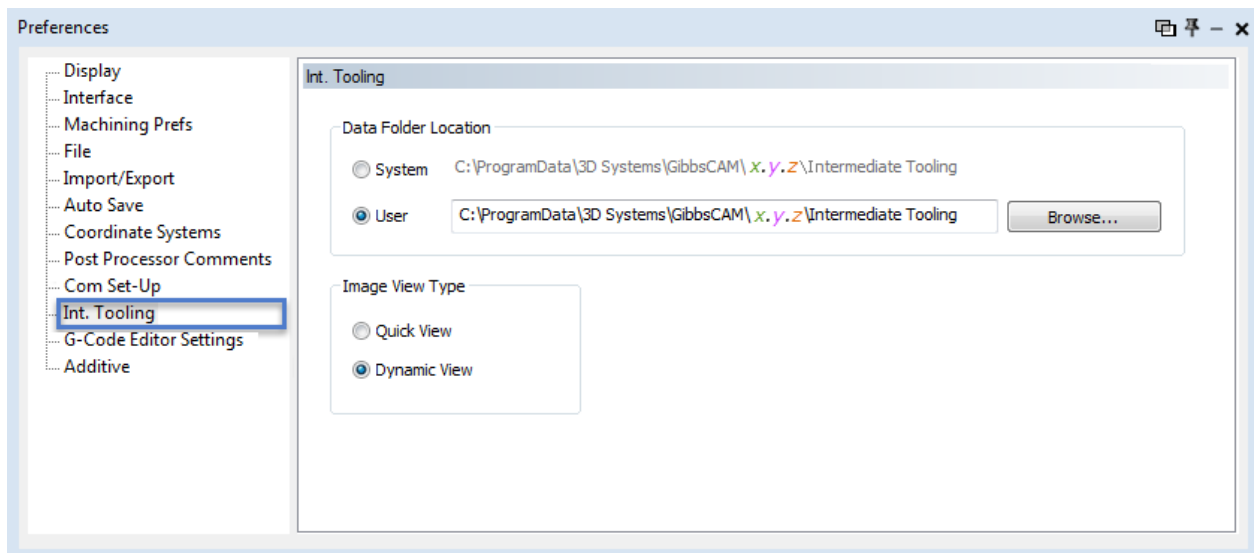


To change protocol information, select the protocol from the list and change the information. The changes are automatically saved when the dialog is closed or when a different protocol is selected in the list.

### Removing a Protocol:

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

## Intermediate Tooling Preferences



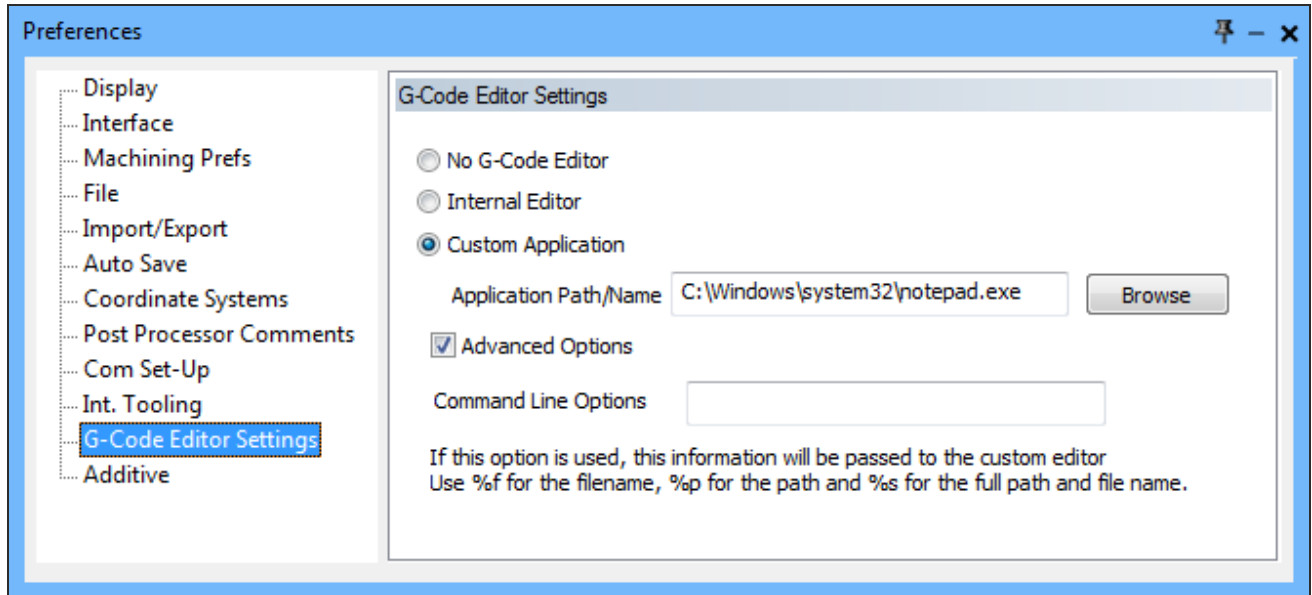
This lets you set the default folder used for Intermediate Tooling.

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



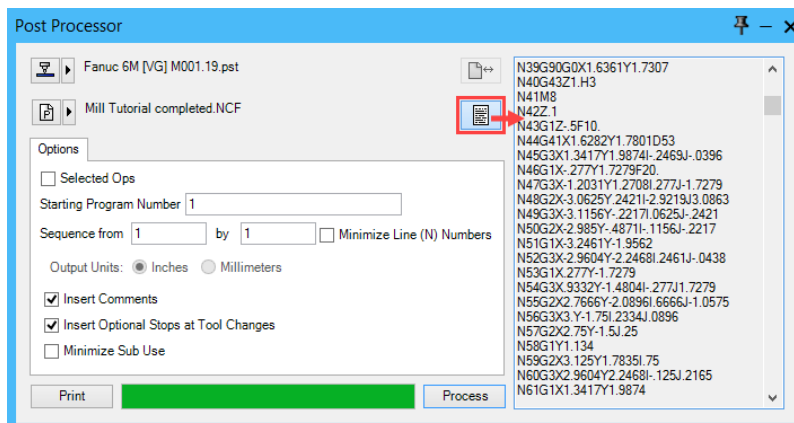
This lets you select an application for displaying and editing post files.

## 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

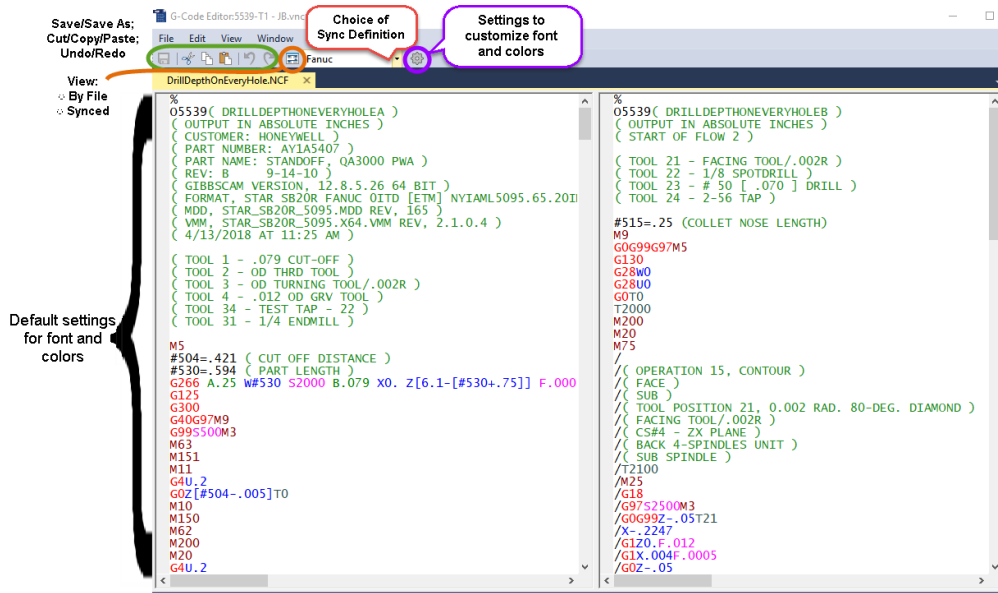
You can choose to disable the editor, in which case you will still be able to look at the output in the post Processor dialog window.



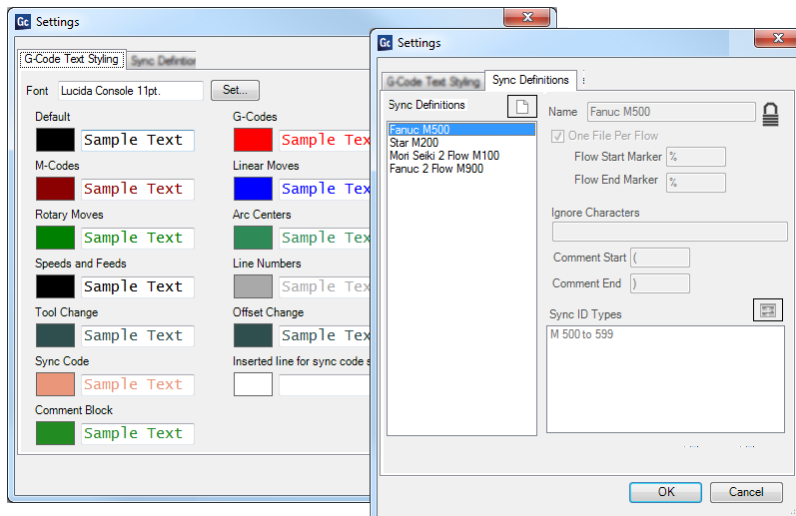
### Internal Editor

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 **synced 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

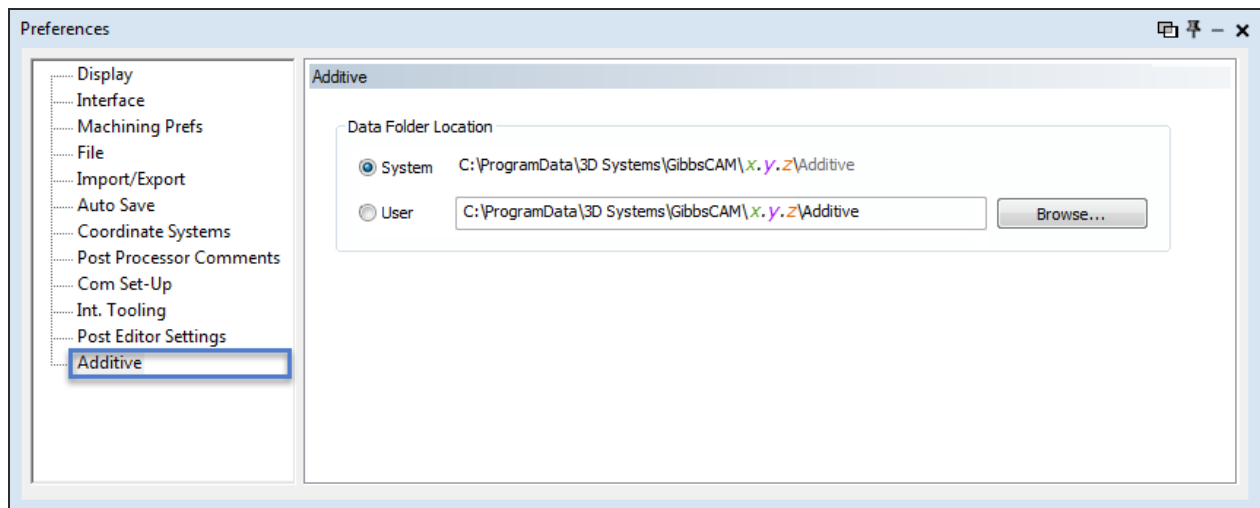
menu using  External G-Code Editor.

### 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.

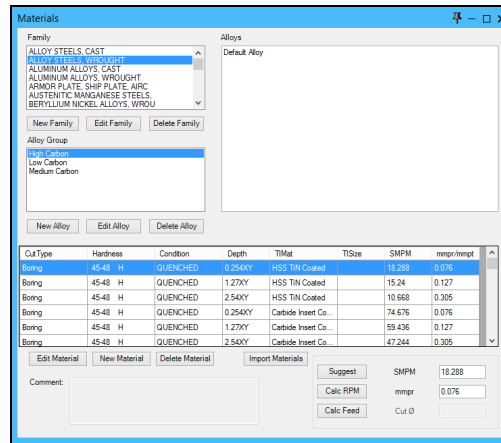
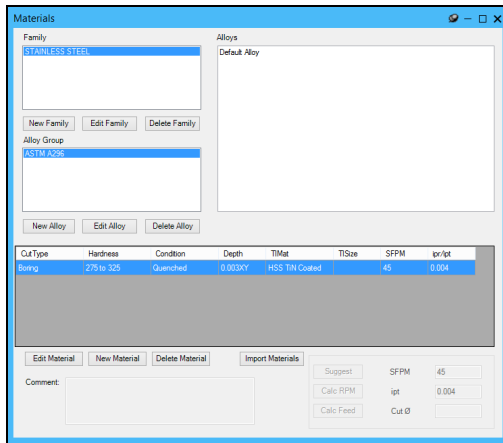
You can select the **Advanced Options** checkbox if you want to use the ability to pass variables to the custom editor. When this option is enabled, **%f** passes the filename; **%p** passes the path down to the folder that contains the file; and **%s** passes the path and filename.

## Additive Preferences


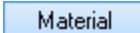


This lets you set the default data folder used for Additive Machining.

## Materials

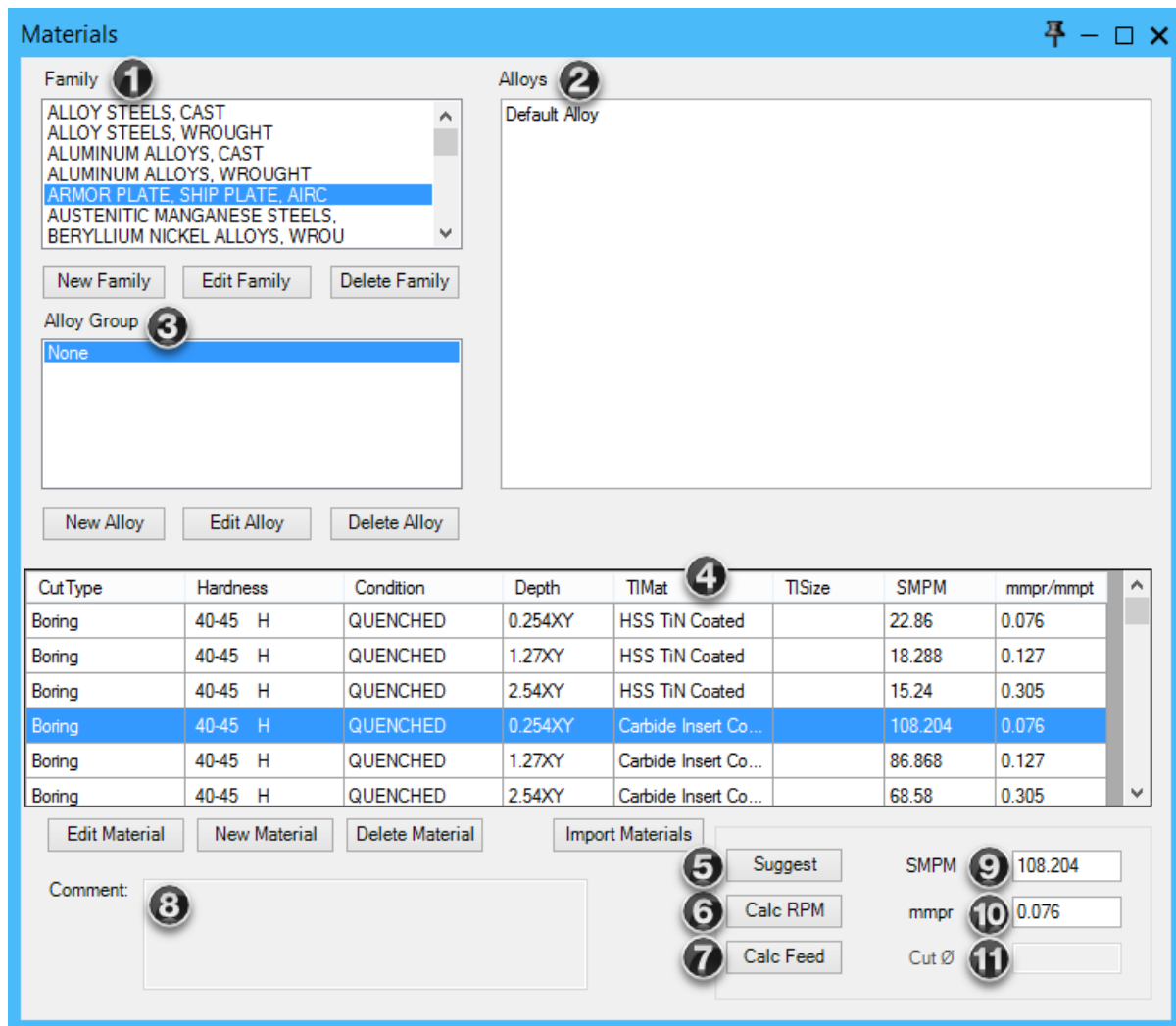


## The Material Database with and without CutDATA

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\CAMBRIO\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-upercase 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.

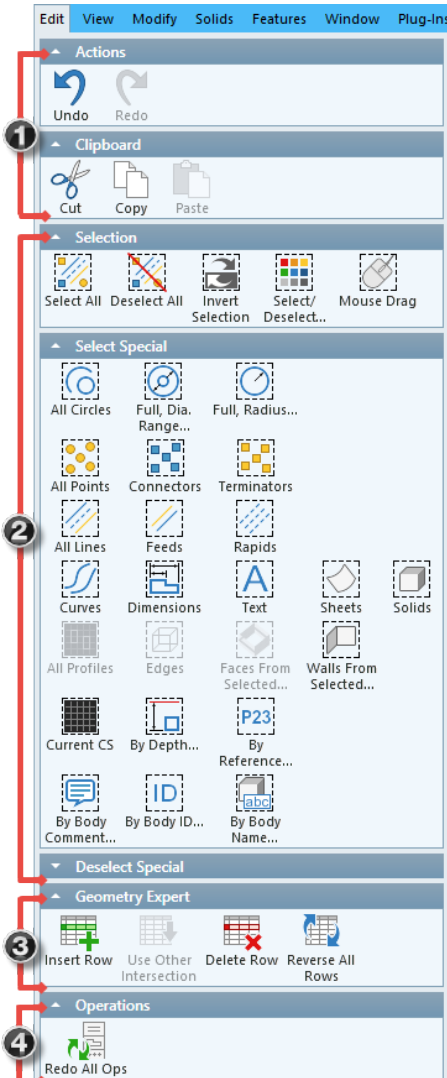
### Calculate RPM

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. If changes have been made to part geometry or solids, selecting Redo All Ops will automatically reprocess all of the operations and the new toolpath will reflect the changes.

# Clipboard



The Clipboard is used as a temporary storage space for geometry, solids, text, tools 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) will delete whatever items are selected and place them on the Clipboard. Any contents of the Clipboard will be replaced at this time.



Using **Copy** (Ctrl+C) will perform the same action as Cut, but will not delete the selected item, and place a copy of the item in the Clipboard.



Choosing **Paste** (Ctrl+V) can do two things. If something is selected, it will replace that selection with the contents of the Clipboard. If nothing is selected, the contents of the Clipboard will be pasted in either the drawing window or in an active text box depending on whether the Clipboard contains text or geometry.

The Clipboard can only hold one selection at a time. This means that it can hold either text or geometry, 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 exact same GibbsCAM version for the paste to work. Items can be Cut, Copy and Pasted even after closing GibbsCAM and relaunching it. 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**

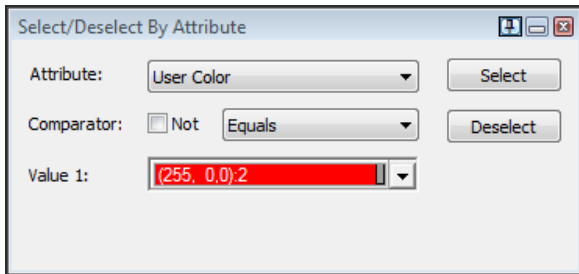


**Deselect All**

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**

When the Mouse Drag item is selected, you can drag the cursor around an area to select all the geometry contained in that area. This is equivalent to holding down the Shift key when click-dragging the mouse.


## Select Special

### Circles

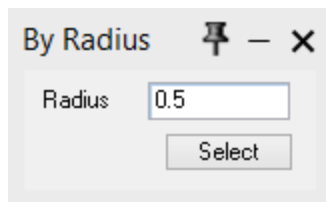
There are three circle selection options.  All Circles will select all circles in the current workgroup. Full, Radius and Full, Dia. Range will bring up the following dialog boxes.

 **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**

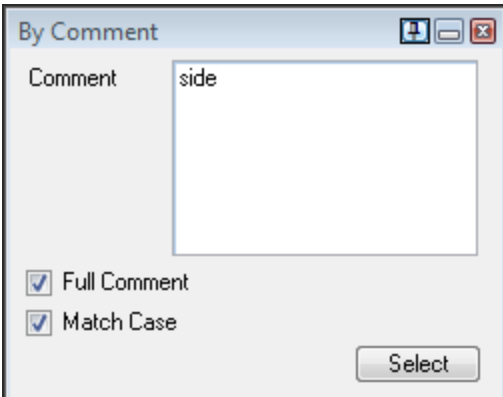
The Current CS item provides a quick way to select all geometry and bodies in the current coordinate system. Geometry or bodies defined in coordinate systems other than the currently active CS will not be selected.

 **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**

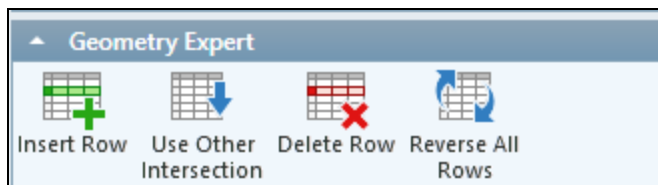
To select bodies (solids and sheets) **By Body Name**, you enter some data in the **Name** box to specify the selection. All bodies have names – either automatically generated (such as Sphere 35) or given by the user. When **Full name** is selected, only bodies whose names exactly match the text entry 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**

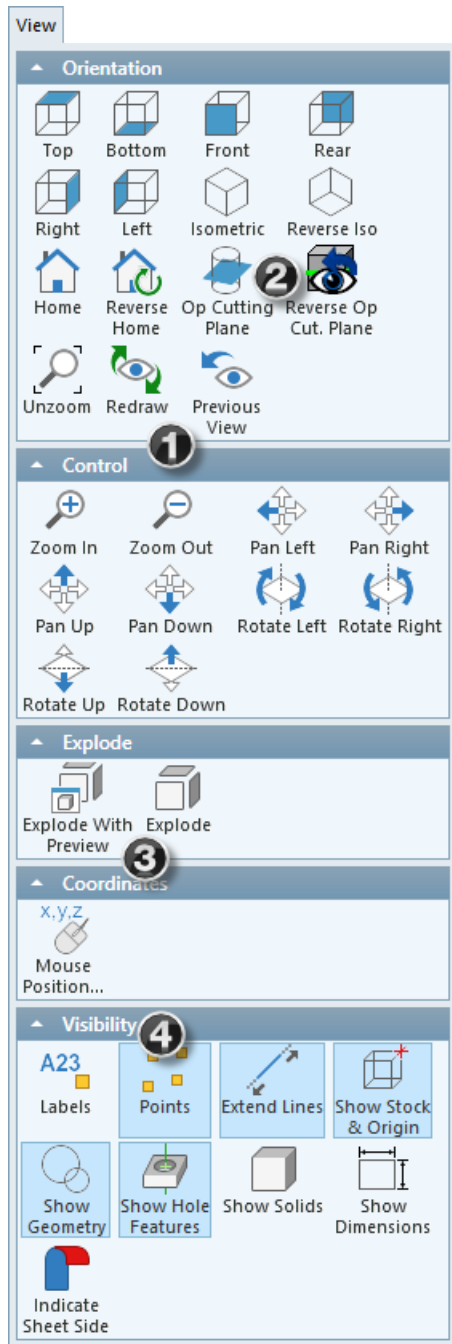
This item will delete the selected row from the spreadsheet. As soon as the row is deleted, the system will adjust the remaining shape to maintain the appropriate connections, if possible.

 **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.



# 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 67

- [“Visibility” on page 67](#)

## View Orientation and Control



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



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



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



(**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.



(**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.



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



(**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.



(**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.



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

 **Zoom In Out**

(**Ctrl++**) Increases/Decreases the scale of the view, enlarging/shrinking by 25% each time you press **Ctrl++**.

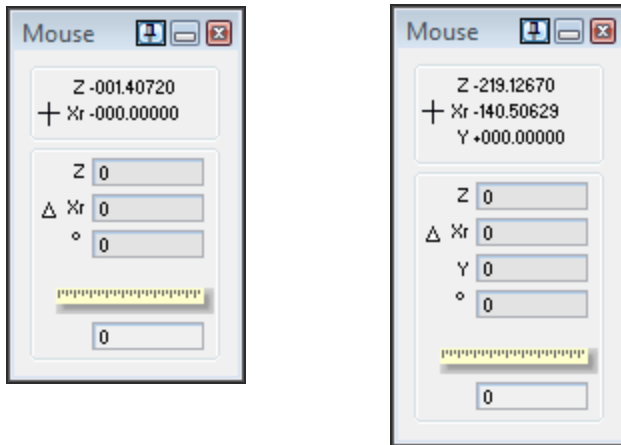
## Explode/Coordinates

 **Explode View with Preview**

View each connected shape and solid with space separating it from the others, displayed from the camera's 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

**A23 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**) If this item is on, all points (plain points, connectors, and terminators) will be shown.

 **Extend Lines**

When this is active, all unconnected lines will extend to the edge of the stock. If it is turned off, lines will only extend 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 will be 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 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**

If this is turned on, solids and sheets in the workspace will be displayed.

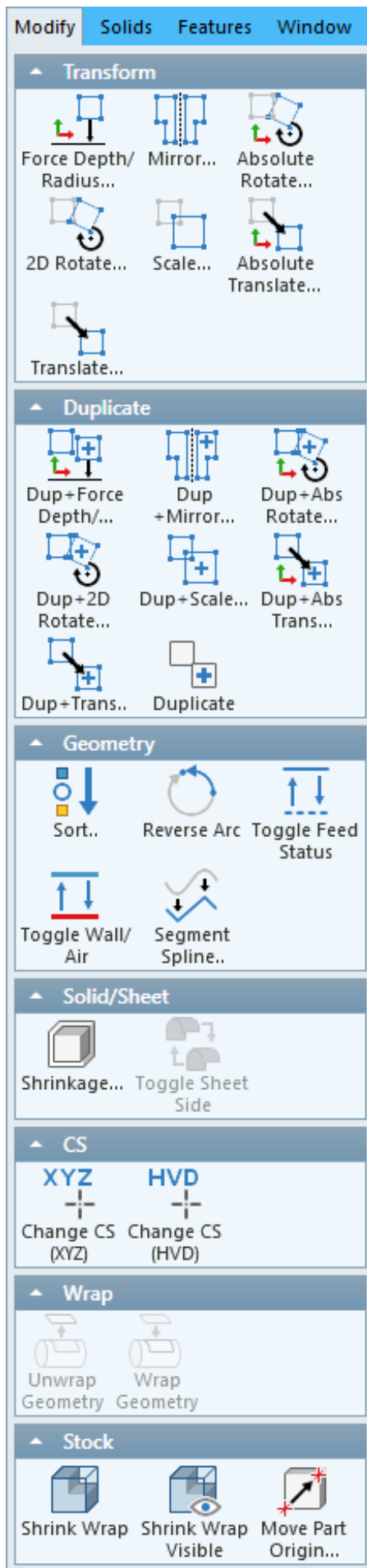
 **Show Dimensions**


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**

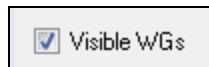
When enabled, the negative and positive sides of each sheet will be drawn in different colors.

# The Modify Menu



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.



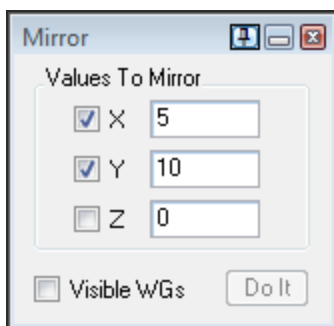
This checkbox option is found in several of the functions accessed from the **Modify** menu. The functions can be applied to entire workgroups as well as to any selected features. This function is accomplished by checking the **Visible WGs** checkbox and clicking **Do It**. Visible workgroups include 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](#)
- [Other Modify Commands](#)
- [Wrap/Stock](#)

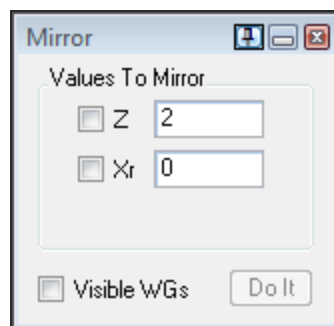
## Transform



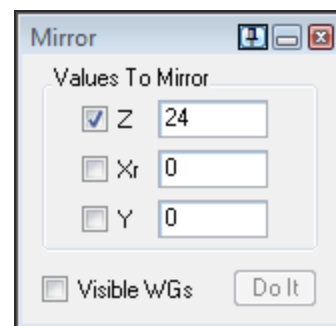
This selection sets the absolute **Z** value of all selected geometry to the **Z** value entered. This option is unavailable when working in the Turning module and therefore the item is grayed out in the menu.



Mill



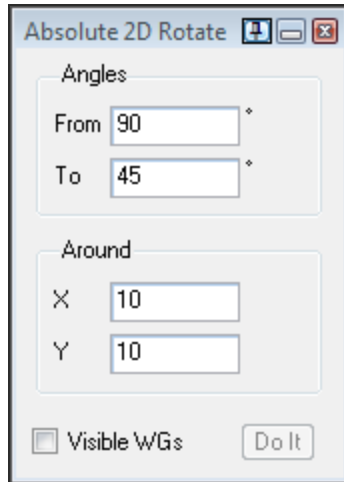
Turning



Mill/Turn

This selection will move the selected geometry or bodies to the other side of the axis or axes specified around an axis point entered in the text boxes. Clicking Do It will perform the mirroring function. This dialog can remain onscreen for further use while other functions are performed.

### Absolute Rotate



Absolute 2D Rotate

Angles

From 90 °

To 45 °

Around

X 10

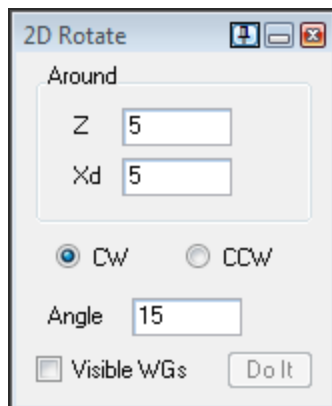
Y 10

Visible WGs

Do It

This selection will rotate selected geometry or bodies 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. The angle of a geometric element can be determined by interrogating the feature. Clicking Do It will perform the rotating function. This dialog can remain onscreen for further use while other functions are performed.

### 2D Rotate



2D Rotate

Around

Z 5

Xd 5

CW  CCW

Angle 15

Visible WGs

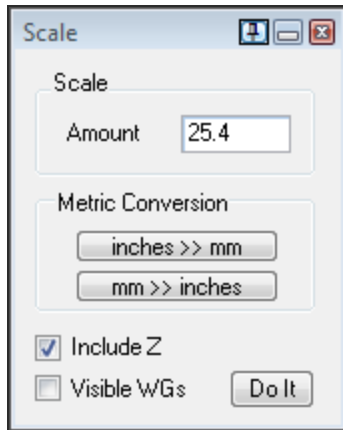
Do It

This selection will 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. Clicking Do It will perform the rotating function. This dialog can remain onscreen for further use while other functions are performed.





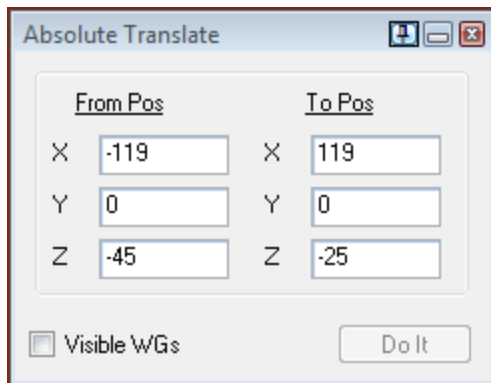
## Scale



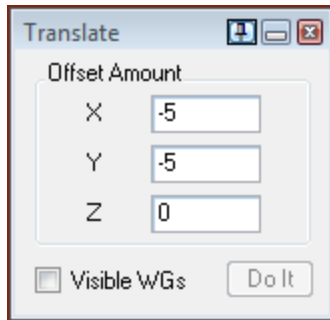
This selection will change the size of the selected geometry by the amount specified. The value typed in the Amount box is a ratio of one, one being full size. The inches >> mm or mm >> inches buttons will change the units of measurement accordingly. The Mill module has an Include Z option. If the Include 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 is at Z0. Clicking Do It will perform the scaling function. This dialog can remain onscreen for further use while other functions are performed.



## Absolute Translate



This selection will move the selected geometry or bodies from a position to a new position. This is an absolute move by a certain value. Clicking on Do It will perform the translating function. This dialog can remain onscreen for further use while other functions are performed. 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.

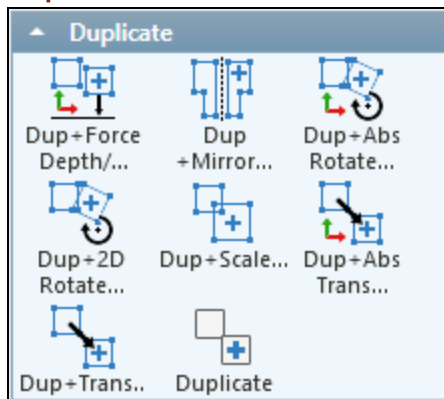
 **Translate**


This selection will move the selected geometry or bodies by the specified amount. Clicking **Do It** will perform the translating function. This dialog can remain onscreen for further use while other functions are performed.

## Other Modify Commands

### Duplicate

#### Duplicate And...

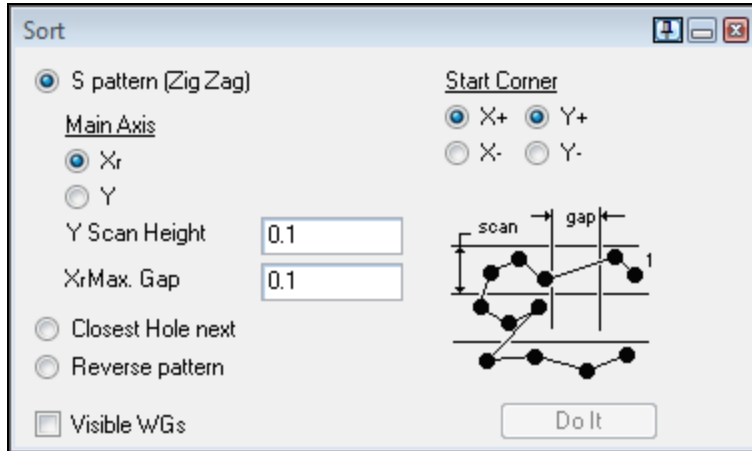


This item will Duplicate and perform the specified action. Force Depth, Mirror, 2d Rotate, Scale or Translate the selection. First, the selected geometry will be duplicated. Then, depending on the option selected, the selected geometry is moved to one or more new locations based on the number typed in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



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**

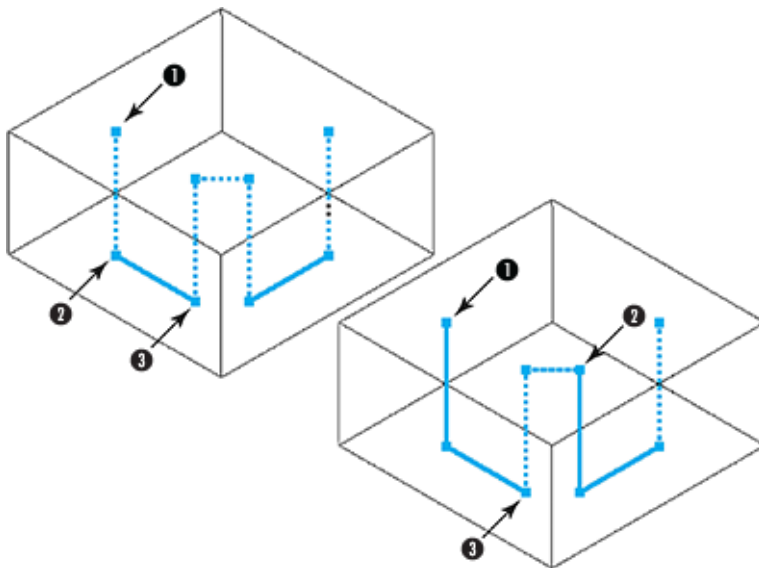
This option is fairly self explanatory. The system starts with the first point or circle selected and then selects the point or circle that is closest. This process continues until all holes have been selected. This option is useful when creating curves.

**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**


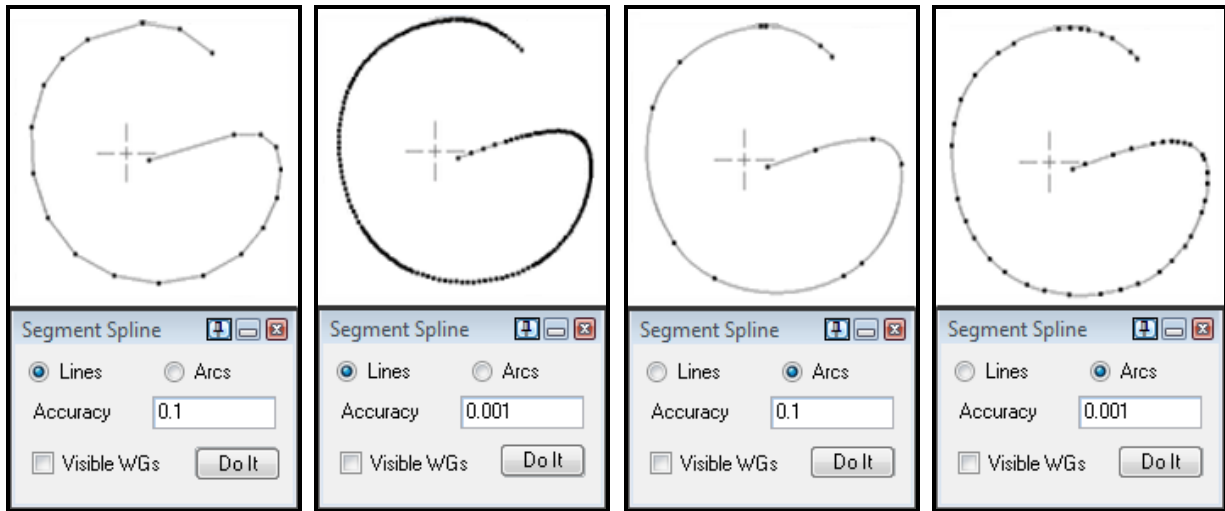
1. Start Point, Begin Rapid
2. Begin Feed
3. Begin Rapid

**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**


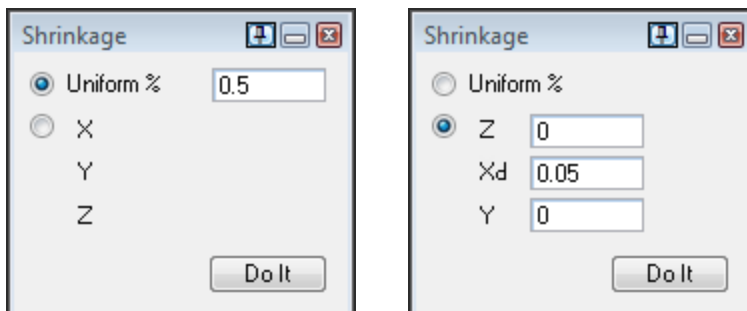
An example of various spline segmentation settings.

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 allows users to perform uniform or axial

reductions or enlargements on selected bodies. The Shrinkage range is -10% to 10%. A selected body can be reduced or enlarged up to 10% of its original size.

$$\text{Final Size} = (100 - \text{Shrinkage}\%) * \text{Start Size} / 100$$

### 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)

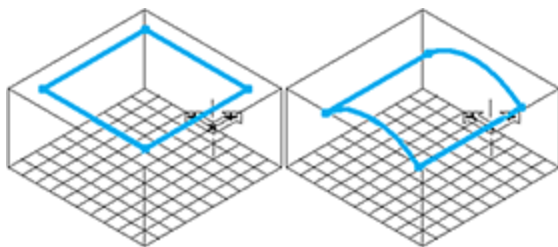
(**Ctrl+V**) When Change CS (HVD) is used, all selected geometry will be assigned to the current coordinate system, and the HVD values of the geometry will be preserved. This means that the geometry will be modified to be planar to the new CS location preserving the relative position. If the Advanced 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.

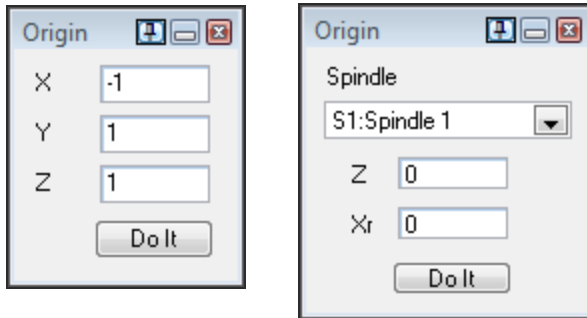
### Shrink Wrap

(**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

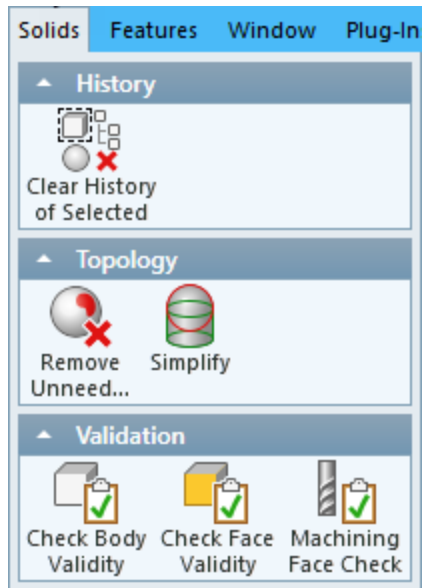
(**Ctrl+]**) When this item is selected, the window zooms out or zooms in to display all *visible* workspace elements without excess blank space. This is especially useful when working with part files that include hidden () workgroups or items in the Body Bag.

### 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 X0, Y0, Z0 or Z0, X0 will change incrementally by the values in this dialog and a new X0, Y0, Z0 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.

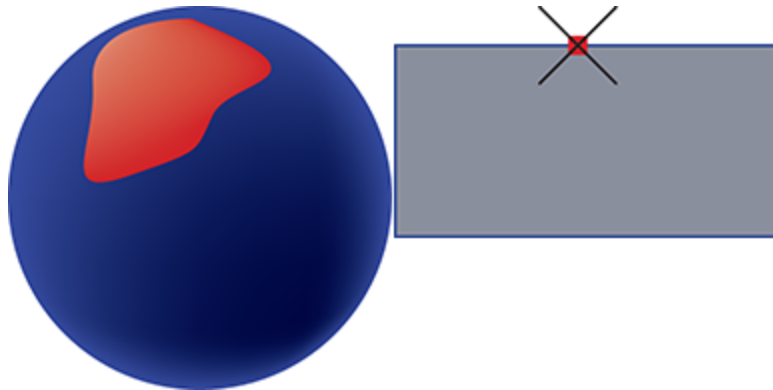
### Check Face Validity

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.

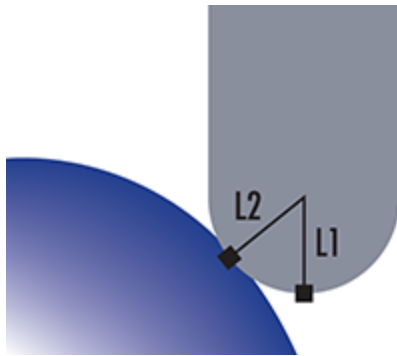
### Check Trimmed Surface Polyline

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

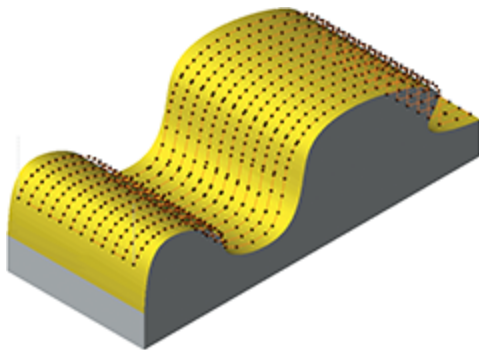
This item verifies the validity of trimmed surface edges to ensure proper machining. Check Trimmed Surf. Edges is only necessary when using the Gen 2 Engine in surfacing operations.

### Check Op Gouges



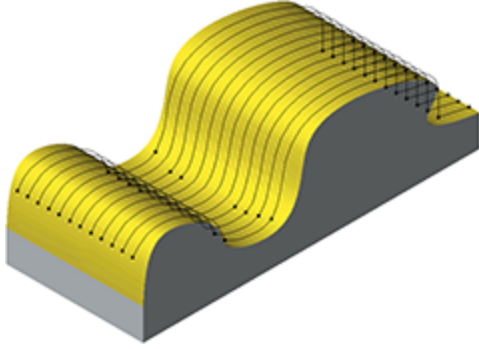
The system will perform a check to see if a selected operation gouges a selected body. A gouge is defined as being off by greater than 50% of the machining tolerance. This command only works with ball endmill tools. If the operation gouges, two lines will be drawn from the tool to show the gouge. The 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 face 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

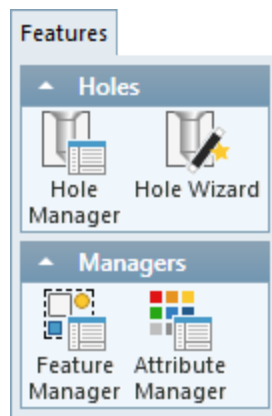
Geometry Expert must be open to use this function. Selecting this command will output the Geometry Expert's contents to a text file named `GEDump.txt`. The file is created on the Desktop.



### 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



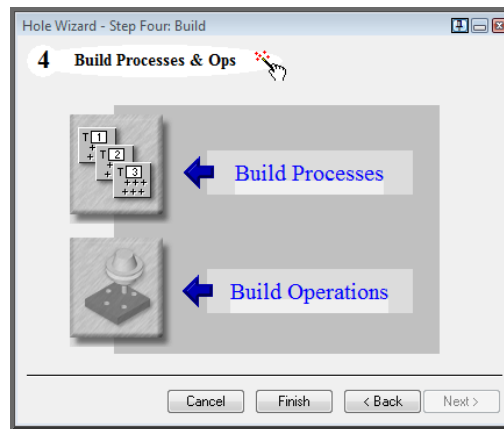
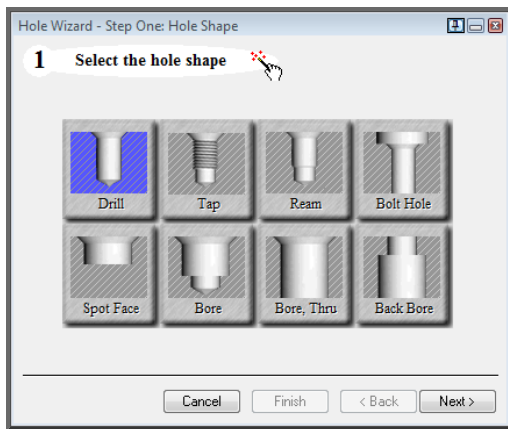
The **Features** menu provides access to two tools specifically for Hole features and two tools that help you manage user-defined features..

- **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

...	#grps	Type	End Condition	
H1	0	Bolt Hole	Through	20.0
H2	0	Bolt Hole	Through	20.0
H3	0	Bolt Hole	Through	20.0
H4	0	Bolt Hole	Through	20.0
H5	0	Drill	Blind(System)	20.0
H6	0	Bolt Hole	Through	80.0
H7	0	Compound	Blind(System)	14.0
H8	0	Drill	Blind(System)	14.0

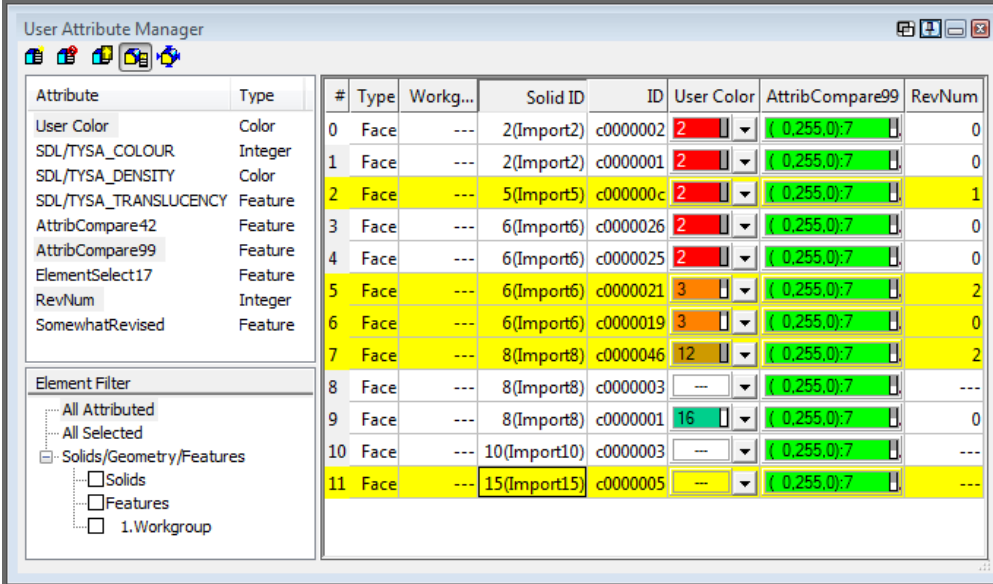
Buttons: Run AFR, Create From Geo, Fit Columns, Delete All, Lock Selection (checked)

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



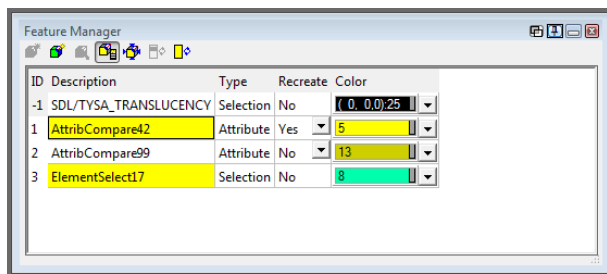
Attribute	Type	#	Type	Workg...	Solid ID	ID	User Color	AttribCompare99	RevNum
User Color	Color	0	Face	---	2(Import2)	c0000002	2	{ 0.255.0}:7	0
SDL/TYSA_COLOUR	Integer	1	Face	---	2(Import2)	c0000001	2	{ 0.255.0}:7	0
SDL/TYSA_DENSITY	Color	2	Face	---	5(Import5)	c000000c	2	{ 0.255.0}:7	1
SDL/TYSA_TRANSLUCENCY	Feature	3	Face	---	6(Import6)	c0000026	2	{ 0.255.0}:7	0
AttribCompare42	Feature	4	Face	---	6(Import6)	c0000025	2	{ 0.255.0}:7	0
AttribCompare99	Feature	5	Face	---	6(Import6)	c0000021	3	{ 0.255.0}:7	2
ElementSelect17	Feature	6	Face	---	6(Import6)	c0000019	3	{ 0.255.0}:7	0
RevNum	Integer	7	Face	---	8(Import8)	c0000046	12	{ 0.255.0}:7	2
SomewhatRevised	Feature	8	Face	---	8(Import8)	c0000003	--	{ 0.255.0}:7	---
		9	Face	---	8(Import8)	c0000001	16	{ 0.255.0}:7	0
		10	Face	---	10(Import10)	c0000003	--	{ 0.255.0}:7	---
		11	Face	---	15(Import15)	c0000005	--	{ 0.255.0}:7	---

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

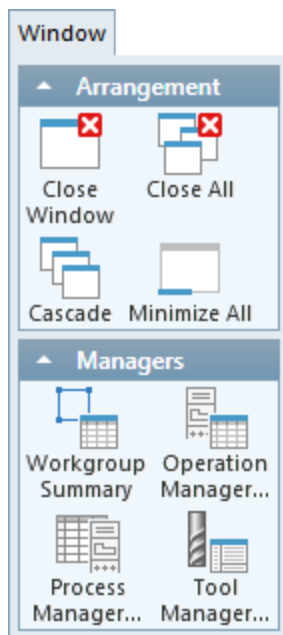
Feature Manager lists user-defined features in a table that displays their **ID**, **Description**, **Type** (element-selection-based or attribute-comparison-based), **Recreate flag**, and **Color**:



ID	Description	Type	Recreate	Color
-1	SDL/TYSA_TRANSLUCENCY	Selection	No	(0, 0, 0, 25)
1	AttribCompare42	Attribute	Yes	5
2	AttribCompare99	Attribute	No	13
3	ElementSelect17	Selection	No	8

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 88

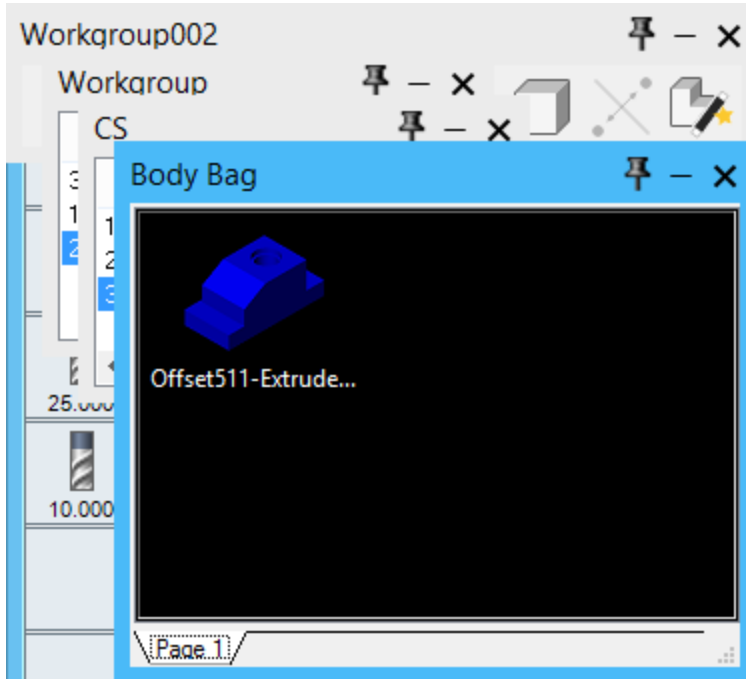
## Arrangement

### Close All

Closes all open sub-menus.

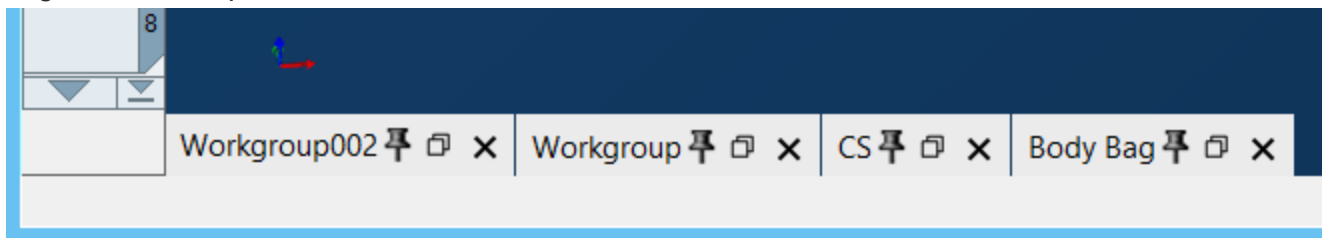
## Cascade

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:

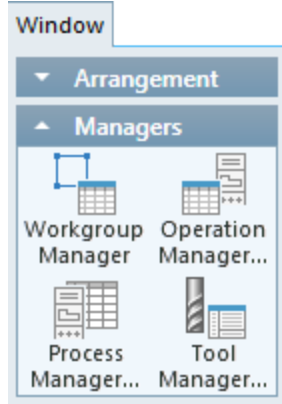


## Managing a Large Number of Processes

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 processes 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.

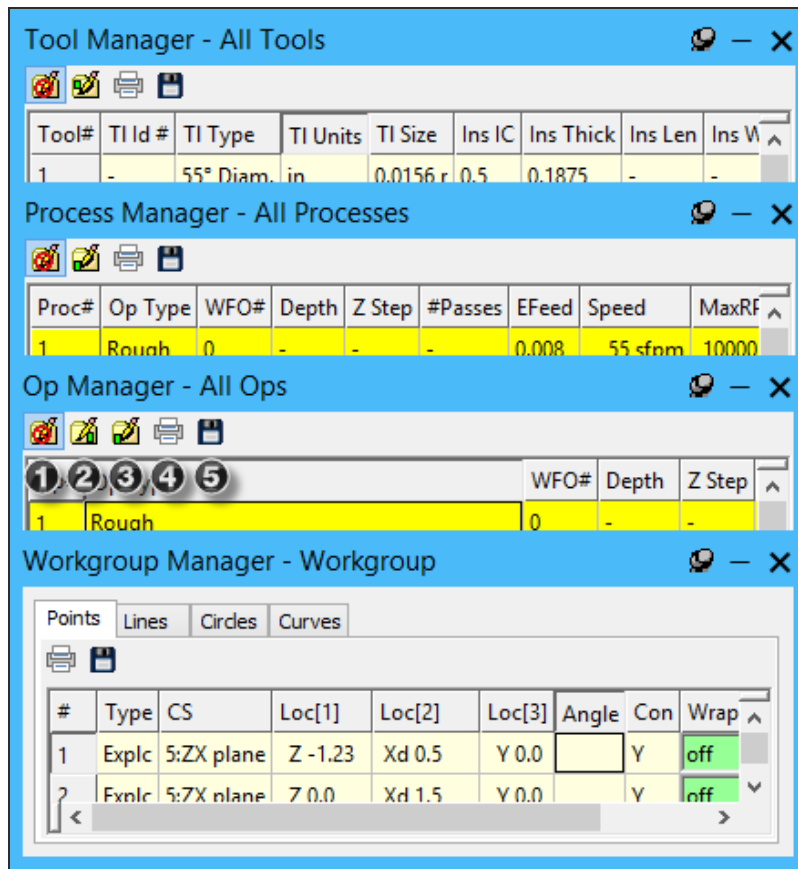
## Viewing Data

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, down indicates high-low.
- Right-clicking anywhere in a row reveals a menu which enables you to:
  - 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. Print
5. Save

## 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 are 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.

- 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.

/&gt;

## Multiple views

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

The screenshot shows the 'Tool Manager - All Tools' dialog box with four views of a tool list grid. Each view has a header row with columns: To...△, TI Id #, TI Type, TI Units, TI Size, Ins IC, and Ins Thic ^.

To...△	TI Id #	TI Type	TI Units	TI Size	Ins IC	Ins Thic ^
1	-	55° Diam.	in	0.0156 r	0.5	0.1875
2	-	35° Diam.	in	0.0156 r	0.375	0.1875
3	-	55° Diam.	in	0.0156 r	0.5	0.1875
4	-	35° Diam.	in	0.0156 r	0.375	0.1875
5	-	Drill	in	0.5 d	-	-
6	-	80° Diam.	in	0.0156 r	0.25	0.0938
7	-	Triangle	in	0.0156 r	0.25	0.1875

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

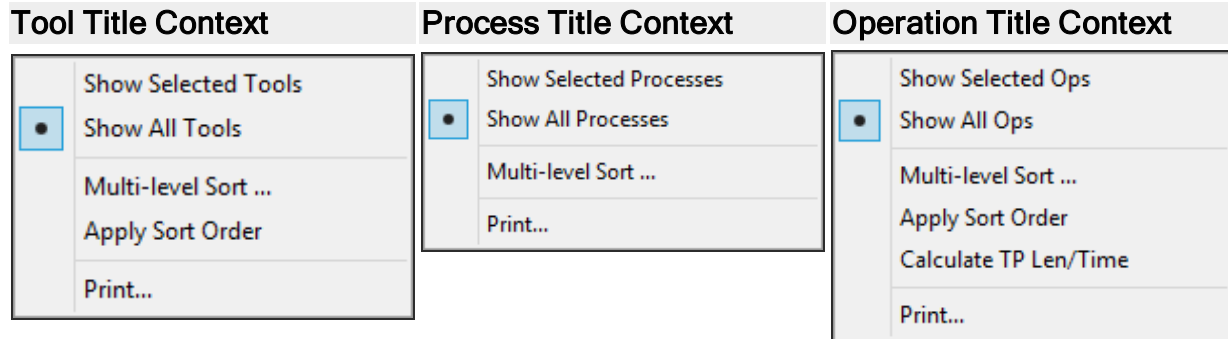
The Manager dialogs provide the following right-click menus:

- 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.

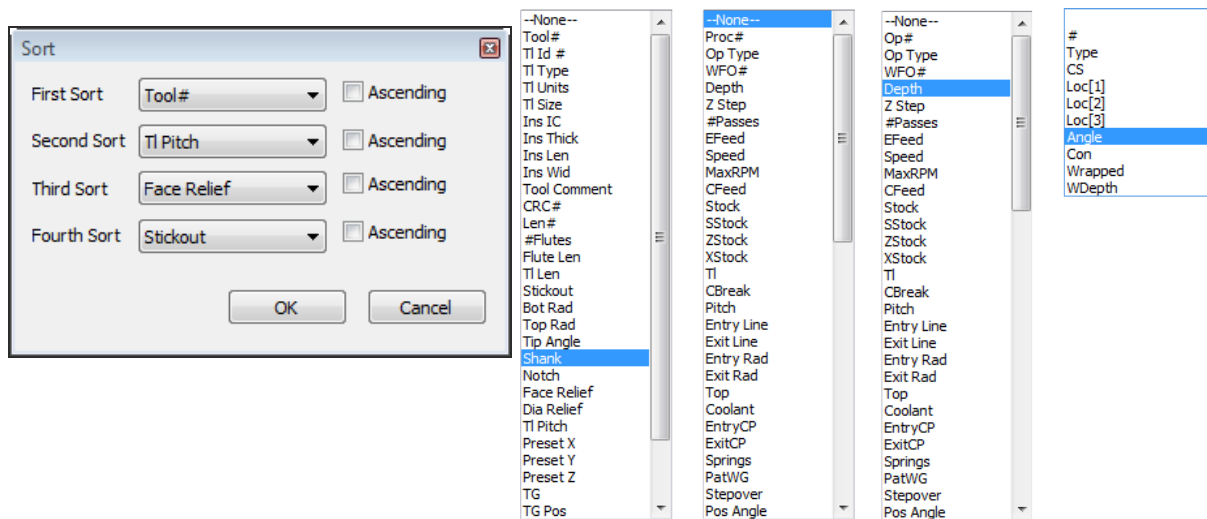


### Show Selected/All:

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.

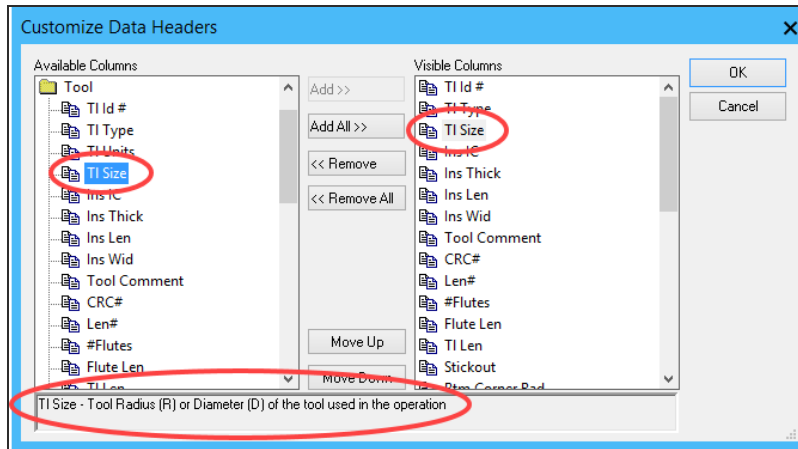


**Important:** Performing a sort changes the order of your operations. If you are not careful you could tell the system to do something you do not want, such as tapping a hole before the hole is drilled. The system does have a reminder to warn you that performing a sort can be harmful but you should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

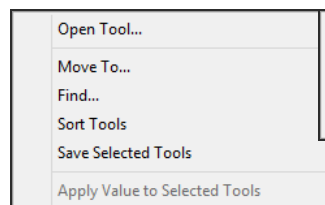
Clicking a data type will provide a brief description of the item at the bottom of the screen. The item is also highlighted in the visible columns window. Please note that data can only appear once in the report.



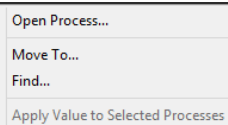
- *To show a hidden column:* Under **Available Columns**, select a heading and click **Add >>**.
- *To hide a visible column:* Under **Visible Columns**, select a heading and click **<< Remove**.
- *To move a visible column:* Under **Visible Columns**, select a heading and click either **Move Up** (to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

## Report Row Context Menu

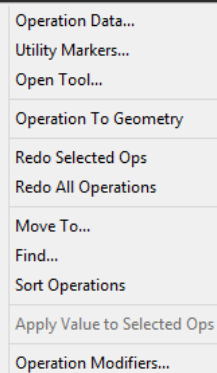
### Tool Manager



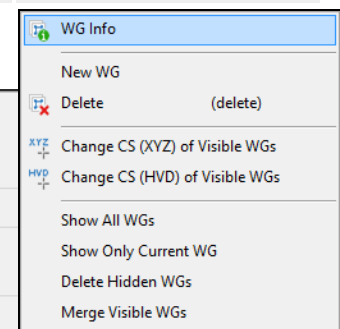
### Process Manager



### 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.



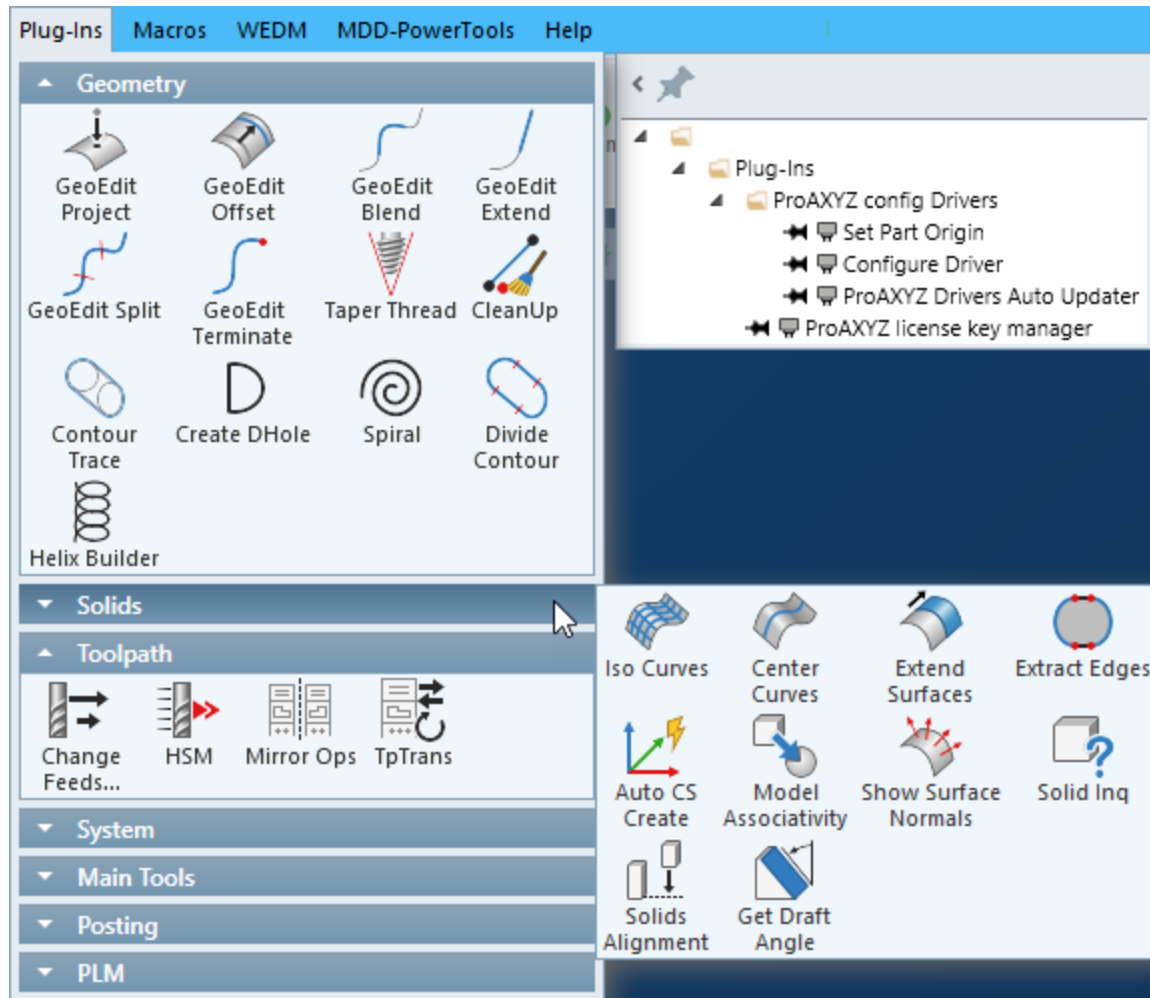
Please be aware that sorting operations can be risky. For example, you could potentially tell the system to tap a hole before it is drilled. Please be sure to review the results of the sort to ensure you get the results you want.

### 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



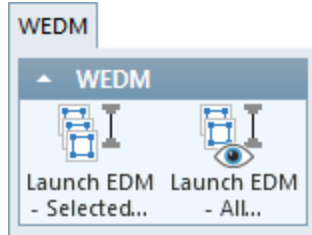
Plug-ins are modular functions that are built onto the system to provide enhanced functionality. One large advantage of plug-ins is that third parties can create software that integrates with GibbsCAM. Another strength is the rapid development of plug-ins, allowing clients' needs to be met quickly. Each 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 Wire EDM Menu

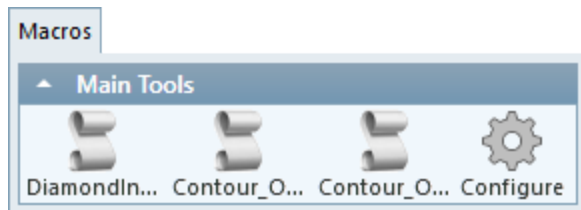


If the Wire EDM option is installed and available to you, the **WEDM** menu provides two options for starting the Wire EDM package:

- **Launch EDM - Selected Geometry**: Starts Wire EDM using only the geometry (2D lines, arcs, circles, and points) in the current selection set.
- **Launch EDM - All Visible Workgroups**: Starts Wire EDM using all geometry in all visible workgroups.

Complete reference information and instructions for using the Wire-EDM package are provided in the [Wire EDM](#) guide. This guide is provided with the installation media and is available for download from <https://online.gibbscam.com>.

## The Macros Menu

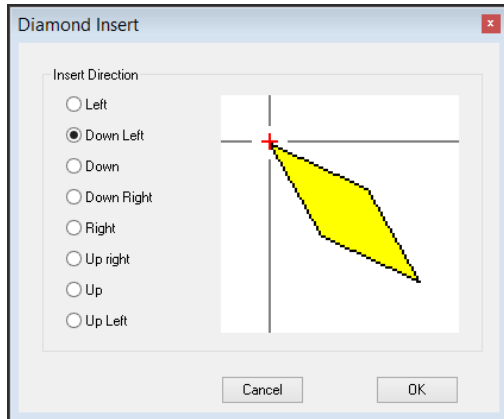


The **Macros** menu provides access to macros that have been installed on your machine's *global data* folder (such as `C:\ProgramData\CAMBRI0\GibbsCAM\<version>`). By default, one macro, **Diamond Insert**, is pre-installed to this folder. If you or your administrator have placed other macros under this folder, they will appear under the **Macros** menu.

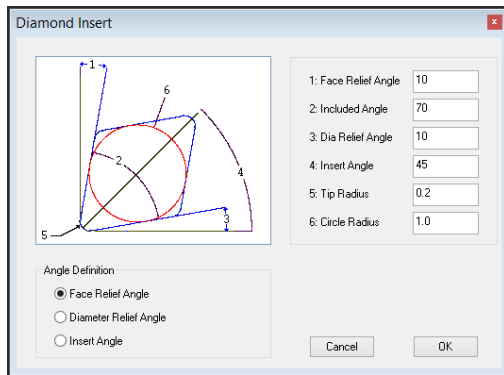
## Diamond Insert

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 96.

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



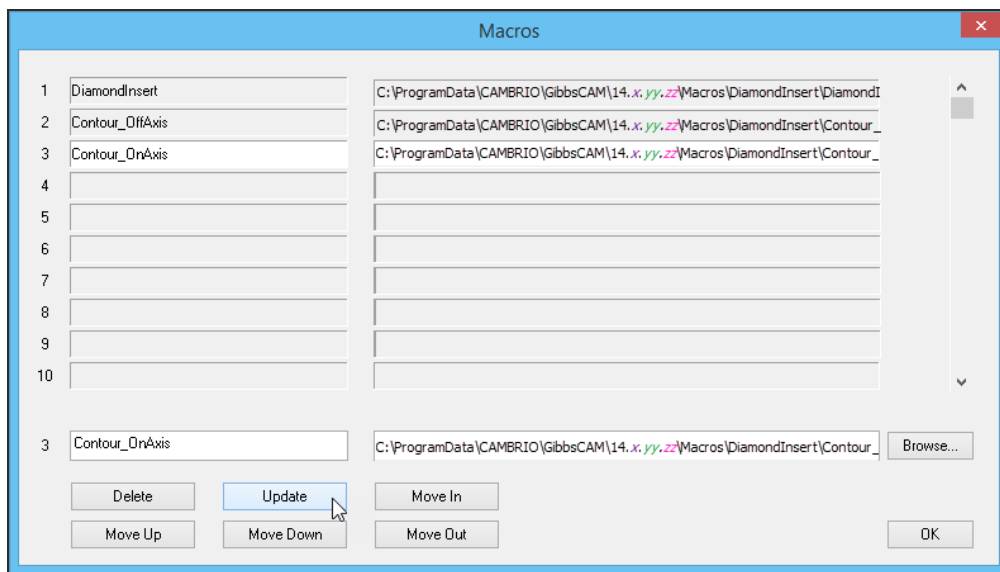
If you have macros in a previous release of GibbsCAM and you have not yet configured the Macros menu in your current release, you can migrate the old macros using the Migration Tool. If, instead, you *have* already configured your current Macros menu and you want to migrate old macros, see the instructions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

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\CAMBRIO\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.

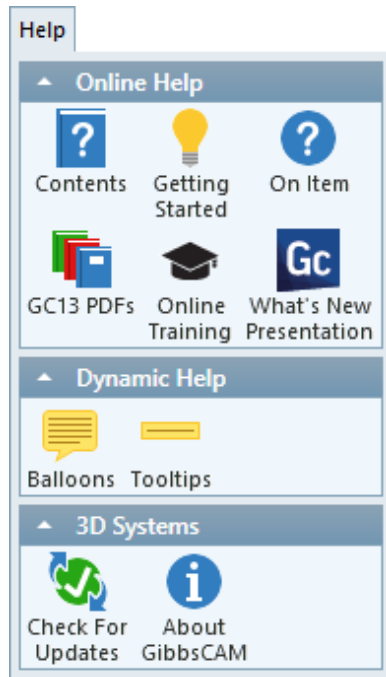
The **Macros** dialog lets you add and organize macros:



- 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](#)
- [CAMBRIO](#)
- [About the Online Help](#)

## Online Help

### **Contents**

Selecting this item will activate the system's online help. A window will open in which you can browse or search for information.

The screenshot shows the GibbsCAM online help system. On the left is a 'TOC' (Table of Contents) pane with a list of topics including 'Documentation for GibbsCAM', 'Text Conventions', 'Links to Websites', 'What's New in GibbsCAM', 'Getting Started', 'Common Reference', 'Geometry Creation', 'Mill', 'Lathe', 'Mill Turn', 'Features', 'Advanced CS', 'MTM', 'Solids Import', '2.5D Solids', 'SolidSurfacer', 'Tombstone Management System', 'Machine Simulation', 'Radial Milling', '5-Axis', '5-Axis MultiBlade', '5-Axis Porting', 'Wire EDM', 'Plug-Ins', 'Data Exchange', 'Basic Utility Operations', and 'VoluMill'. Below this list are buttons for 'TOC', 'Index', 'Search', and 'Favorites'. The main content area has a title bar '> Documentation for GibbsCAM' and a large heading 'Documentation for GibbsCAM'. Below the heading is a paragraph: 'To navigate through this online help system, you can use any of the following from the toolbar or navigator pane.' This is followed by a table:

Icon	Label	Function
	TOC	Displays the online table of contents, where books and topics are arranged hierarchically.
	Index	Displays the merged index, where you can search topics by keyword.
	Search	Displays the search box, where you can run a full-text-search against all topics.
	Favorites	Displays a list of topics and searches that you have previously designated as your favorites.

For details, see [About the Online Help](#).

### Getting Started

Selecting this option will launch Adobe Reader and open the PDF version of [Getting Started](#).

### On Item

This item is the same as clicking the On Item help button found on the GibbsCAM Floating Toolbar. Once active, the cursor changes to the On Item Help cursor . Clicking on a dialog, palette or window in GibbsCAM will then open the online help to the section that is relevant to what you clicked on. This is known as context-sensitive help.

### 14 PDFs

Selecting this option will launch Adobe Reader and open 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. Courses entitled “Whats New in GibbsCAM 14”, “Production Milling for GibbsCAM 14” and “Production Milling/Turning for GibbsCAM 2016 and Older Versions” are available free of charge. A subscription-based training program to learn more advanced options such as 2.5D Solids, 3D Machining and more is available.

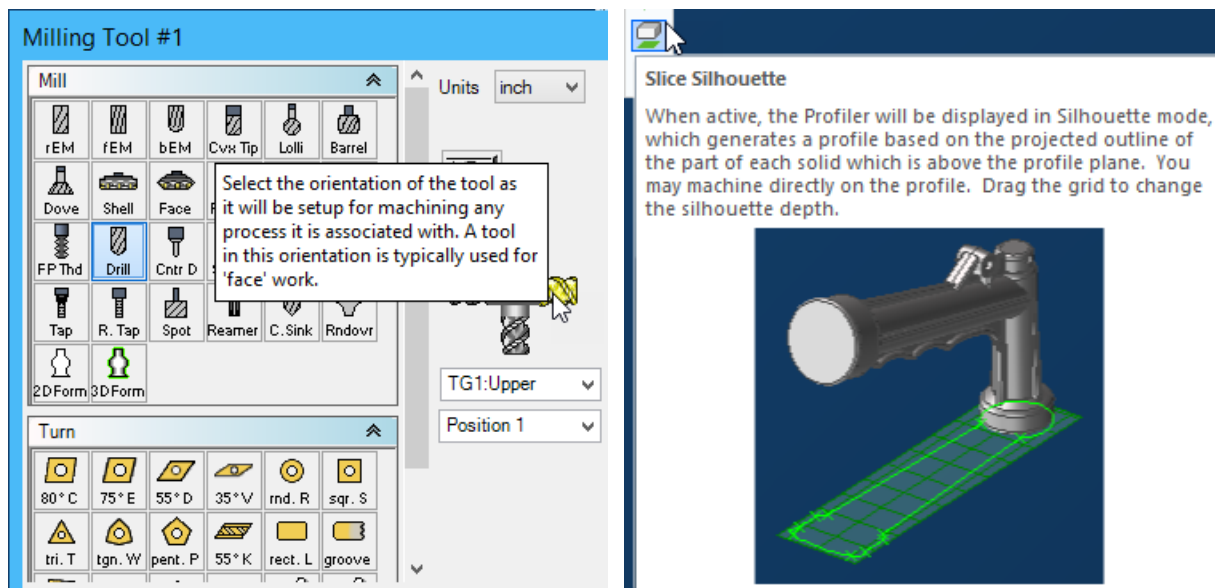
## Whats New Presentation

Opens a short video displaying highlights of the newest features added in the latest GibbsCAM version.

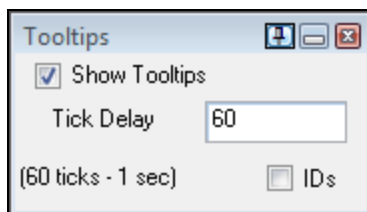
## Dynamic Help

### 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.

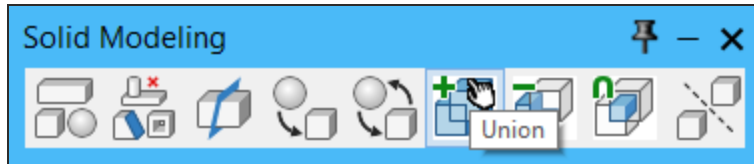


### Tooltips



This option brings up the **Tooltips** dialog. Tooltips will show the name of a button or item the mouse is held over. The **Tooltips** dialog allows the user to set whether tooltips are shown and to set the delay before tooltips are shown. The delay before a tooltip is displayed is set using the **Tick Delay** option. Each “tick” is 1/60th of a second. The **IDs** item is not of any real use to end-users, as it will display the identifying number of a dialog’s components, rather than a description. It is a development tool.

The image below provides an example of a tooltip. Here the cursor is 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.



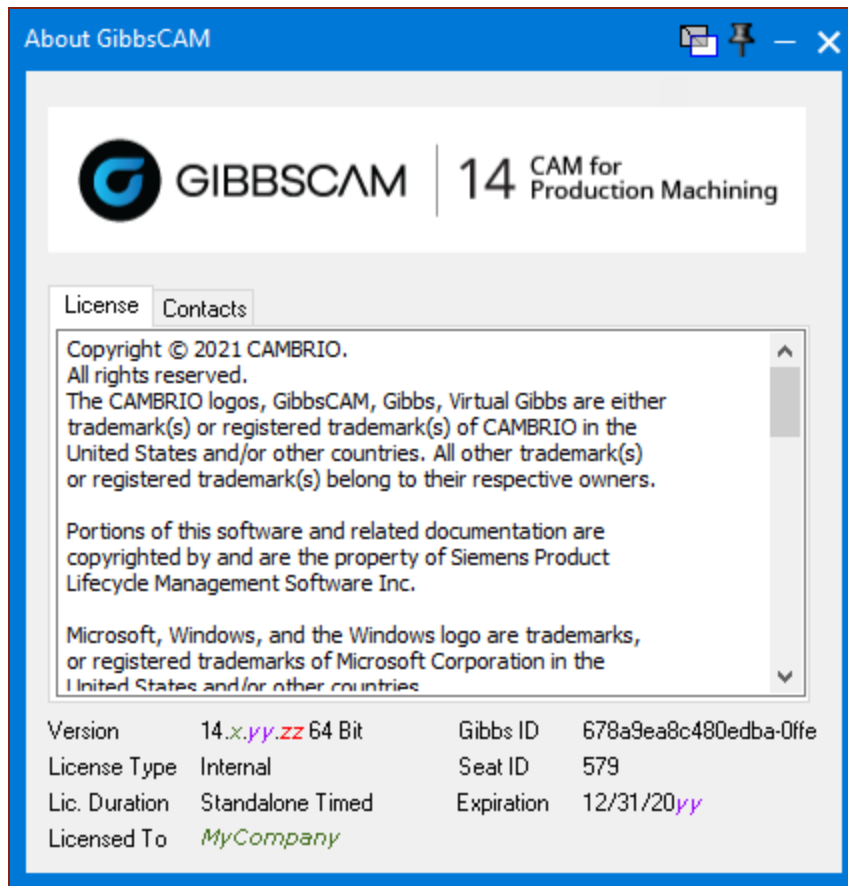
## CAMBRIO

### 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

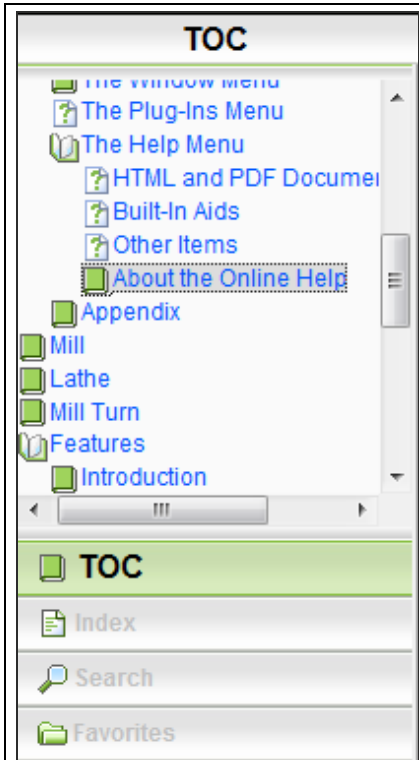



The **About** dialog provides information about your GibbsCAM software as well as phone numbers and e-mail contacts if you have questions or problems.

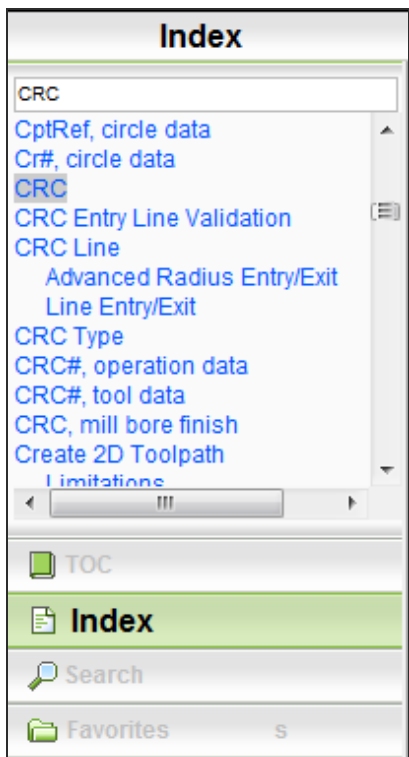
The About box displays your Gibbs ID, which can be copied and pasted directly from this dialog. If applicable, your HASP key expiration is also displayed here.


## About the Online Help

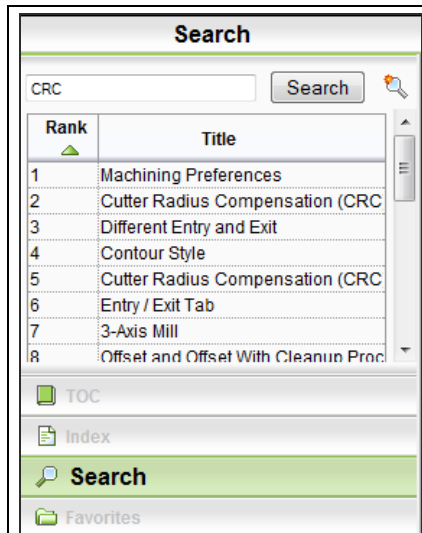
The system provides several navigation panes for working with online help:




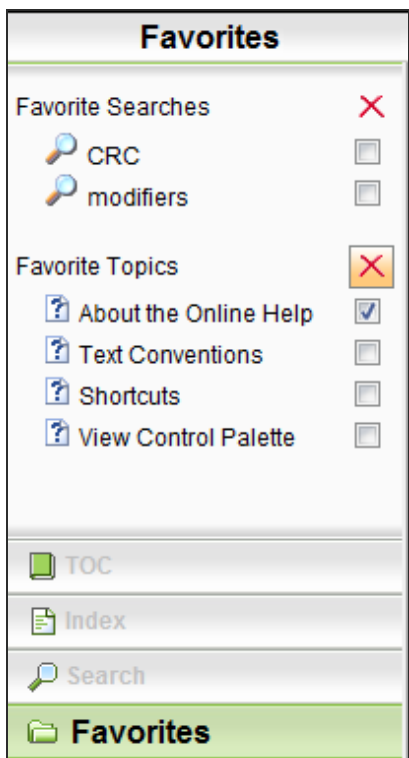
The  icon on the Help toolbar activates the TOC pane, which presents a hierarchical list of help topics organized into folders and subfolders like a table of contents.







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 -]**.

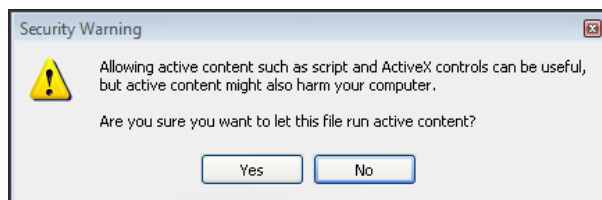
Generally the size of the text in the Help window text is determined by the size of the default text set for Windows.

### Allowing Blocked Content

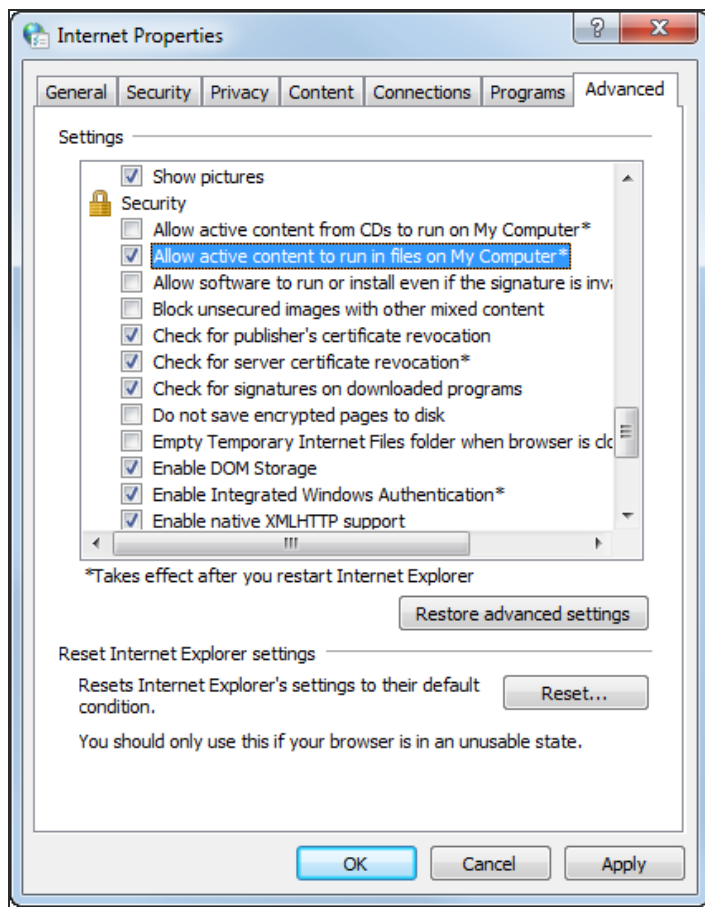
 To help protect your security, Internet Explorer has restricted this file from showing active content that could access your computer. Click here for options... 

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...**

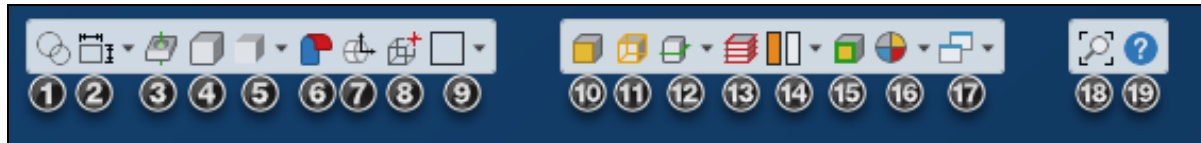


- You will receive a Security Warning. Click **Yes** to continue.



- 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.

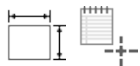
# Floating Toolbar



- |  |  |   |
|--|--|---|
| 1. <a href="#">Show Geometry</a>       | 7. <a href="#">Wrap WGs</a>                | 13. <a href="#">Toggle Hidden-line Toolpath</a>       |
| 2. <a href="#">Show Dimensions</a>     | 8. <a href="#">Show Stock &amp; Origin</a> | 14. <a href="#">Toolpath color mode</a>               |
| 3. <a href="#">Show Hole Features</a>  | 9. <a href="#">Show CS</a>                 | 15. <a href="#">Toggle Pre-Selection Highlighting</a> |
| 4. <a href="#">Show Solids</a>         | 10. <a href="#">Face Selection</a>         | 16. <a href="#">Select Color Display mode</a>         |
| 5. <a href="#">Render/Wireframe</a>    | 11. <a href="#">Edge Selection</a>         | 17. <a href="#">Viewport Configuration Manager</a>    |
| 6. <a href="#">Indicate Sheet Side</a> | 12. <a href="#">Toggle Profiler</a>        | 18. <a href="#">Unzoom</a>                            |
|  |  | 19. <a href="#">Get Help on Item</a>                  |

## Show Geometry

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



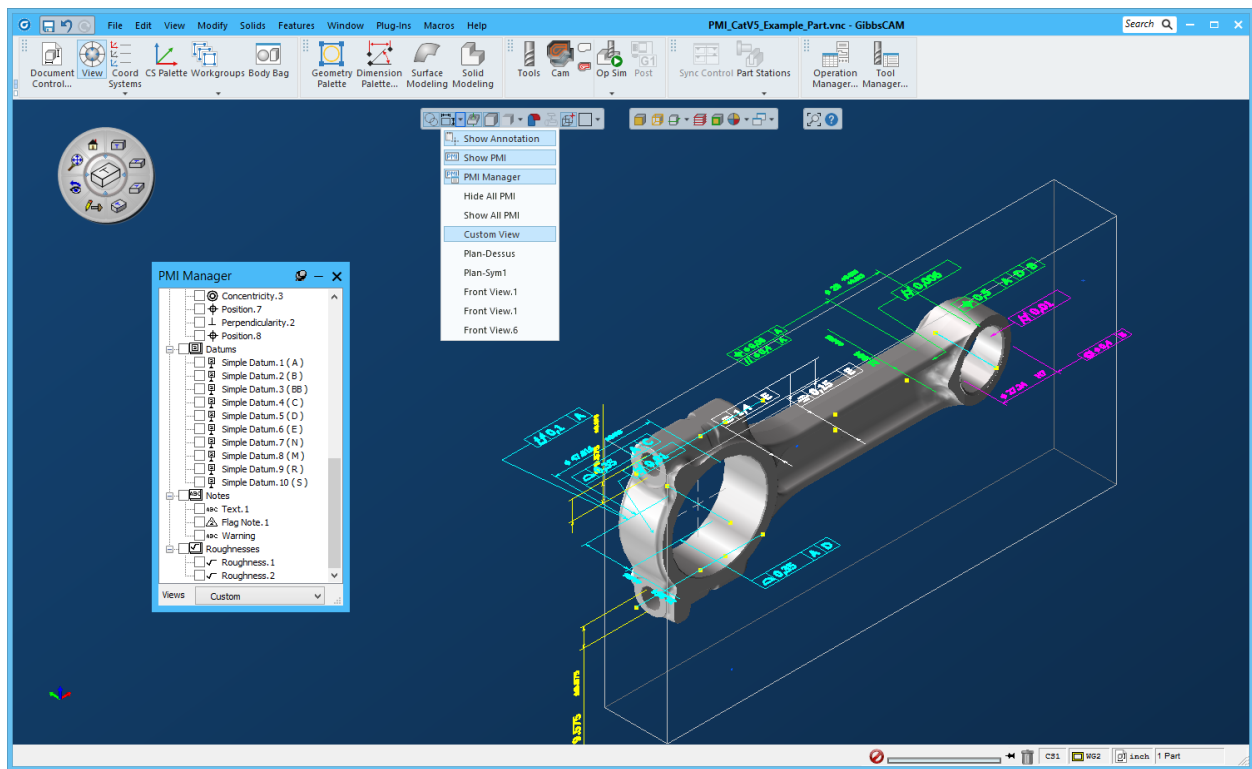
## Show Dimensions/Annotations

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>

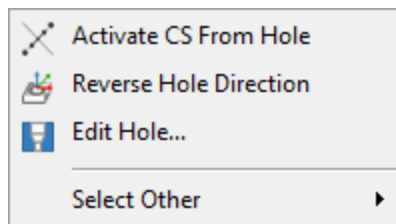
When checked, PMI is displayed using the named PMI view.



### 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 186](#).) 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 shown below.




**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**

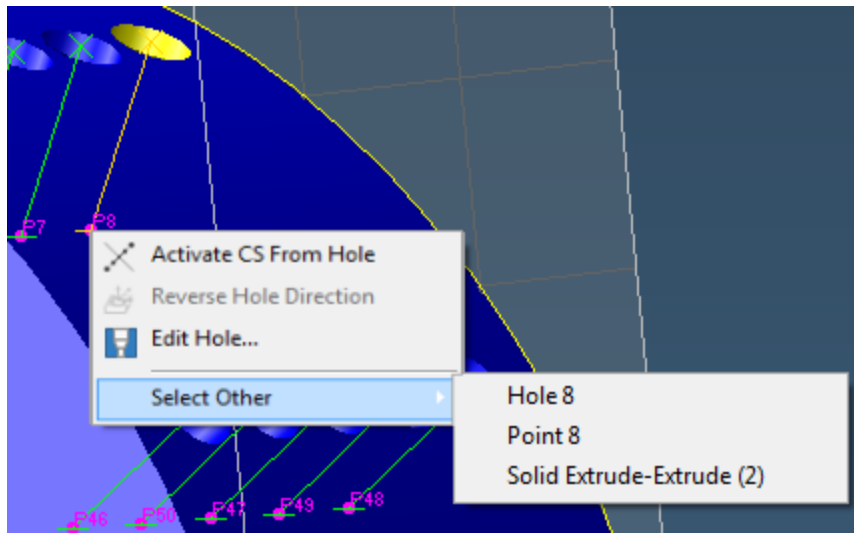
This is not available for blind holes and will be greyed out.

**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.




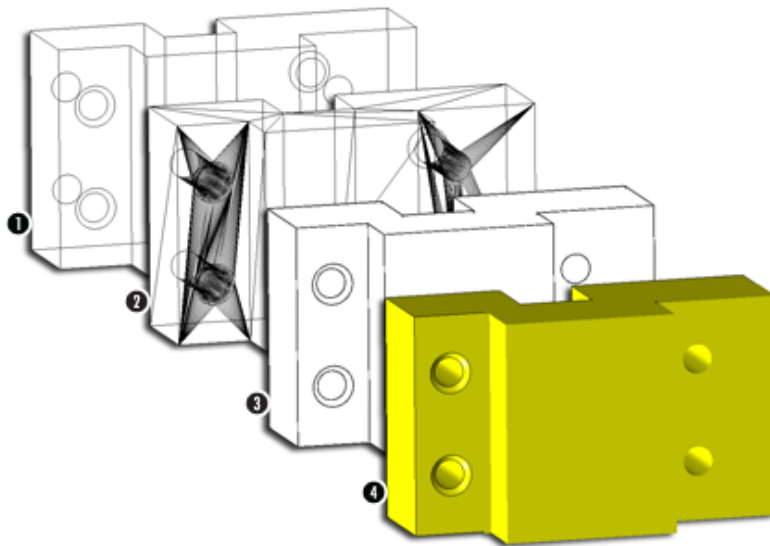
Render Solid & Edges renders the bodies but shows the edges of the bodies.



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



## 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 only available with the **Solids Import** or **SolidSurfacer** options.



## Wrap WGs

When 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.



## Show CS

This button controls the visibility of coordinate systems and the plane that fills it. This button has four states.



Hide CS Grid. The grid and plane are both hidden.



CS grid + plane, will show 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 will disable the grid but still show a shaded plane, representing the coordinate system.



Grid only shows the grid but 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 depressed (ON), the edges of all solids and sheets in the workspace are displayed. These edges can be selected. An edge is the curve or spline between two faces. In order



to select edges for such functions as blending, geometry extraction, and body unstitching, you must be in Edge Selection mode. Also, when using the stitching function, Edge Selection mode is useful for viewing the external edges of a model. 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. This item is available with any of the solids packages.



## Profiler

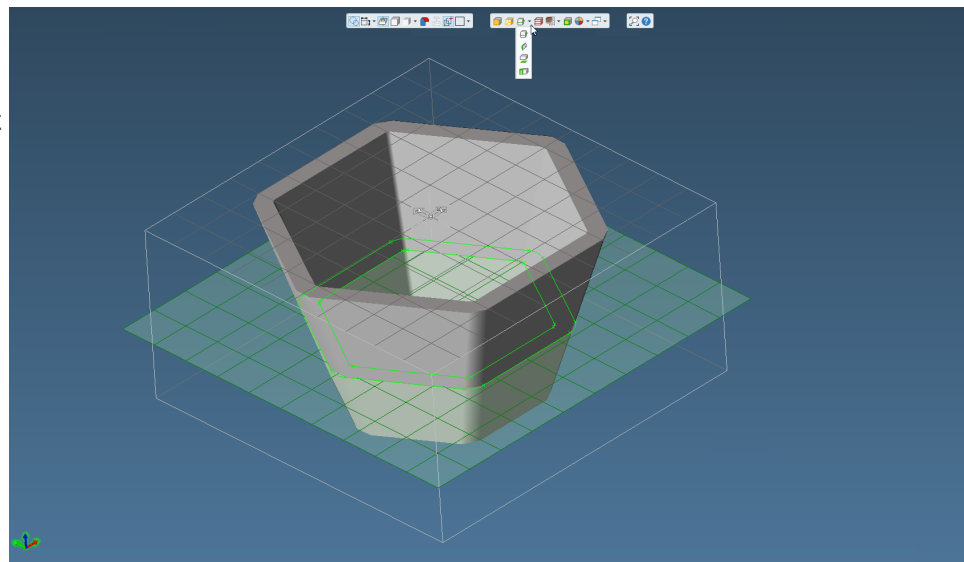
The Profiler is a moveable cross-section tool that can be toggled off or on in any of four different modes.


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.

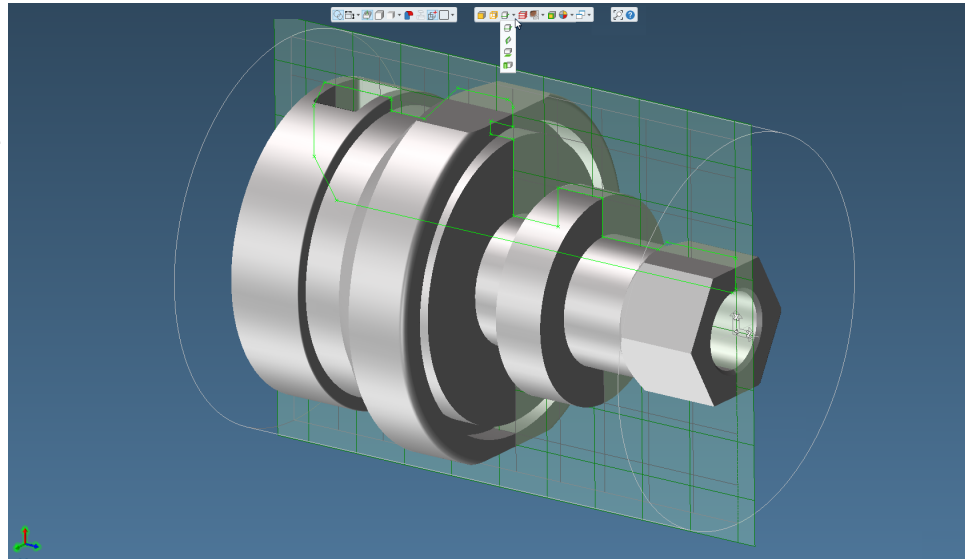
In **Slice Plane** mode, the Profiler is planar to the current CS. To dynamically display the part's cross-section geometry, drag the slicing plane up or down along the depth axis.


Sample part:  
"Profiler\_Slice-Plane\_Slice-Silhouette.vnc"



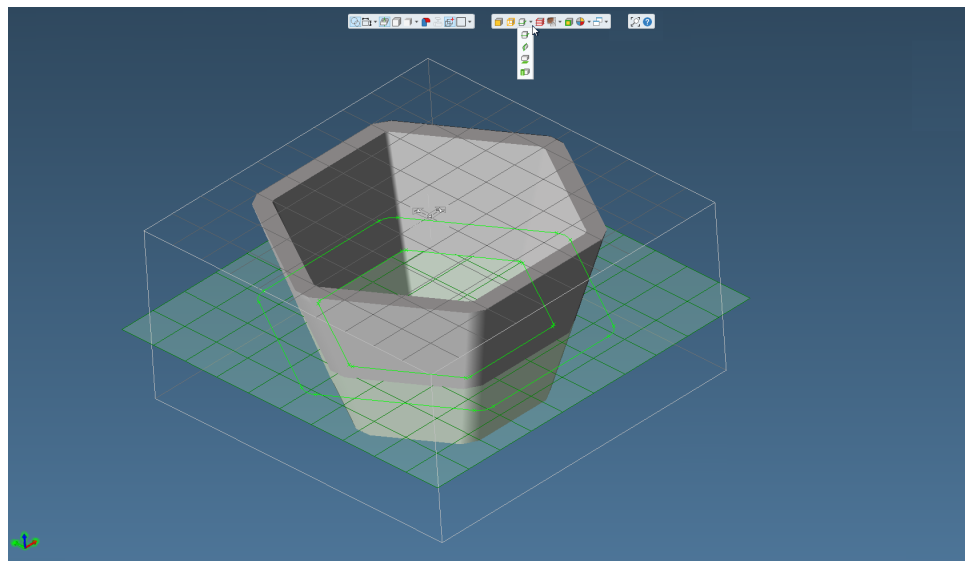
In  Slice Spun Body mode, the Profiler is planar to the current CS, and slices the outline of the body that would result from spinning all elements around the rotary axis.


Sample part:  
["Profiler\\_Slice-SpunBody\\_Slice-Cylinder.vnc"](#)



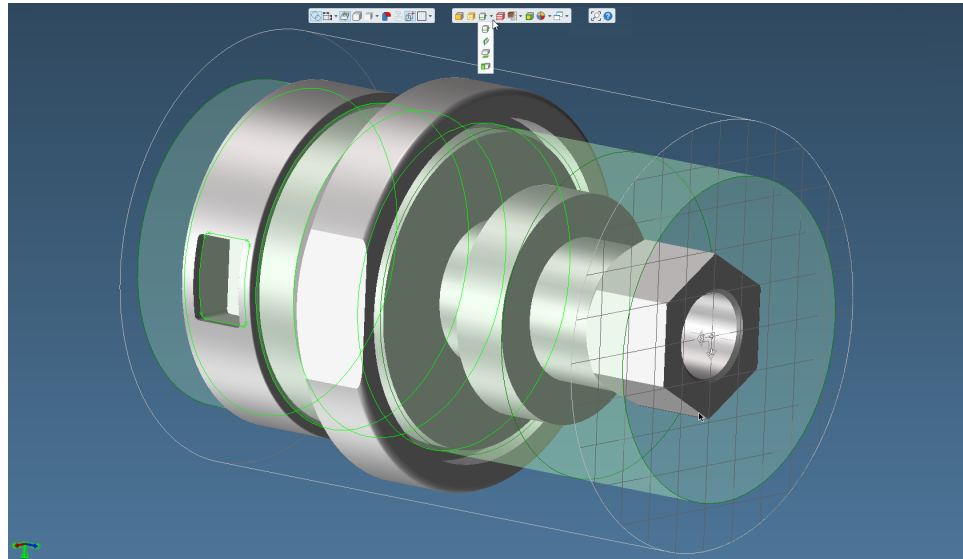
In  Slice Silhouette mode, the Profiler is planar to the current CS, and displays the outline of the "shadow" cast by all bodies from rays of light normal to the plane and shining down on it. Each body (even a multi-lump body) casts a single shadow with a single outline. Multiple bodies cast separate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

Sample part:  
["Profiler\\_Slice-Plane\\_Slice-Silhouette.vnc"](#)



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.

Sample part:  
 "Profiler\_Slice-SpunBody\_Slice-Cylinder.vnc"

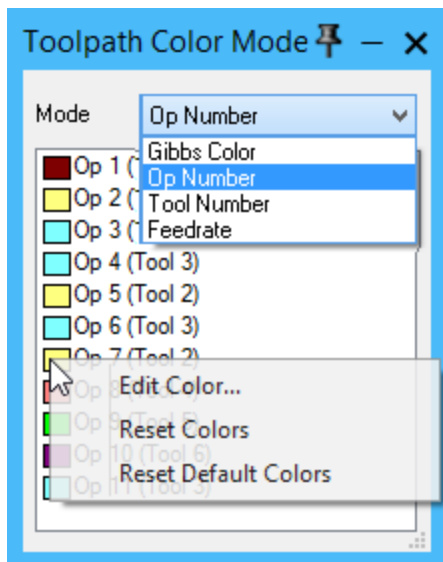


Profiler modes: See sample parts "Profiler\_Slice-Plane\_Slice-Silhouette.vnc" and "Profiler\_Slice-SpunBody\_Slice-Cylinder.vnc"

## Hidden-Line Toolpath

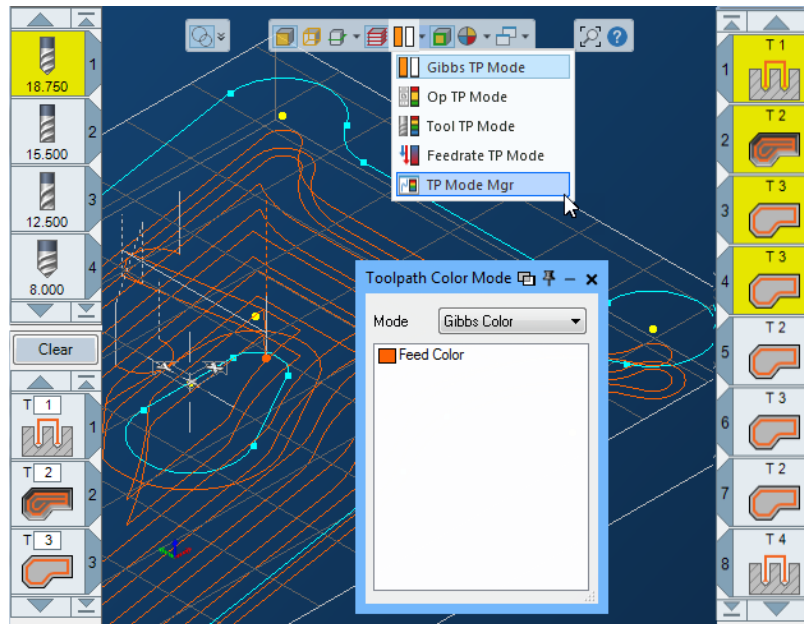
This is a mode for displaying the toolpath with hidden-line removal activated. When on, toolpath that lies behind solid objects is not shown.

## Toolpath color Mode



In addition to the standard red GibbsCAM toolpath color, Toolpath can be displayed with the Operation numbers, Tool numbers or Feedrate highlighted in user-editable colors.

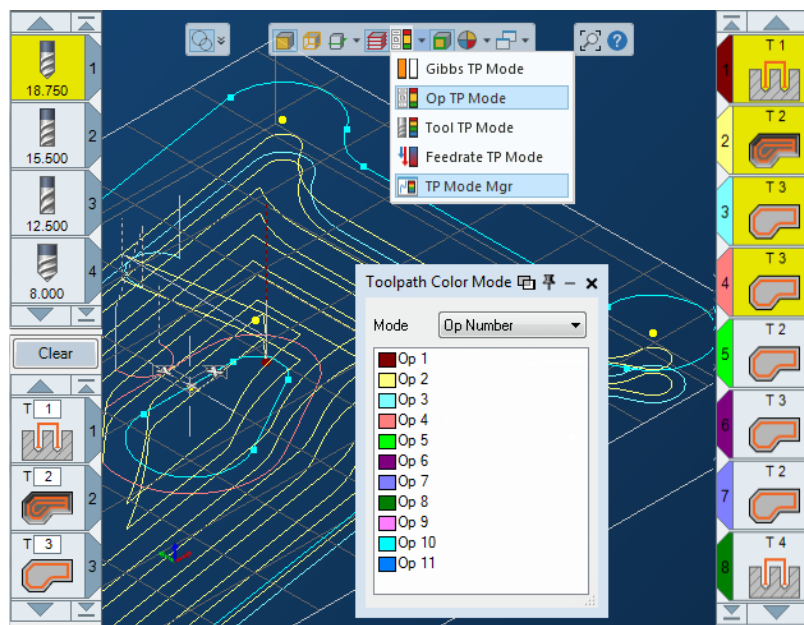
Gibbs Toolpath Color Mode



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.

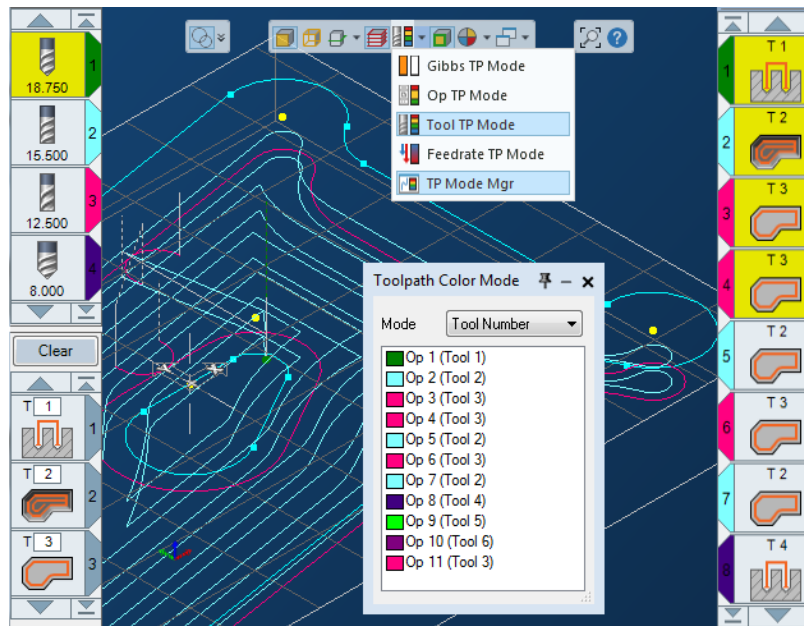
Op Number Toolpath Color Mode



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.

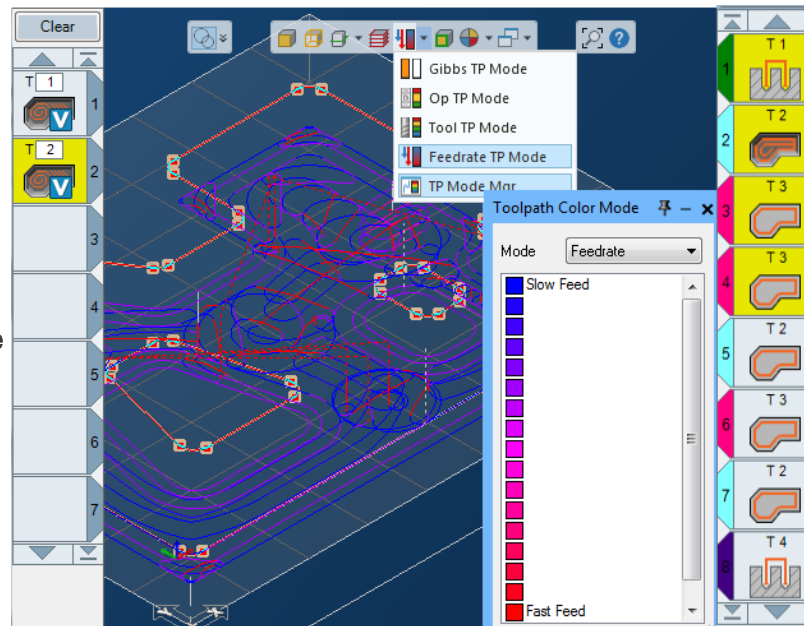
**Tool Number Toolpath Color Mode**



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 blue “slow feed” to hot red “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

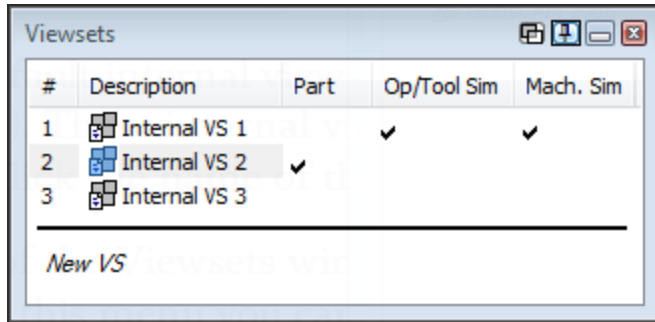
This mode highlights objects on the display as if they were selected whenever the cursor passes over them. The preselection face and edge colors, halo transparency, and thickness can all be set by the user in preferences on the Color Tab.

## Color Mode

This mode determines the color mode in effect for the workspace if Feature Color or one or more User Colors have been defined. Hover over the dropdown button to display all modes that can be selected.

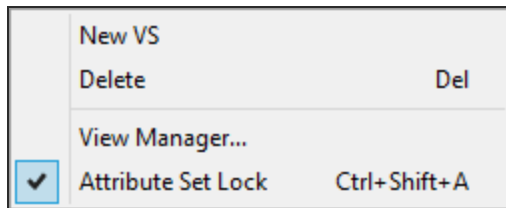
## Viewport Manager

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.

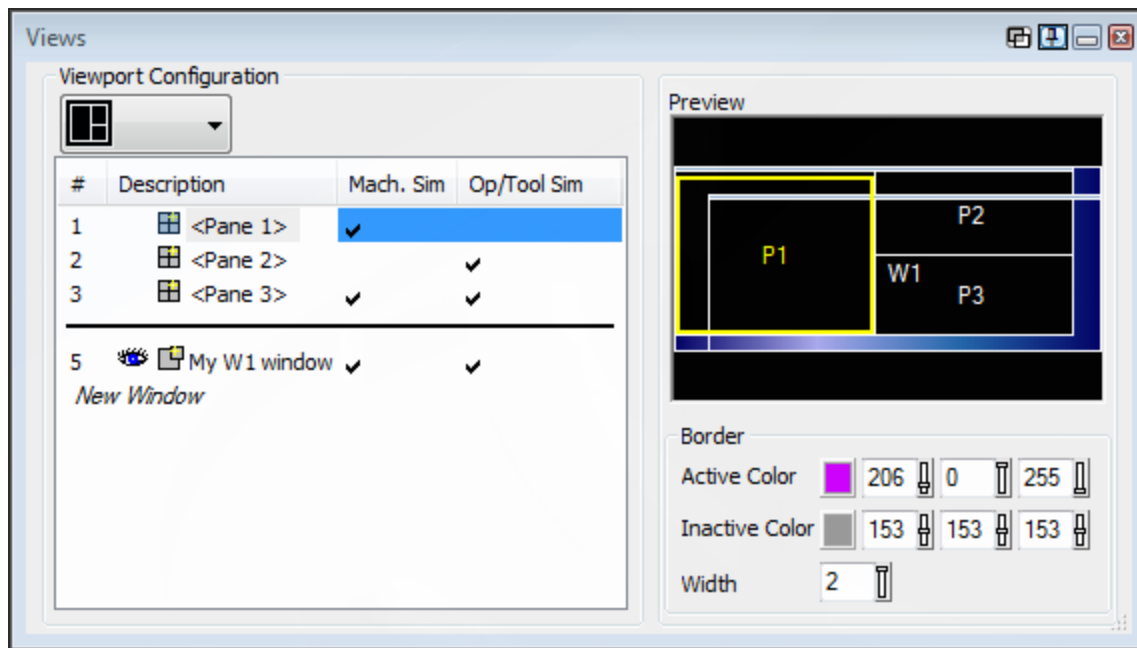
Right-clicking the title bar of the **Viewsets** window, or right-clicking the name of a viewset, displays a context menu. From this menu you can create a new viewset, delete a viewset, access View Manager, or lock the attribute set.



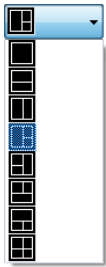
If Attribute Set Lock is checked, any changes made (turning on edge selection, render wireframe, etc.) will affect ALL viewports, active or inactive. If Attribute Set Lock is unchecked, changes will affect only the active viewport.

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

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



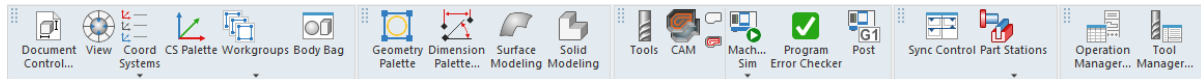
Online Help opens to display the help topic for that the selected item. As soon as you click on an item, the cursor switches back to its normal state.



# 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](#))

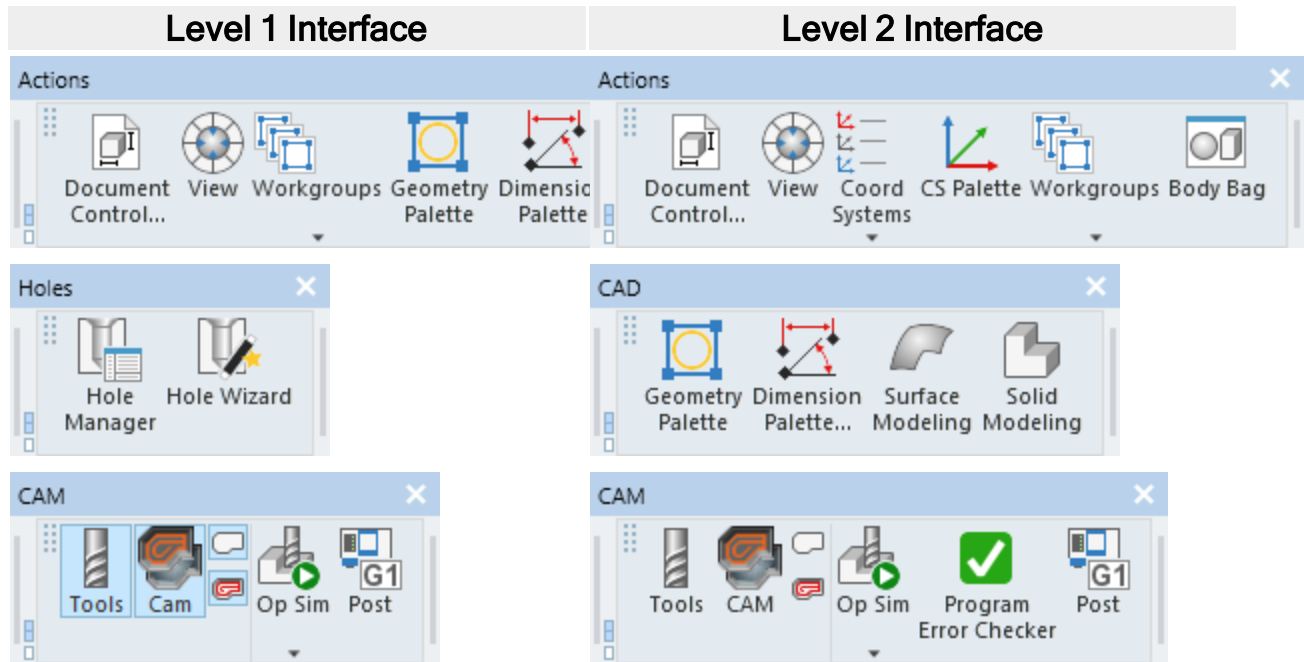
The standard display is as shown below:

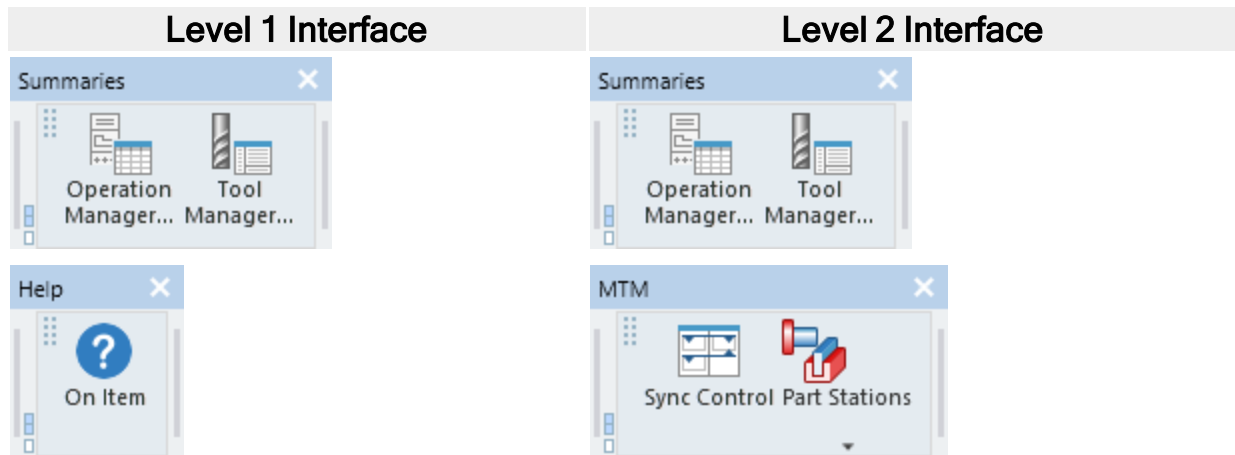


The toolbar can be minimized by using the small box controls situated on the left of the menu. (In the example below the command text has also been removed).

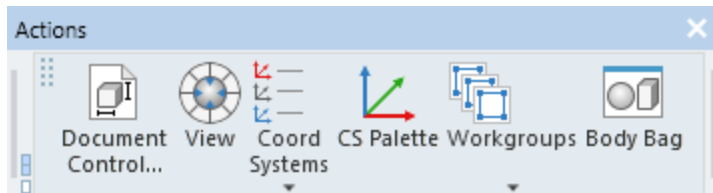


The contents of the default toolbars change according to the current Interface Level set in the File>Preferences>Interface Tab.



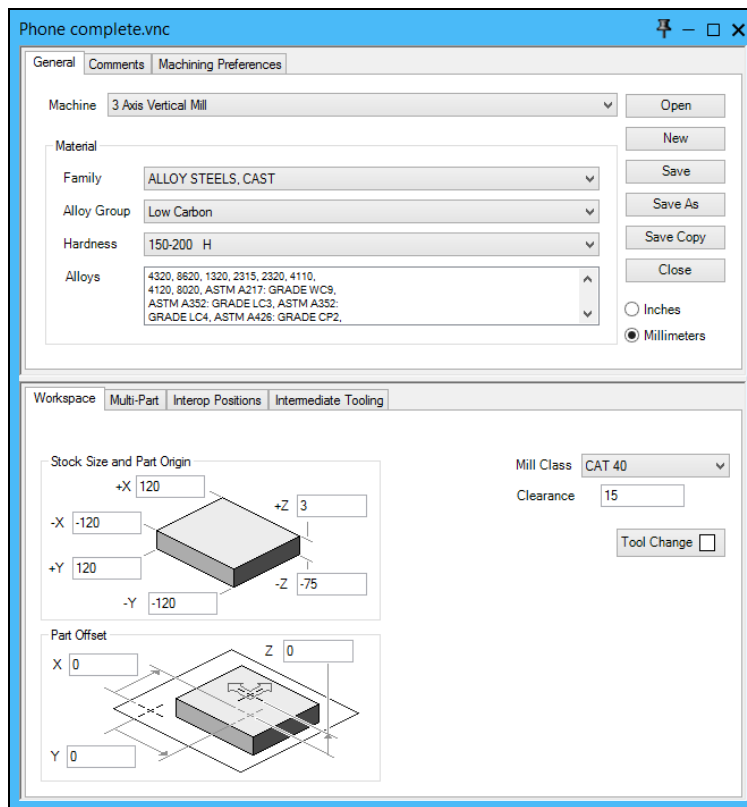


## Actions Command Toolbar Group



### 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 operate 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).

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

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

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


  This key combination rotates the part to the left by about 10%; east moves toward you and west 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**.

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

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

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


  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.

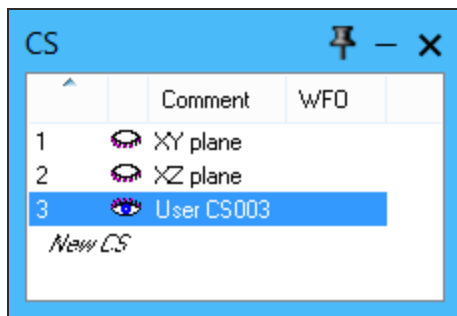
**Keyboard.** You can zoom in and out of the part is by using the **Ctrl** key with the **+** or **-** keys.

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

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

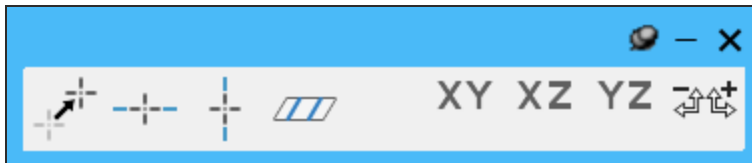
## CS List

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.



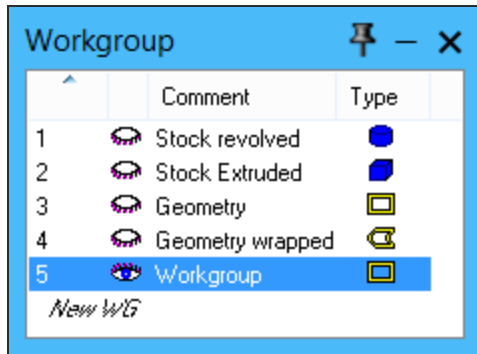
## CS Palette

This activates the Coordinate System Palette. For more information on Coordinate Systems see the [Advanced CS](#) guide.



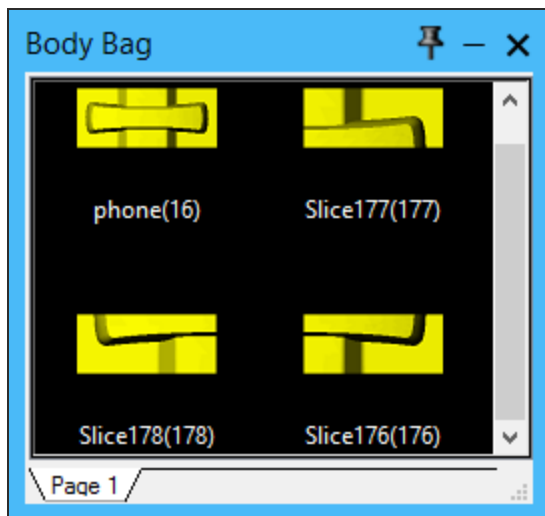
## Workgroup

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.

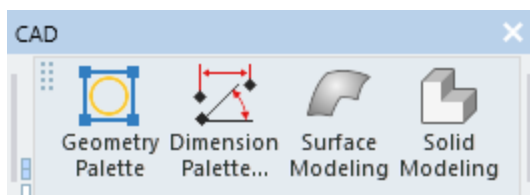


## Body Bag

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.



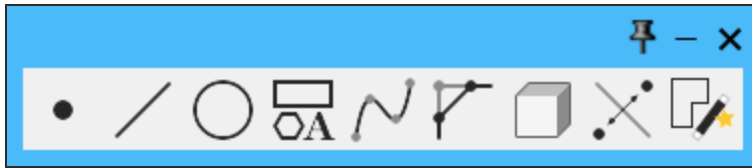
## CAD Command Toolbar



## Geometry Palette

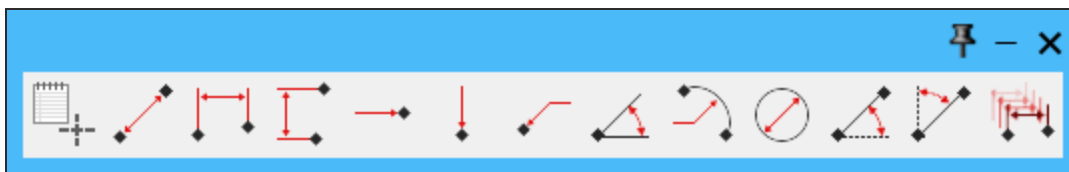


Accesses the main Geometry creation palette. For more information, see the [Geometry Creation](#) guide.



## Dimension Palette

This opens the Dimensioning Palette. For more information, see the "Dimensioning" section in the [Geometry Creation](#) guide.



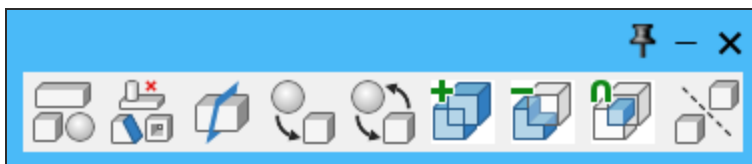
## Surface Modeling

This activates 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.

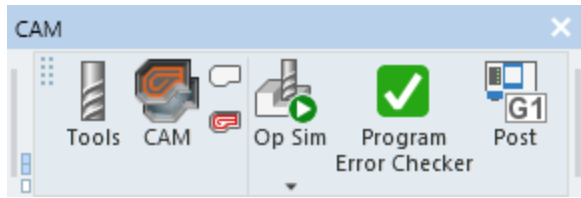


## Solid Modeling

This activates the main Solid Modeling palette. For more information on Solid Modeling see the "Solid Modeling" section of the [SolidSurfacer](#) or [2.5D Solids](#) guides.



# CAM Command Toolbar

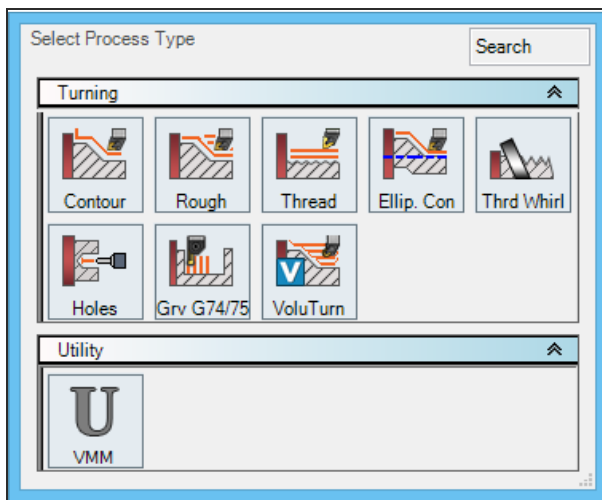


## Tools

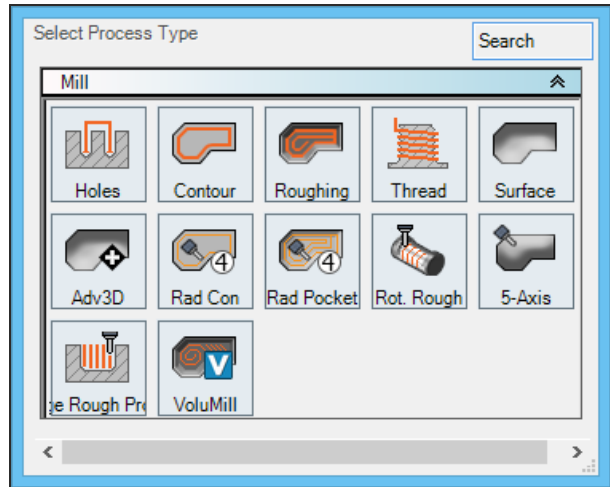
This button activates the Tool List on the top left of the screen, containing the tiles that you use to define tools. For more information see "Tool Creation" in the [Getting Started](#) guide.

## 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

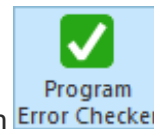
When you click the CAM button, both the Process and Operation Tile lists are displayed. The  Processes and  Operations buttons will toggle the individual lists on and off.

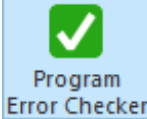


## Cut Part Rendering and Simulation

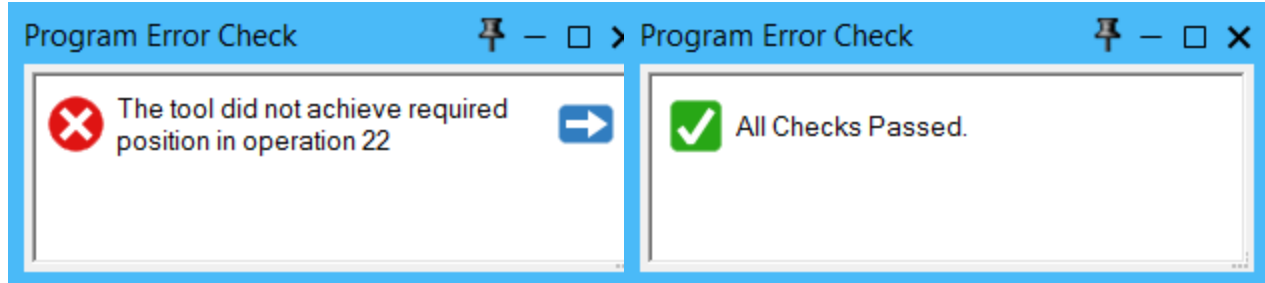
For more information see the section on Simulation [Cut Part Rendering and Simulation](#):

## 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:

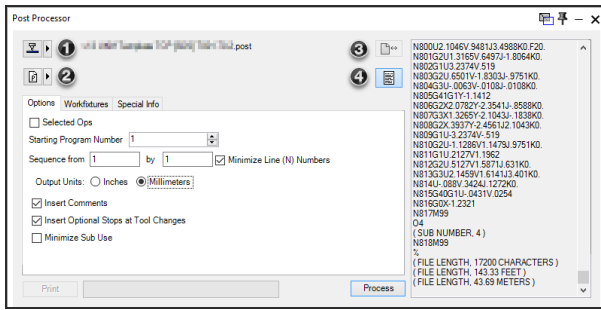
To run Program Error Check, click the button. The system will then test the current \*.vnc file for errors and incompatibilities and displays any problems found, or else the message “All Checks Passed.”



## Post Processor dialog

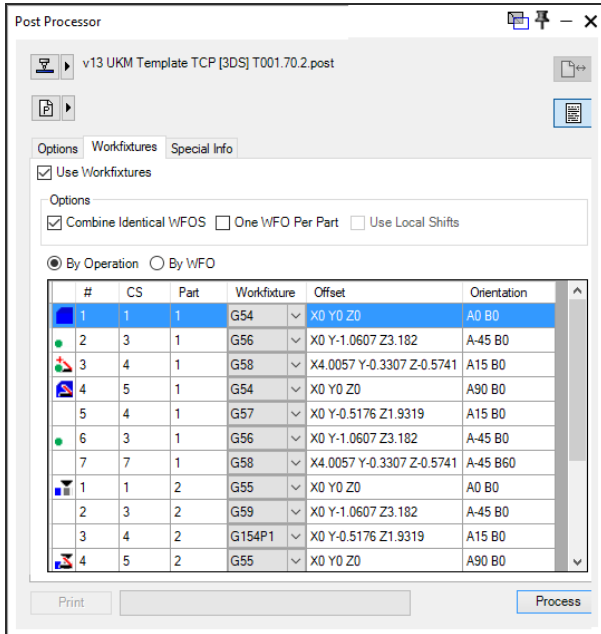
The Post Processor button in the Command Toolbar becomes active once machining operations have been created in a file. Clicking the Post Processor command in the CAM command menu will display the Post Processor dialog. This dialog allows you to select a Post Processor, an .NCF

Program name, and the format of the output. The Document Control dialog  in the Actions command box is used to set up Multiple Parts or TMS, if available.




For details, see [Options tab](#)

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



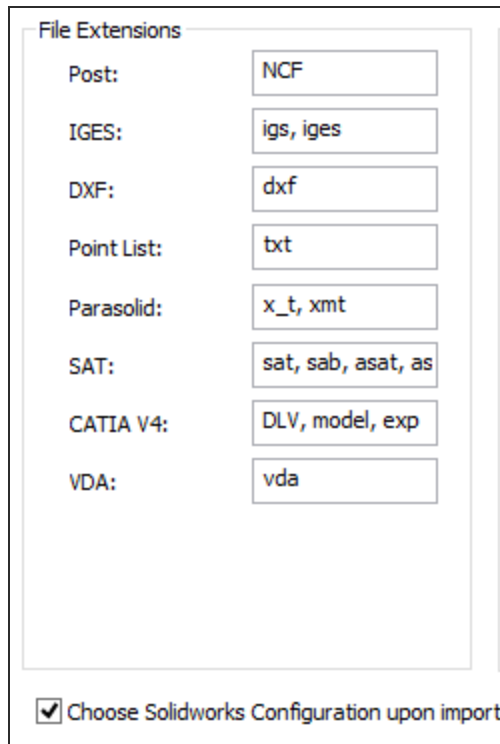
For details, see [Workfixtures tab](#)

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 allows the user to stop the scrolling of the output as it is being processed. The Print button will be 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 will appear that allows the user to access the directory or folder where the post processors are stored in the system.

The actual file names of the post processors are different depending on the operating system. However, when selecting a post processor, the full name (including the Control and Machine) will be displayed in the Open dialog.



Category	File Extensions
Post:	NCF
IGES:	igs, iges
DXF:	dxf
Point List:	txt
Parasolid:	x_t, xmt
SAT:	sat, sab, asat, as
CATIA V4:	DLV, model, exp
VDA:	vda

Choose Solidworks Configuration upon import

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.

### Selected Ops

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 Selected Ops and the same operations when posted together with others.

**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**

Items in the second tab of the **Post Processor** dialog let you do the following.

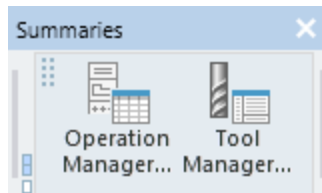
- Distinguish at a glance a part station from a multi-part part instance, with markers for whether the WFO is manually changed, the op is shifted from the WFO origin, op oriented differently from the WFO definition, or any combination, as shown in this table.

	No shift, same orientation	Op shifted from WFO	Op oriented differently from WFO	Shifted and oriented
Part Station				
Multi-Part				
<i>(marker: manual change)</i>				
Part Station, manually changed				
Multi-Part, manually changed				

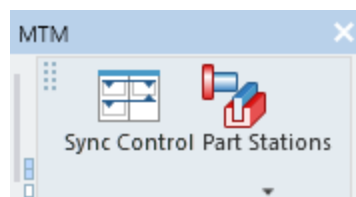
- Assign a WFO per op rather than per CS (including multi-part instances).
- See the WFO list before you post.
- Tell the system how to handle cases where a WFO does not match a CS.

If your post is from before GibbsCAM 12, the interface simply lists CS's and WFOs.

## Summaries Command Toolbar

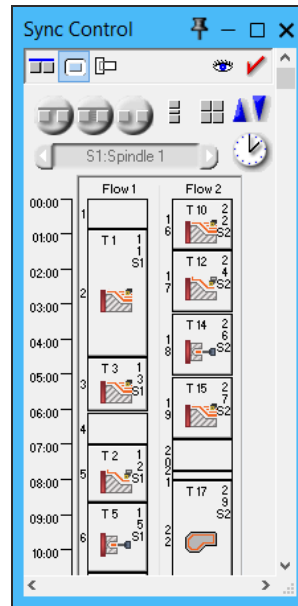


## MTM Command Toolbar



### 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.



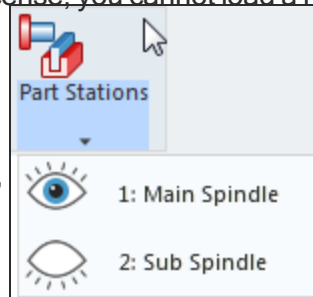
**In-Op Simulation** display is part-centric toolpath rendering, showing the part with material cut away by the operations.



**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.

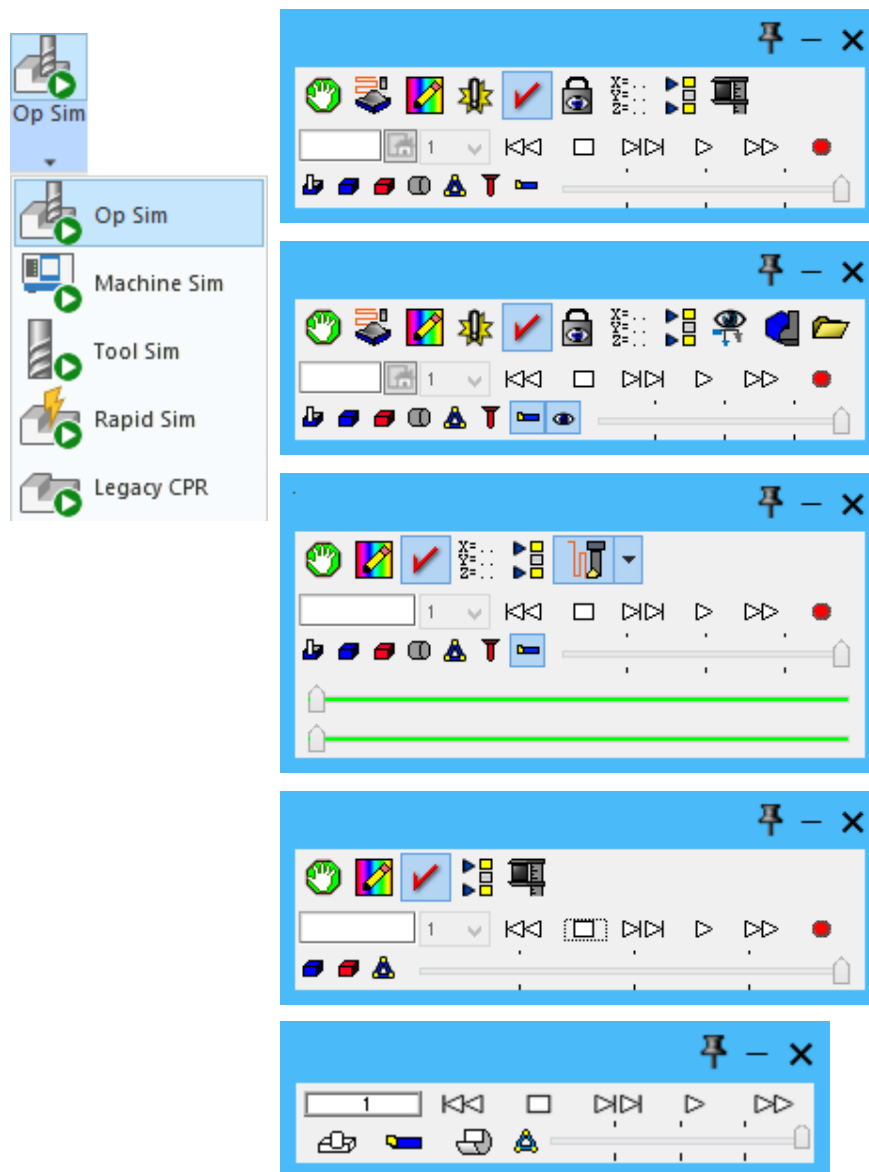
All operations are rendered in their current order. Selected operations render in shades of yellow, and all others render in shades of grey. Shades of red appear when non-cutting tool surfaces or rapid moves hit the material. Selected tools appear in shades of yellow. Deselected tools appear in shades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

- [Cut Part Rendering and Simulation Options](#)
- [“Playback Controls” on page 139](#)

- “CPR/Rendering Visibility Controls” on page 141
- “Simulation Control Icons” on page 144
- “Simulation Context Menus” on page 151
- “Settings for Op Sim, Tool Sim, and Machine Sim” on page 25

## 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. Current Display
2. Rewind
3. Stop
4. Step Forward
5. Play
6. Next Operation
7. Record (not Legacy CPR)

8. Speed control

Note: If you click the Stop or Step Forward 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 Play button to continue rendering.

## Current Display

This box displays the number of the current operation being rendered or the current runtime.

## Rewind

Click this button to return playback to the first operation.

## Stop

Click this button to pause rendering.

## Step Forward

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

## Play

Click the Play 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 Step Forward buttons pause the rendering. If you click the Play button during rendering, the rendering pauses. The Next Operation 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 Play button to resume.

## Next Operation

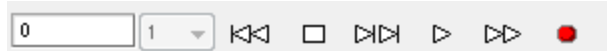
If rendering is in progress, clicking Next Operation finishes rendering the current feature, skips the remaining features for that operation, and continues rendering the next operation. If rendering is paused, clicking Next Operation advances the operation number in the Current Display box to the next operation. Click the Play 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
<p>The Simulation toolbar contains various icons for visibility control and machine display. Callouts 1 through 10 point to specific icons: 1 (Tool Visibility Control), 2 (Stock Visibility Control), 3 (Fixture Visibility Control), 4 (Lathe Stock Cutaway), 5 (Run time or Number of Operation Display), 6 (Show Rapid Tool), 7 (Show Tool Holders), 8 (Show Machine), 9 (Run time or Number of Operation Display), and 10 (Active Flow Number).</p>	<p>The Legacy CPR toolbar includes icons for overlay geometry and tool visibility. Callouts 1 through 5 point to: 1 (Tool Visibility Control), 2 (Stock Visibility Control), 3 (Fixture Visibility Control), 4 (Lathe Stock Cutaway), and 5 (Run time or Number of Operation Display).</p>	<p>The Rapid CPR toolbar features icons for EOP (End of Part) and visibility control. Callouts 1 through 5 point to: 1 (Run time or Number of Operation Display), 2 (Stock Visibility Control), 3 (Fixture Visibility Control), 4 (Lathe Stock Cutaway), and 5 (Run time or Number of Operation Display).</p>
<ol style="list-style-type: none"> <li>1. Tool Visibility Control</li> <li>2. Stock Visibility Control</li> <li>3. Fixture Visibility Control</li> <li>4. Lathe Stock Cutaway</li> </ol>	<ol style="list-style-type: none"> <li>5. Overlay Geometry</li> <li>6. Show Rapid Tool</li> <li>7. Show Tool Holders</li> <li>8. Show Machine</li> </ol>	<ol style="list-style-type: none"> <li>9. Run time or Number of Operation Display</li> <li>10. Active Flow Number (For MTM machines)</li> </ol>

## 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 solid 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). Rapii render more smoothly, and the part renders faster with this choice selected.

 /  Transparent Tool:

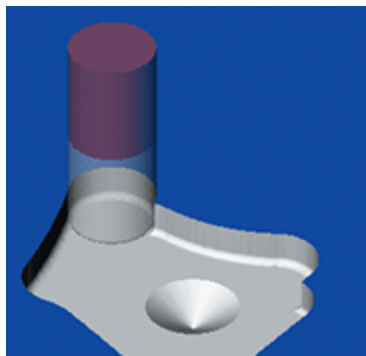
If this button is selected, transparent tools appear during the rendering process.

 /  Visible Tool:

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



Invisible Tool



Transparent Tool

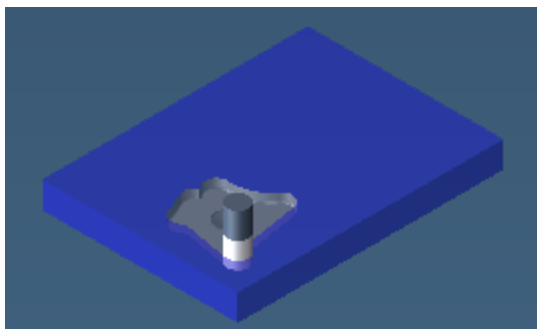


Visible Tool

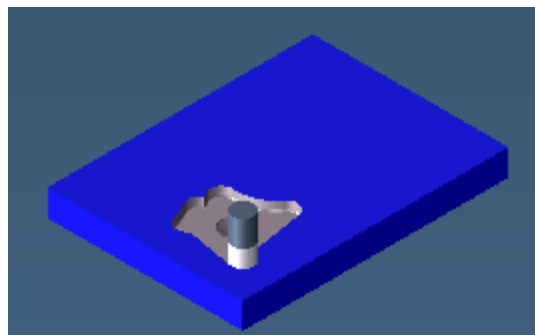
### Stock Visibility Control

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

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



Translucent Stock

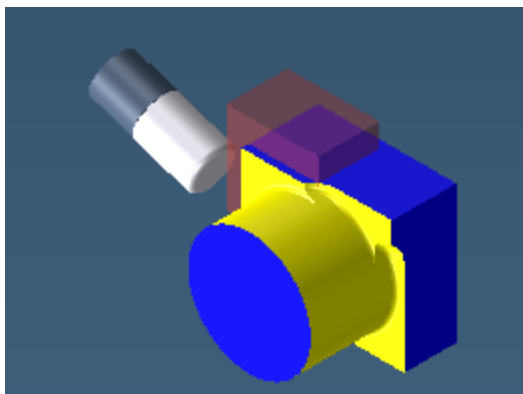


Solid Stock

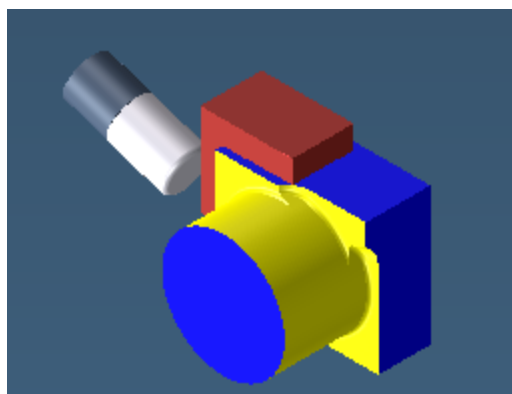
### Fixture Visibility Control

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

For Rendering, fixtures can either be displayed as Translucent  or Solid .






Translucent Fixture

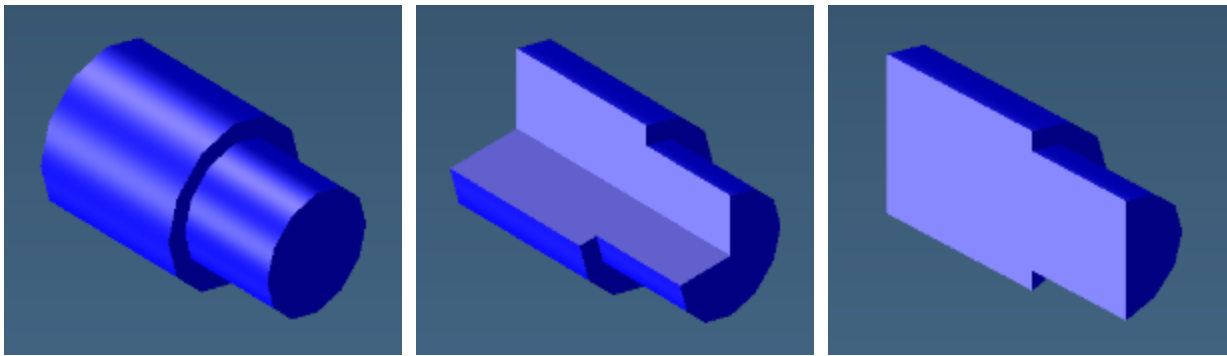


Solid Fixture

### 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

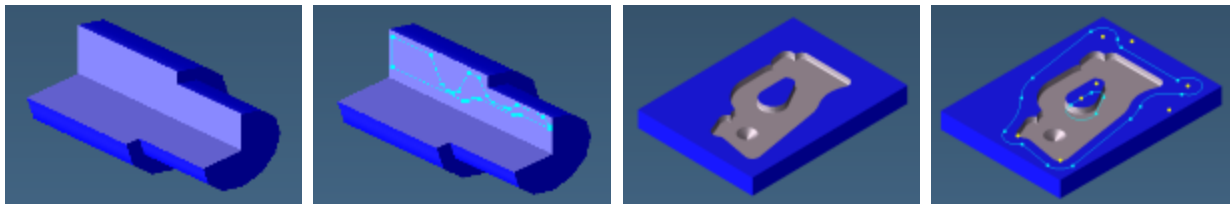
1/4 Cutaway

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

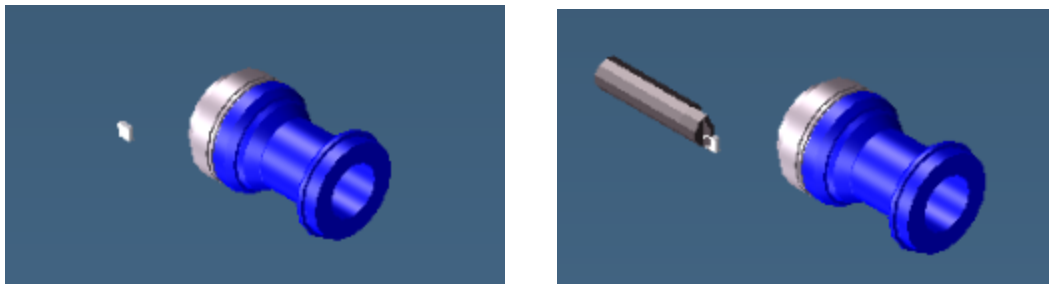
 **Show Rapid Tool**

(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.



**Run time or Number of Operation Display**

(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.

## Simulation Control Icons

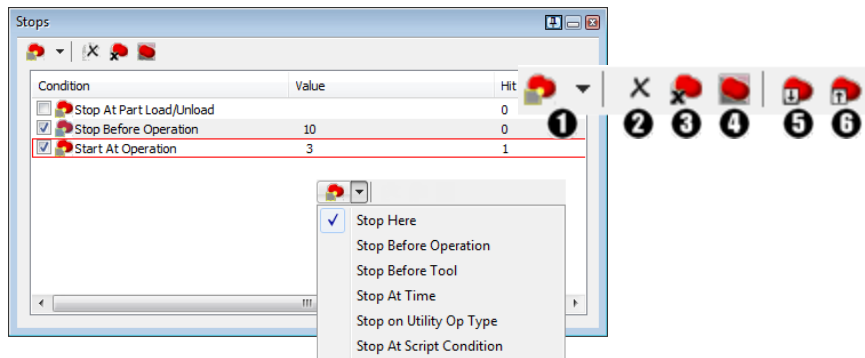
The Rapid, Part, Tool and Machine Simulation palettes have various control options. To select an option, click on the icon. In many cases, more than one icon can be selected at once.

 **Stops**

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

Selecting this command allows you to set a point before which the rendering should stop. Use the checkbox to add/deselect options. Rendering will stop after completion of the condition before the number specified in the dialog.

Three stop options appear on the initial page. Select a stop using the checkbox, then double-click the option to input conditions. All options except for "Stop at part Load/Unload", provide a window to input specific operation/tool numbers, times, and so forth. 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.

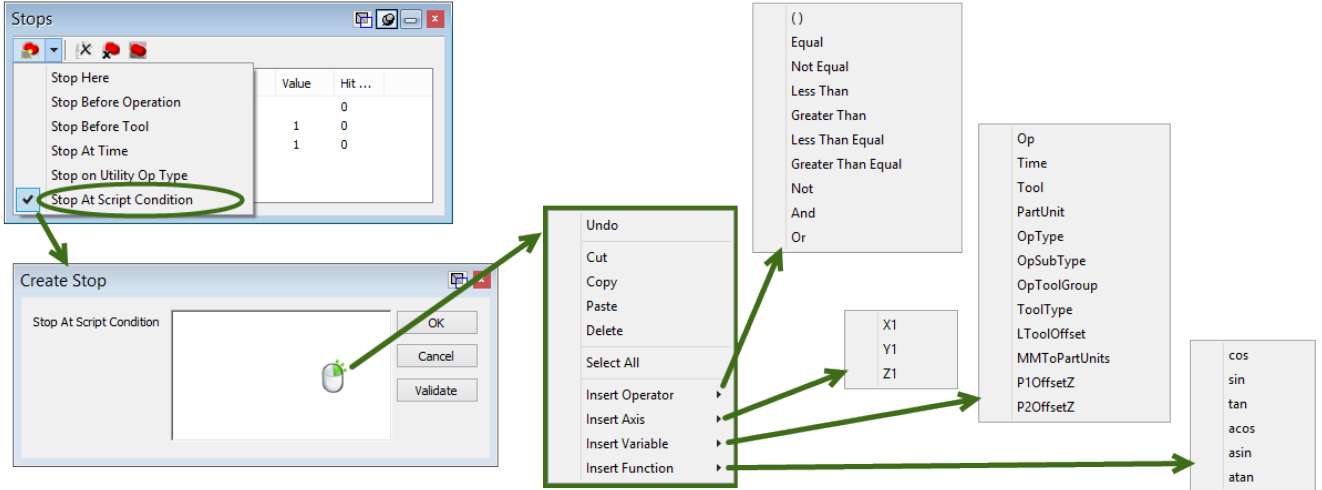


1. Stop Dropdown menu
2. Delete Selected Stops
3. Delete all Stops
4. Enable/Disable selected stops



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. The **Validate** button will ensure there are no script parsing errors.

### Stop at Script Condition Options

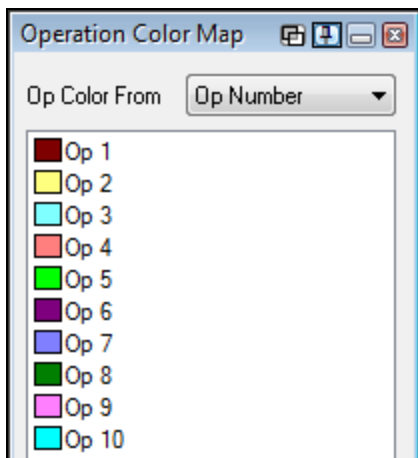


### Tool motion on target body (Available for Operation and Machine Simulation)

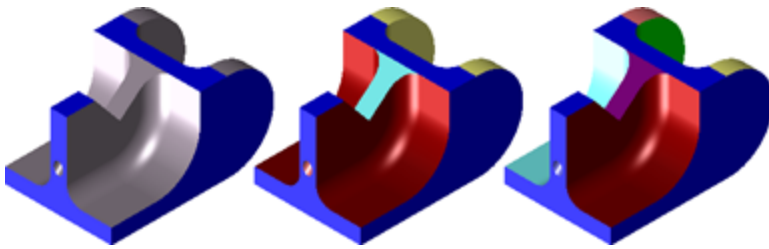
This option provides a non-cutting simulation of the tool motion, like the simulation in Tool Sim Rendering. Selecting this option will increase the speed of the simulation, where the display of material removal is not as important as being able to see what the tool is doing.



### Op 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 Cut Color, Tool Number, and Op Number use

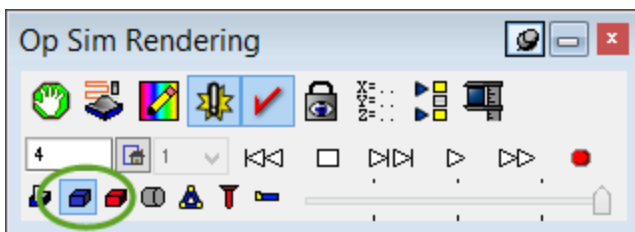
 **Collision Checking**

(Available for Operation and Machine Simulation)

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)

This option enables checking of collision events. The result of a collision event is controlled by setting the Collisions/Limits parameters; for details, see [Settings for Op Sim, Tool Sim, and Machine Sim](#). The tolerances for collision detection are set in the Preferences. 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.

Please note that to fully report gouges the target must be completely visualized. To do this, ensure that Stock Transparency is set to Translucent.



 **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.

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 the user 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.

Machine Sim Rendering	Op Sim Rendering
Operator	Fixed Part
Operator	Fixed Part
Part	Tool Rotary Axes
Machine Component	Tool Linear Axes
Tool Rotary Axes	Tool Rotary & Linear Axes
Tool Linear Axes	Machine
Tool Rotary & Linear Axes	

**Operator** (Machine Sim) Fixes the view to outside the machine.

**Part/Fixed Part** - (Machine/Op Sim) 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) Locks the view to a particular machine component. Choose from the pull-down menu which component is to remain stationary during simulation.

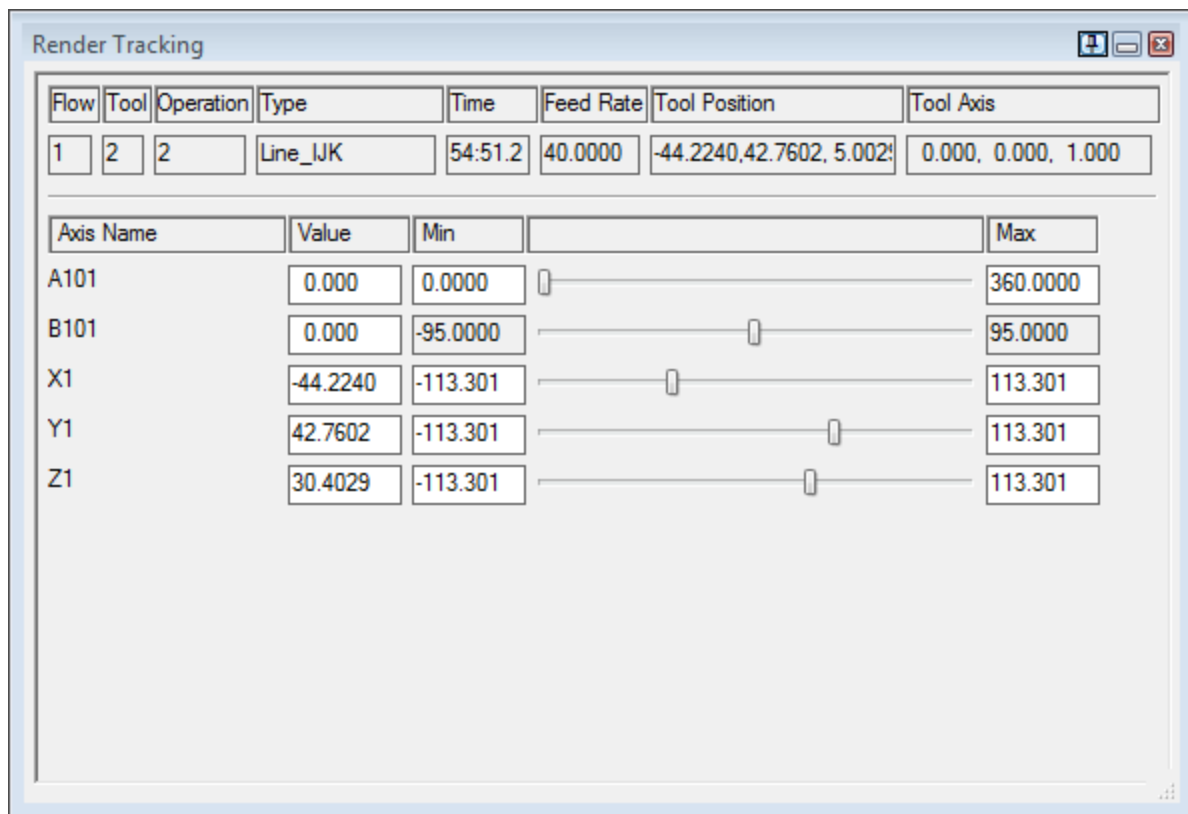
**Tool Rotary/Linear Axes** (Machine/Op Sim) The view will lock on the tool and its movements along the chosen axis.



**Show position**

(Available for Tool, Operation and Machine Simulation)

This option provides a dialog box showing continuous tracking of axis motion and current feature information. Sliders allow the user to explicitly move machine axis.



### Skip Unselected Ops

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

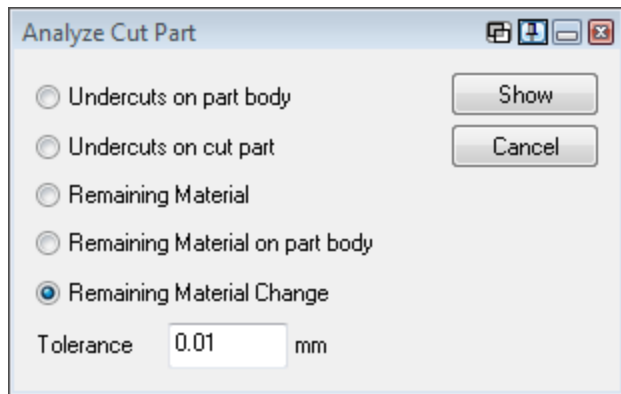
This option will only render the currently selected operations. (The other operations are still generated.) This option reduces the rendering time.



### Analyze Cut Part

(Available for Rapid and Operation 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) or if any cuts violate the part. You must select a solid prior to starting Simulation to use the Analyze Cut Part option.



### Trace Options

(Available for Tool Simulation)

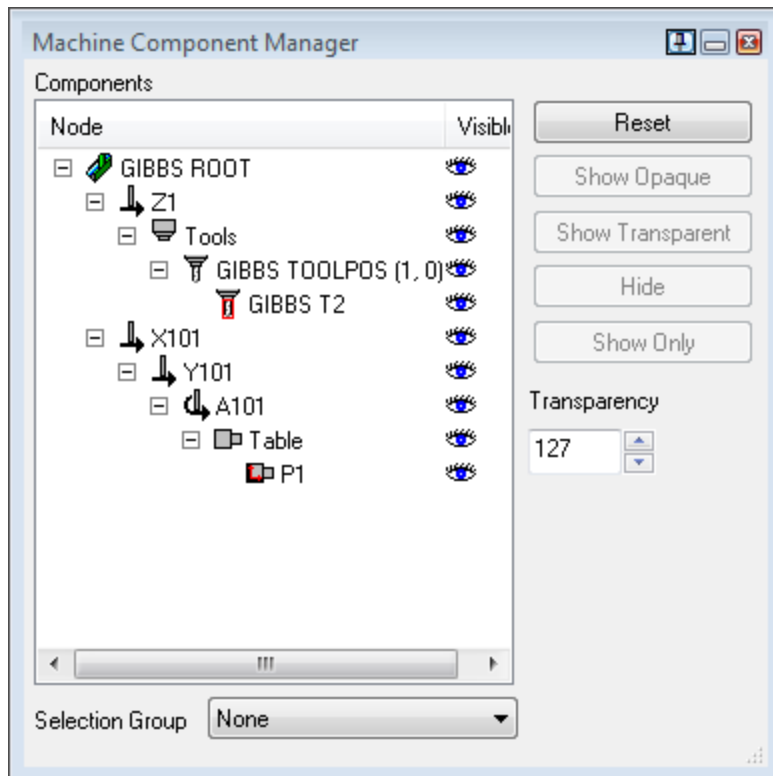
This option let you choose how much of the toolpath will remain on the screen – 0%, 1%, 5%, or 100% of the toolpath – as the tool follows the toolpath. **Trace Operation** will draw one operation at a time. **Trace From Run** will clear the previous toolpath and begin redrawing if the user stops the render midway through and restarts.



### 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 (solid).



### **Don't Block Stock**

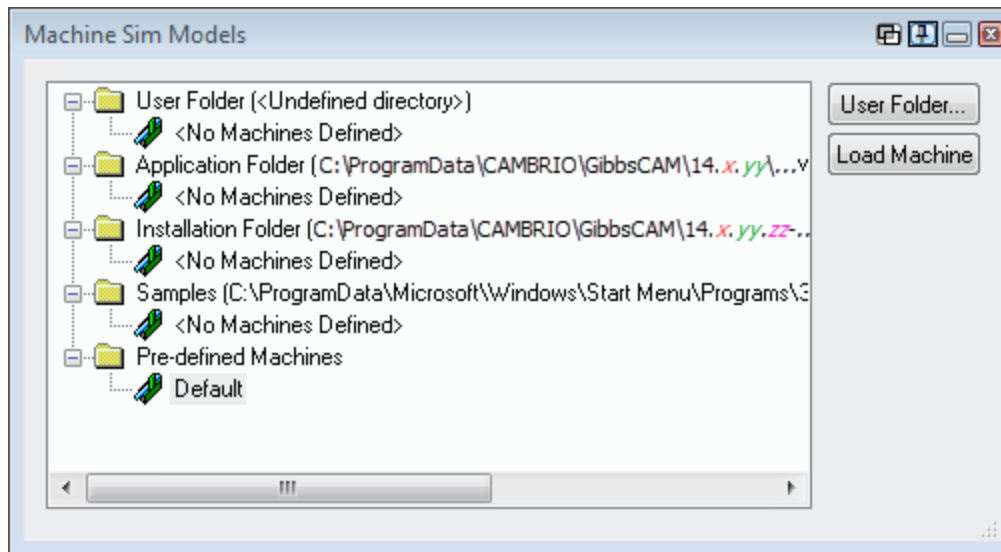
(Available for Machine Simulation)

Don't Block Stock is a rendering mode that renders all fixture bodies that appear in front of the stock bodies as transparent. This allows the user to see material removal on stock bodies even when some other bodies move in front of them.

### **Load Machine**

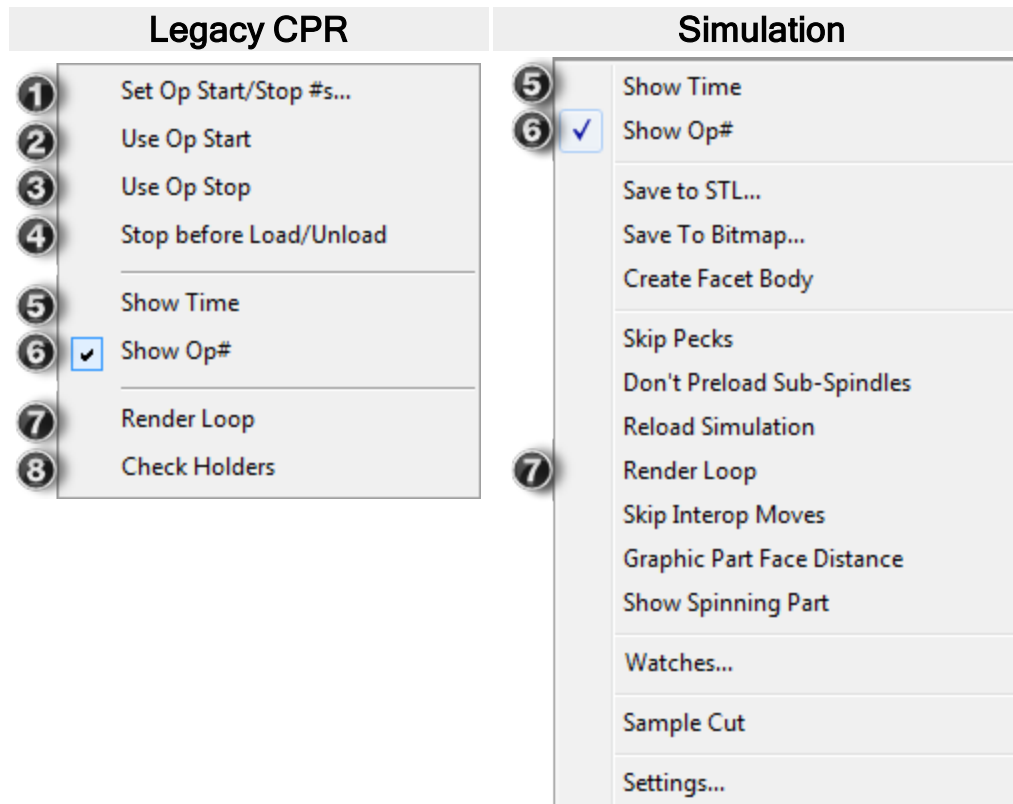
(Available for Machine Simulation)

This option allows you to select which machine assembly file will be used for the current part. Once 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**.



## 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.

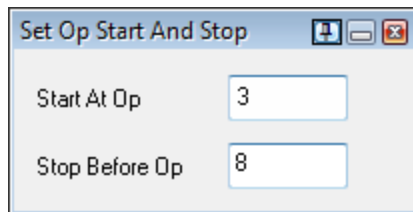


## Legacy CPR Context Menu

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.



### 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

This will set the Current Display to show the elapsed cut time.

### Show Op #

This will set the Current Display to show the current operation number.

### Render Loop

Replays the simulation until the user presses the stop button.



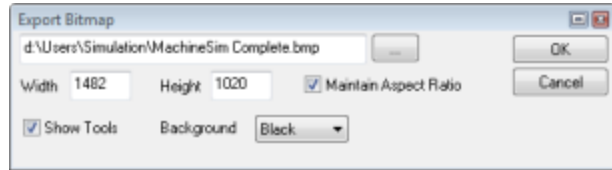
The following options are only to be found in the Simulation dropdowns:

**Save to STL**

Select this option to save a copy of the simulation to an .stl file for viewing later.

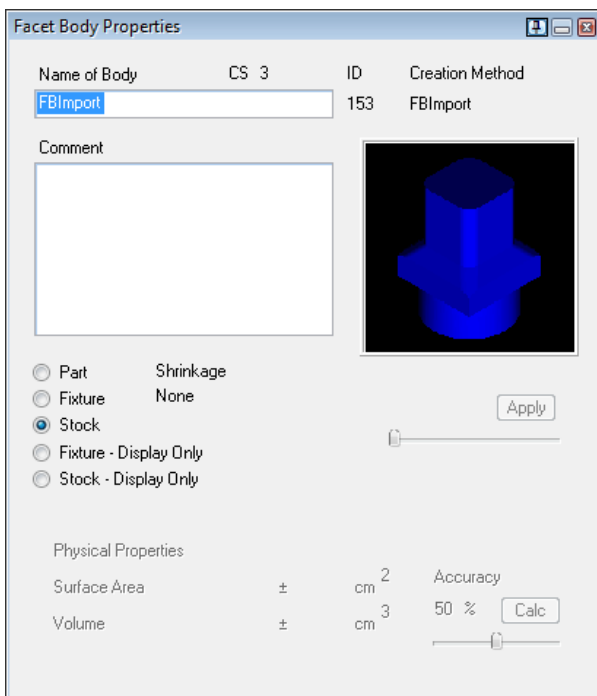
**Save to Bitmap**

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 most 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.

**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.

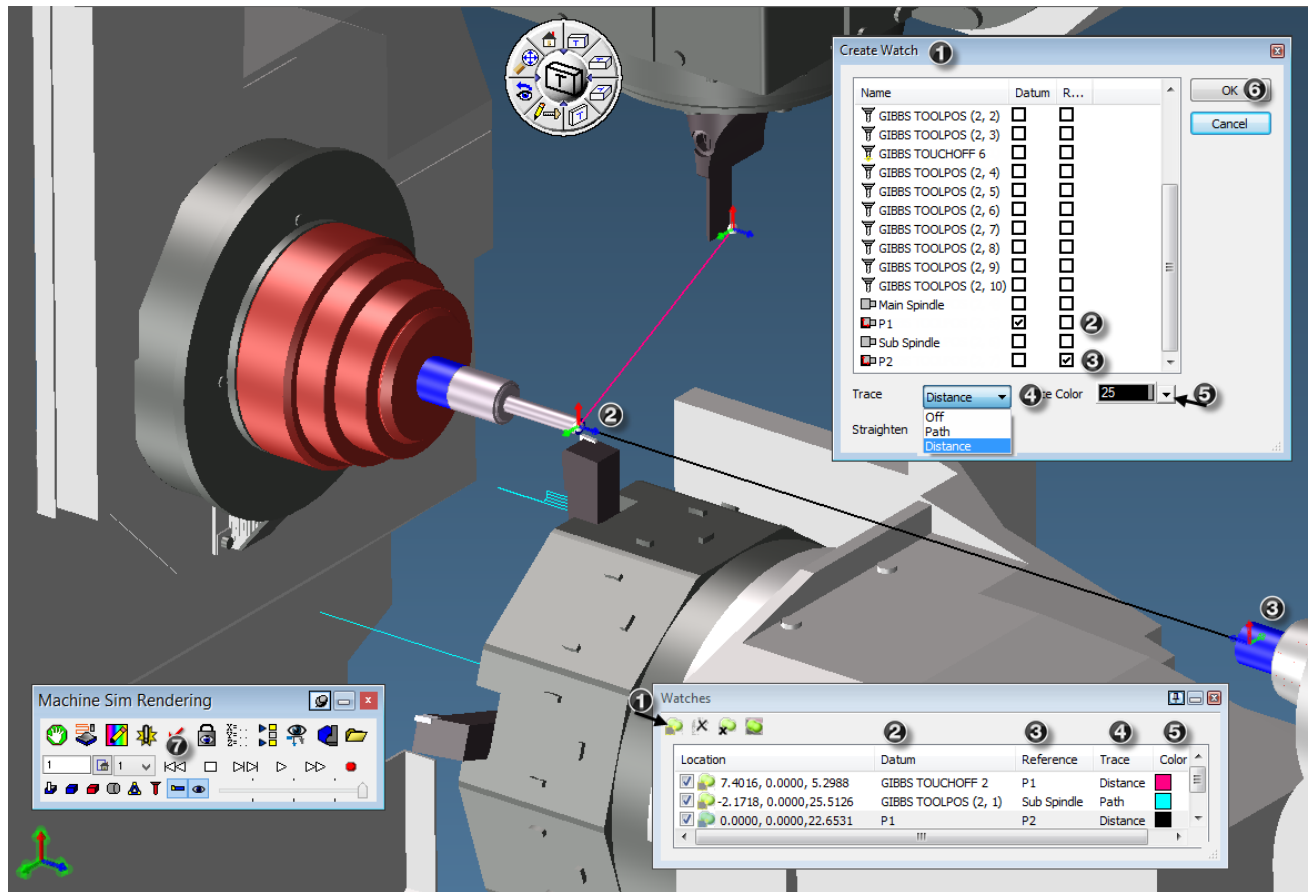
**Watches**

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 new 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 which can be of benefit in particularly dense surfacing toolpath where users have traditionally 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 [“Printing” on page 24](#).

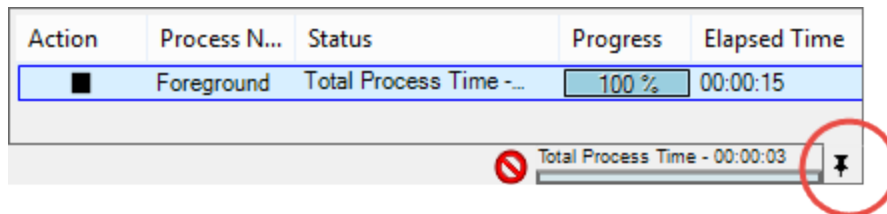
# Status Bar




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.

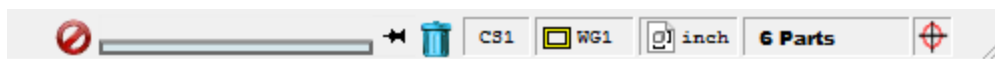
Double-clicking on the Workgroup and CS indicators will open their respective dialogs. If present, double-clicking the TMS status indicator will open the DCD dialog to the Multi-part tab.

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

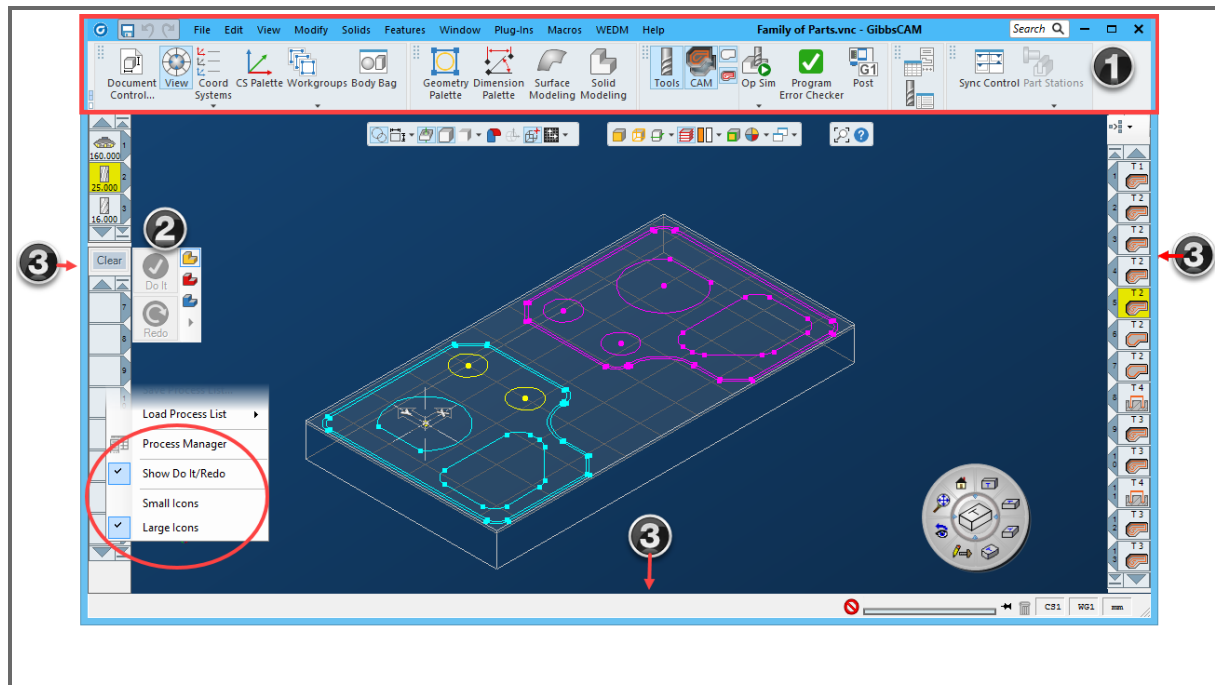
When you right-click the trackball, its context menu now offers a powerful choice: Customize.



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.

The image shows the **Trackball Customization** dialog box and a grid of trackball button layouts. The dialog box is divided into two sections. The top section, labeled "Plain", shows a list of commands such as "Previous View", "Zoom In", "Zoom Out", "Pan Left", "Pan Right", "Pan Up", "Pan Down", "Rotate Left", "Rotate Right", "Rotate Up", "Rotate Down", "View Labels", "View Points", "Extend Lines", "Show Stock & Origin", and "Show Geometry". A green arrow points from "Pan Down" to a trackball button in the "Plain" layout. The bottom section, labeled "Alt", shows a list of commands such as "Additive Report", "Additive Technology Data", "Additive Material", "Open the External G-Code Editor Application", "Open the Internal G-Code Editor Window", "Additive Weave Data", "Explore the Harvey Tool Library", "Explore the Helical Solutions Tool Library", and "Continue". A green arrow points from "Move the part towards the bottom of the screen." to a trackball button in the "Alt" layout. The grid of trackball layouts is organized by modifier combinations: "Plain", "Shift", "Ctrl", and "Alt". Each layout shows a trackball with eight buttons, each with a different icon and label. The labels for the buttons are: "Previous View", "Preferences", "Wrap Geometry", "Save Process List", "Shape Palette", "Workgoup Manager", "User Color Mode", and "Dup and Scale".

# Customizing the Workspace



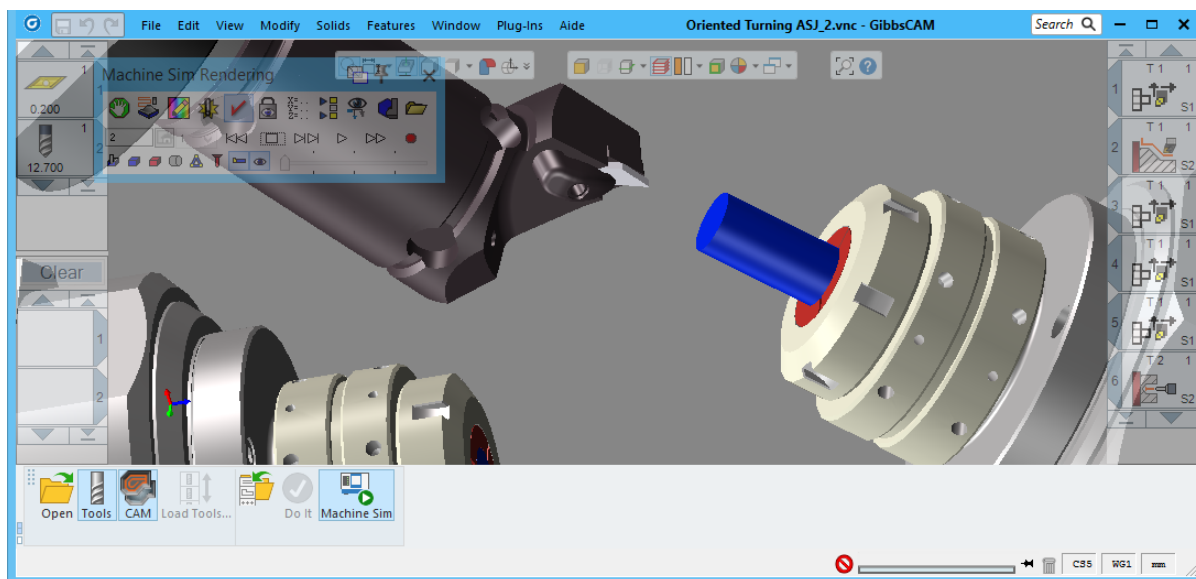
1. Right-click here to Customize
2. Do it/Redo Command.
3. Alternative command docking areas

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.

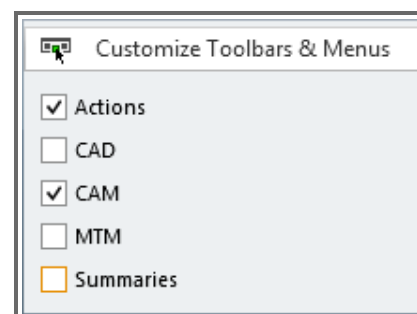
Other more sophisticated customization is possible by right-clicking within the header area highlighted within a red box in the screenshot above. You can totally rearrange, or duplicate command icons and place them into existing or custom Toolbars and Menus. For instance this would enable you to create a command menu specifically for your own oft-used workflows.



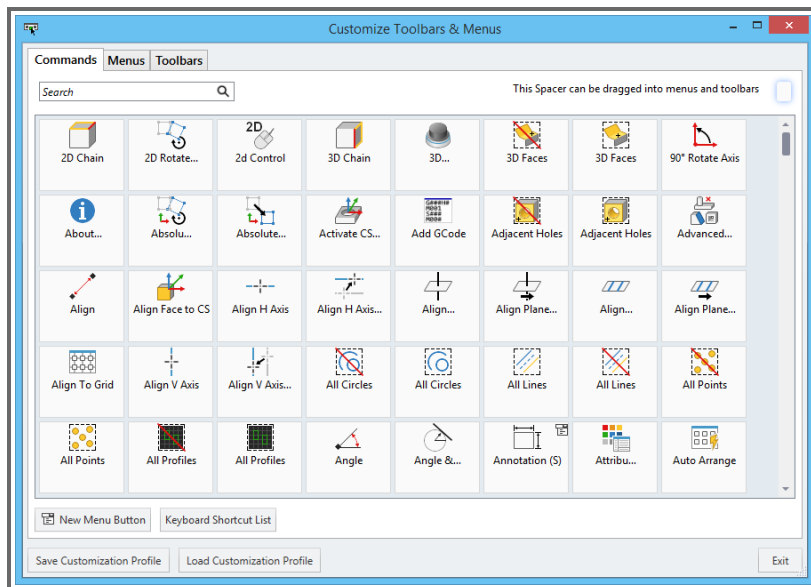


## Customizing Toolbars and Menus

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 temporarily 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.



## 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\3D Systems\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

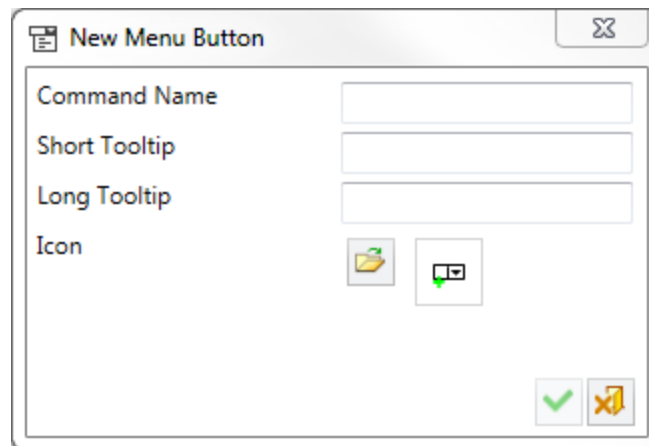
The text string that will appear when the mouse hovers over the command container.

### 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 .



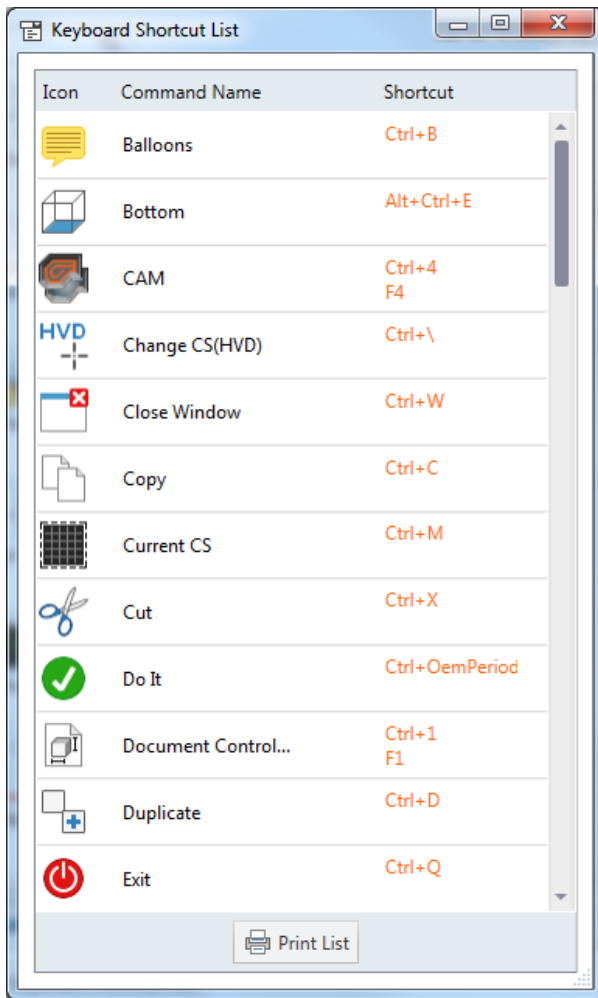
OK: Accept changes and close the dialog.



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.

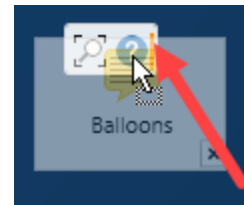
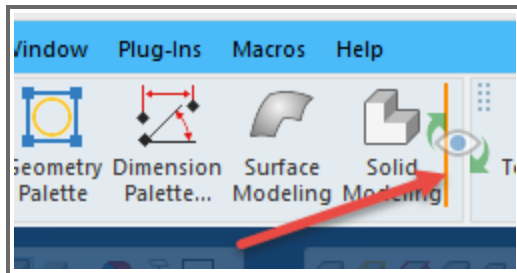
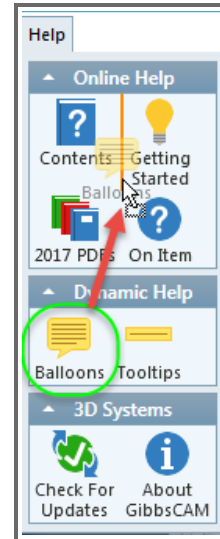
### Shortcuts

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.

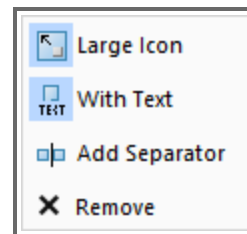
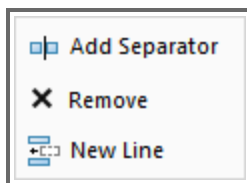


**Reposition an icon**

If your command has been placed 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**

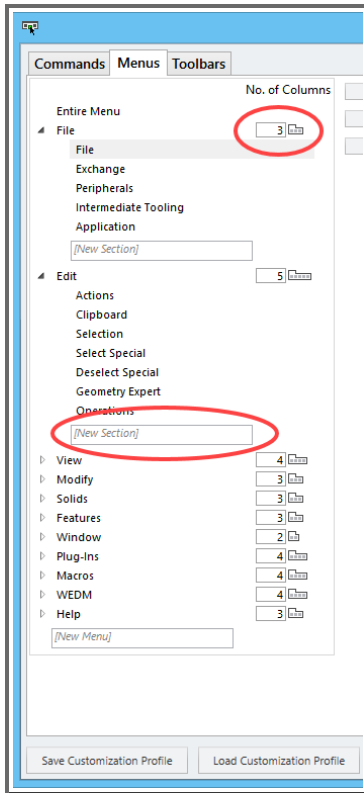
Right-clicking a command icon in the UI while in Customization mode gives additional options to arrange the commands by adding separators or extra lines. Please note that only one separator can be added in a particular space. Clicking Add separator a second time will remove it.



Right-clicking an icon in the Main-menu dropdown

Right-clicking an icon in the toolbar

## Menus tab



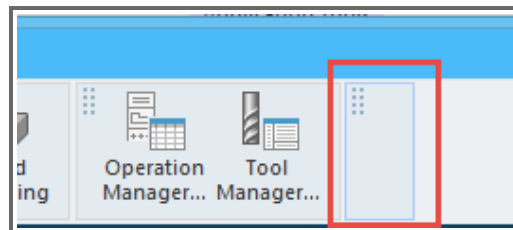
Use the **Menu** tab to create a new Main Menu option or to add a sub-section to an existing menu. You can also change the width of the Main menu dropdown - just type in the required number of columns and click the column icon next to it. Menu options can be renamed, or if need be, deleted.

## 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.

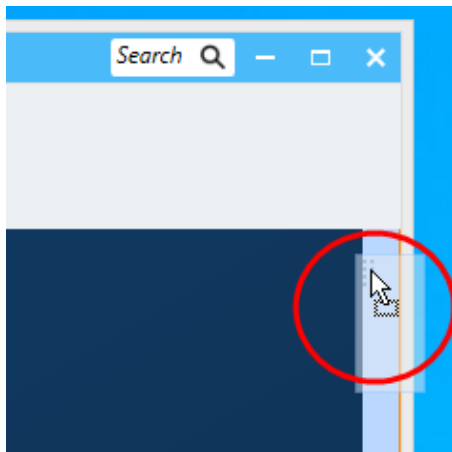
## Toolbars Tab



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.

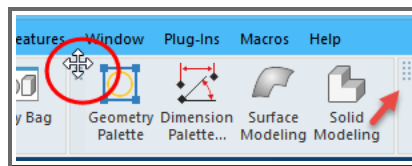
To add items to the Quick Access or Floating Toolbars, just drag the required command into the toolbar.

## 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.

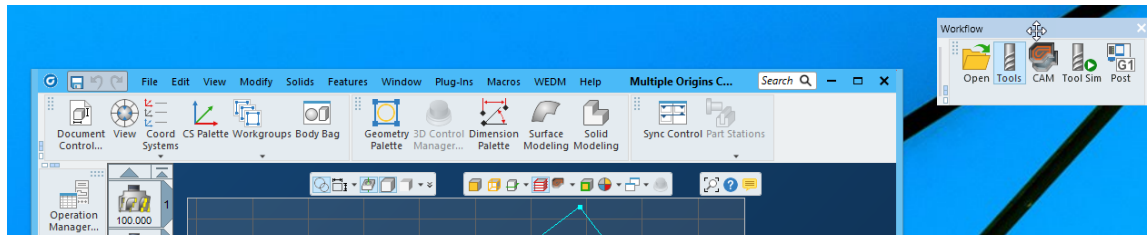
The orientation automatically adjusts to fit the available space.

## 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.

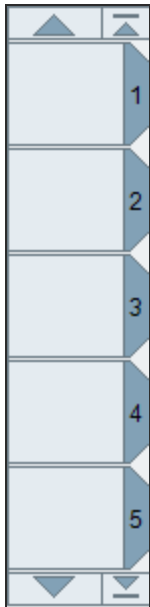





# Miscellaneous


- Lists
- “Other Workspace Items” on page 179
- “Shortcuts” on page 180
- “Cursors” on page 181
- “Eyeball icon” on page 183
- “Freehand Actions” on page 183
- “Right-Click” on page 185
- “Colors” on page 185
- “Math Functions” on page 187
- “Interrogate ” on page 187
- “About Clearance Volume ” on page 189

## Lists



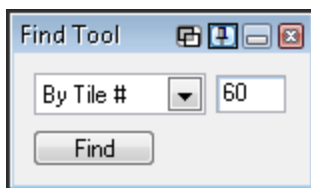
There are three main lists in the system. These are the Tool List, the Process List and the Operations List. These lists hold tiles and are not moveable. Each tile is used to describe an individual tool, process or operation. A list can hold many items.

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.

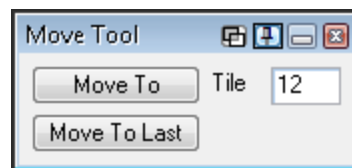
You can also scroll to the first or last non-empty tile by clicking the smaller button  next to the single-scroll button.

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.

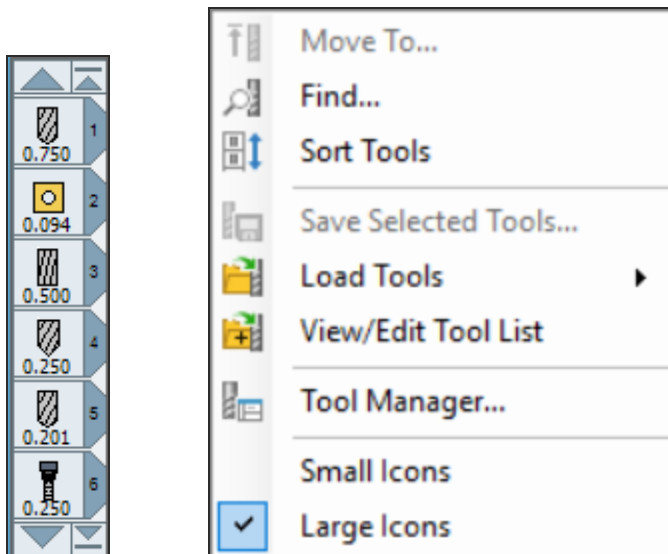


Find Tile dialog



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. By **double-clicking** on an empty tile a new tool is created and a Tool dialog opens. A tool is then fully defined within 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 both milling and turning tools at the same time. Each tool has a unique graphic to help the user quickly determine what each tool is.

The Tool List is accessed from the Tool button in the Commands palette. There can be a maximum of 999 tools in a part. The tool tiles do not have to be contiguous; there can be empty spaces between tiles.

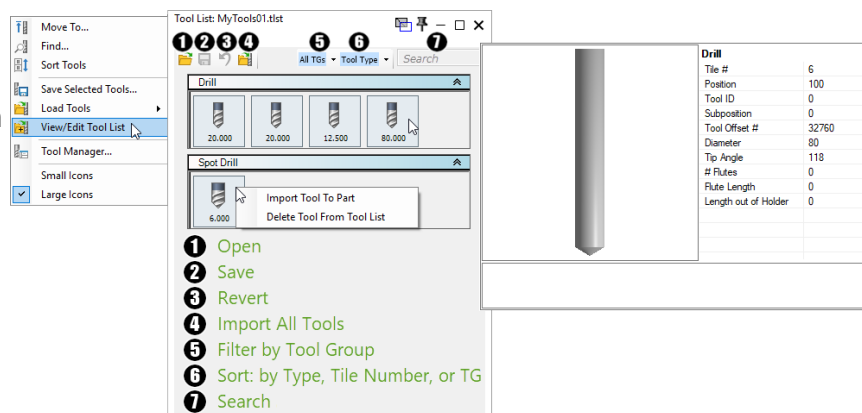
To navigate through the Tool Tiles, click on any part of the Tool List to select it. Then use the mouse wheel to scroll through the list. Use **Ctrl**+the mouse wheel to scroll through multiple tiles at a time. The Process List and Operations List, described elsewhere in this section, can also be navigated in this manner.

Tool lists can be saved and even automatically generated by using default part files. See “[Type Default Part](#)” on page 12.

## View/Edit Tool List

Tool List Manager 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.

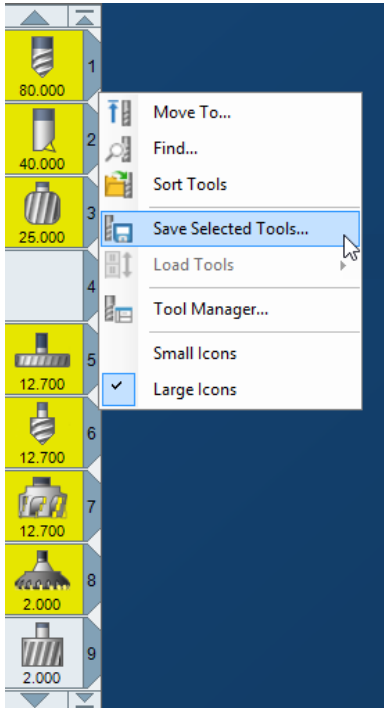
## Save or Load Selected Tools

*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 (\*.t1st) that you want to load.

**Tool List**

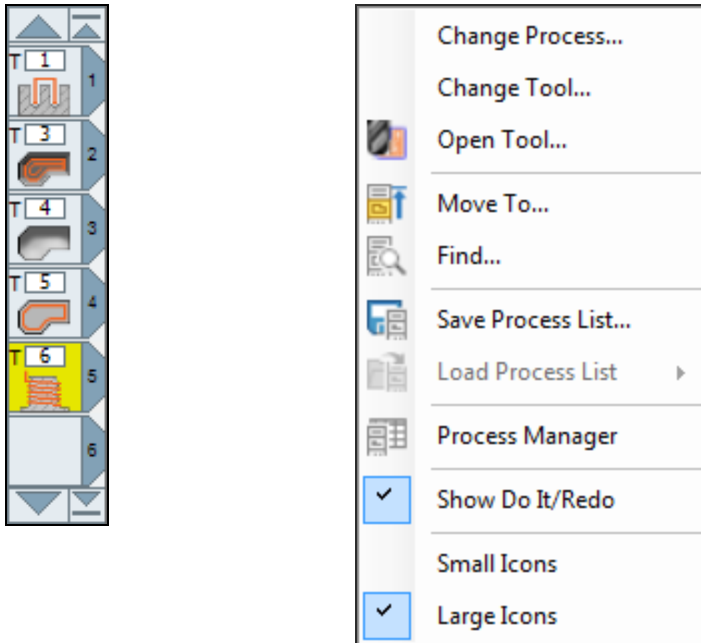


The screenshot shows a vertical list of tool tiles. Each tile has a number (1-9) and a numerical value. A context menu is open over the third tile (value 25,000). The menu items are: Move To..., Find..., Sort Tools, Save Selected Tools... (highlighted), Load Tools, Tool Manager..., Small Icons, and Large Icons (checked).

**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.

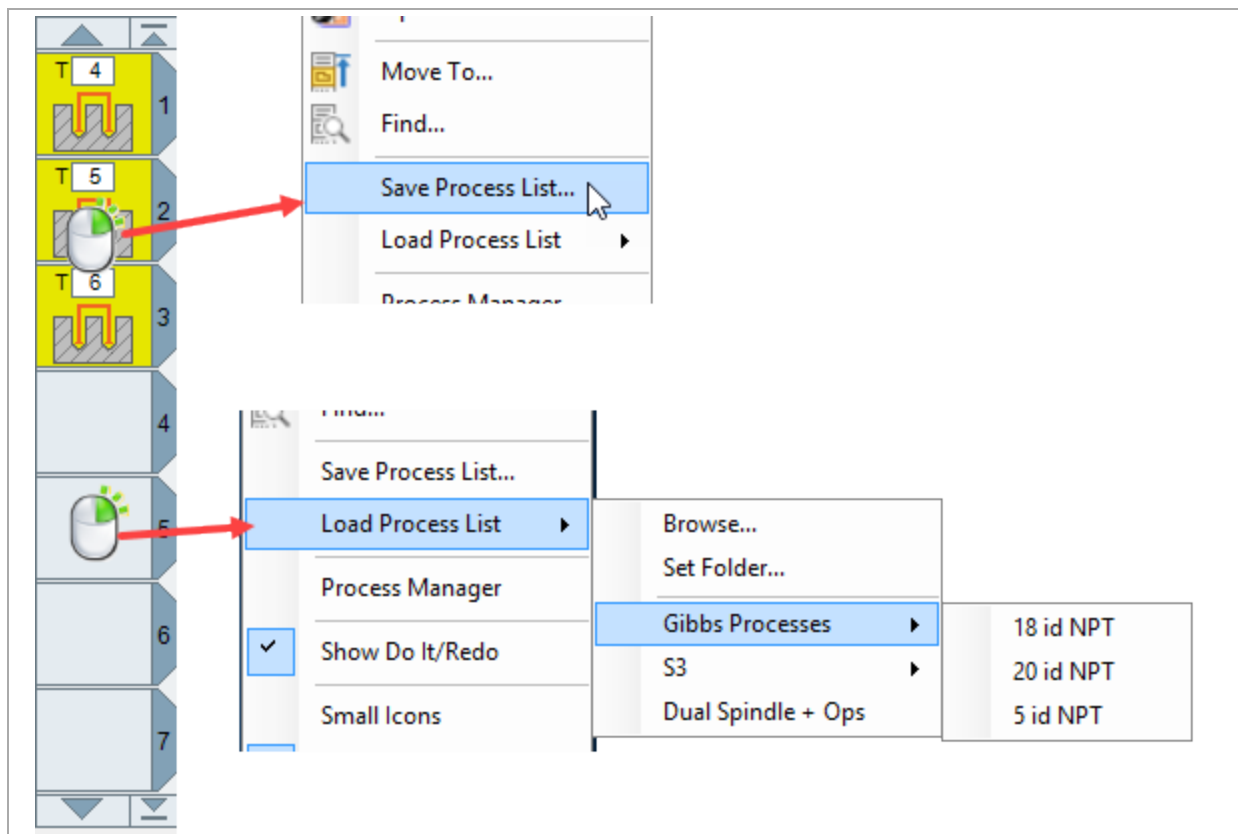
Clearance values, cut depth, speeds and other items are specified in the Process dialog. In most cases, geometry or a solid is then selected to generate toolpath by **clicking** the Do It button. Once toolpath is generated, one or more operations are created for each process. After operations are created, 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

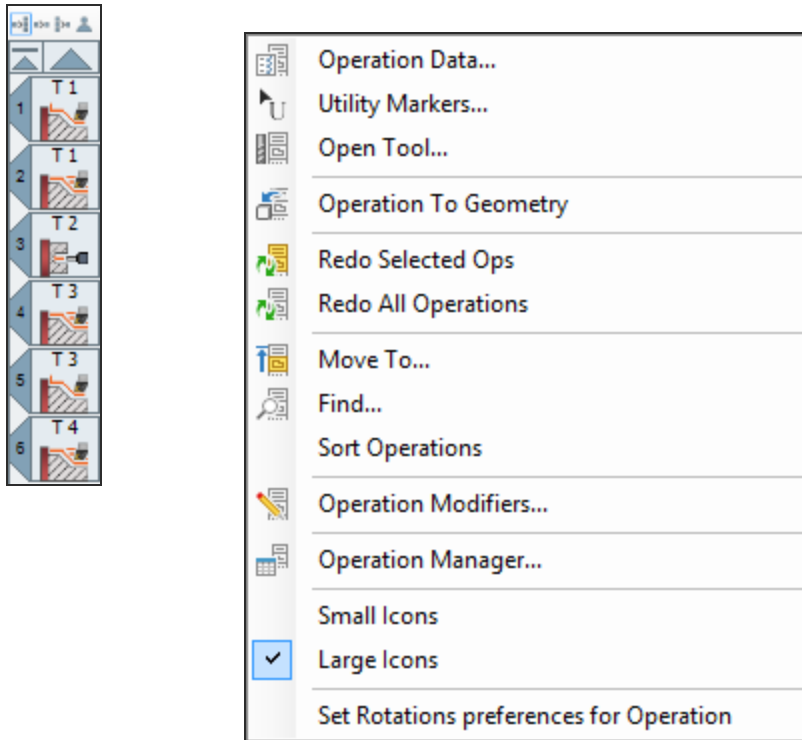
Process List



**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



### The Operations List and its (right-click) context menu

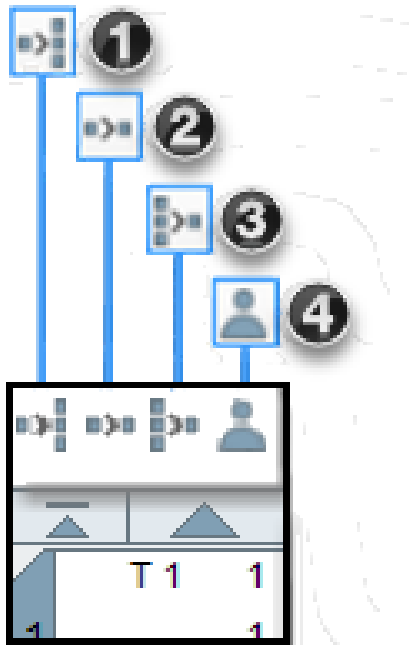
Operations are created from processes and tools. Operations store the toolpath (the cutting motion of the tool) and the information defined by a process. A process generates at least one operation, some processes will result in more than one operation. The software will use the list of operations to generate the final program that is sent to the CNC machine.

There may be a maximum of 16,000 operations in a part. 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 [Lathe](#) guides.

### Operation Tile Stacking

The Operations list offers four choices for organizing Op tiles:



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.

## 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.



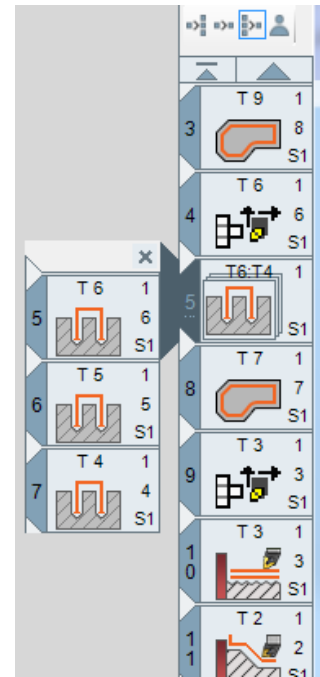
For **Manual**, the stack icon shows the process type of the earliest operation.

## 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

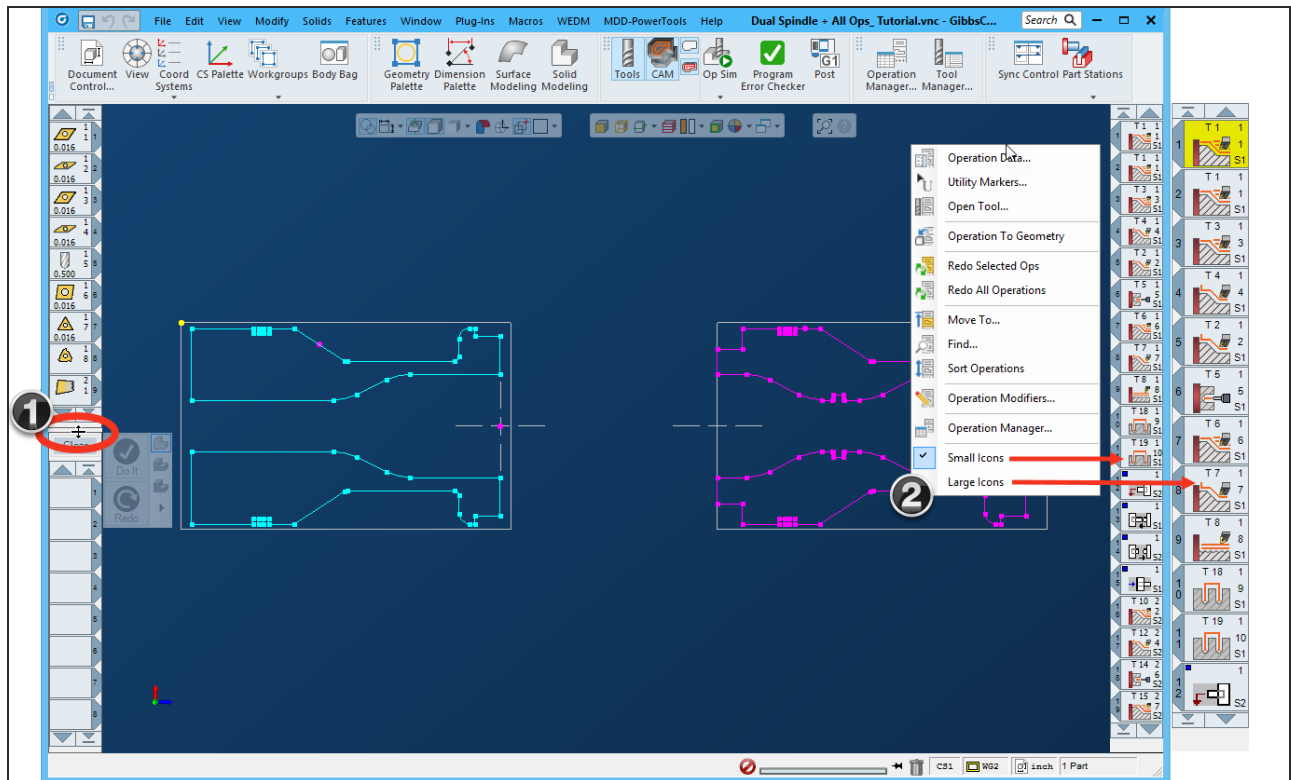
You can drag an entire stack and drop it anywhere in the op list or into another stack. When a stack flyout is open, you can drag tiles into or out of the stack. However:

- 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.

## 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.









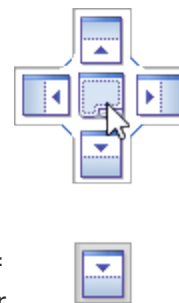
❶ You can view more tool icons by dragging down the bottom margin of the Tool Tile list. Drag it up to see more Process Tiles.

❷ To increase the number of tiles on view you can also switch to Small icons.

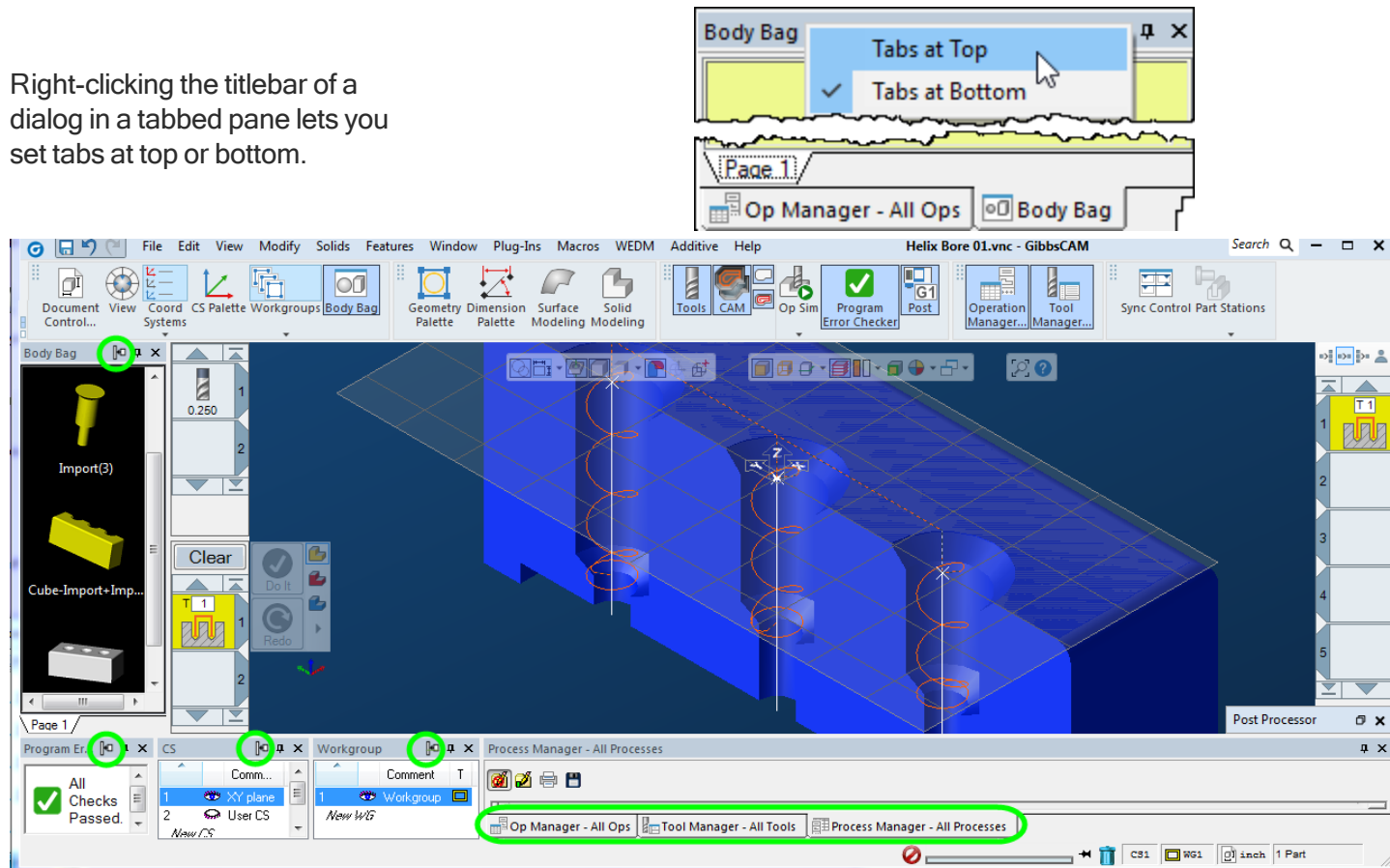
## Dockable Panes

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.



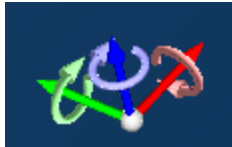
### Axis Block

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 17.



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**.

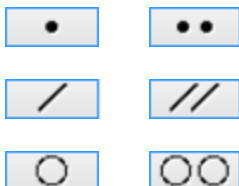
## Shortcuts

### 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.

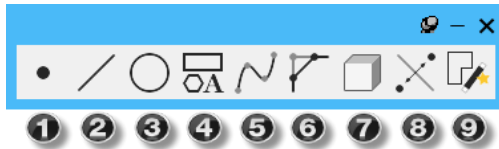


### Command Shortcuts

Keyboard shortcuts can be set up for often-used commands. See the section on [Customizing the User Interface](#).

### Palette Shortcuts

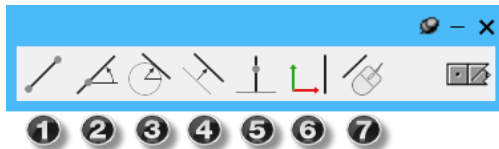
There are some keyboard shortcuts for functions other than menu choices in the system. Simply hit the appropriate number to activate the button.



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.

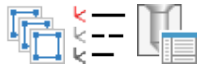
The Help cursor activates context-sensitive online help for the item you click on. Online Help will open and will attempt to load the section regarding the item you clicked. Some items go to a very specific topic; others will lead to a general section of the help.







## 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:

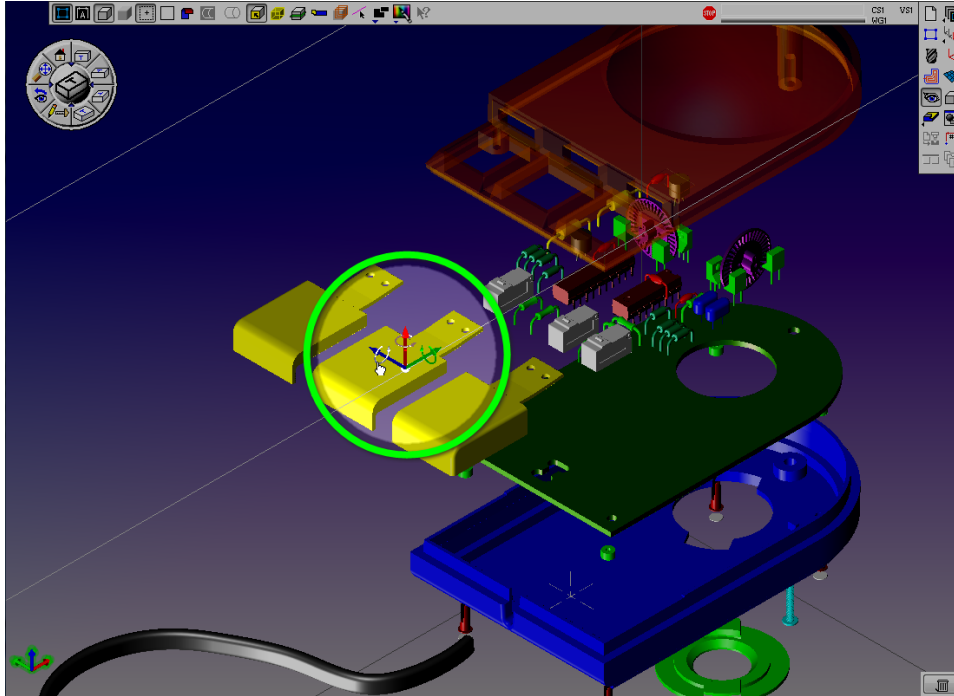
- All mouse actions in the View Control Palette, such as:
  - **Drag** and **right-drag** (◦ cursor) of the trackball axis handles, for constrained rotation or of the trackball itself, for unconstrained rotation.
- **Ctrl-drag** ( cursor) to move the workspace.
- **Mousewheel-turn** to zoom and **mousewheel-drag** 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.

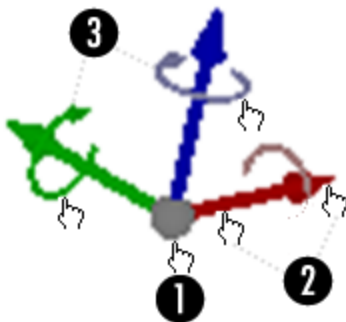
## Freehand Move and Rotate

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



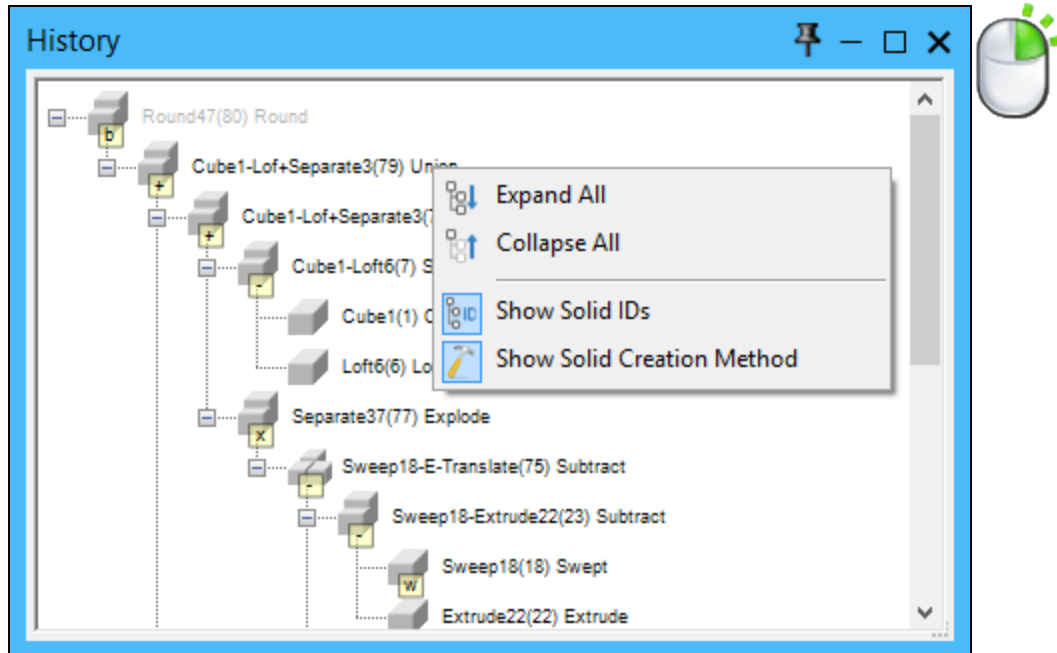
1. For a freehand move operation of selected elements, click and drag the origin of the pop-up axis block.
2. For a freehand move operation along one axis of the pop-up, click the axis arrow's shaft or arrowhead and drag along the axis.
3. For a freehand rotation operation around one axis of the pop-up, 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.

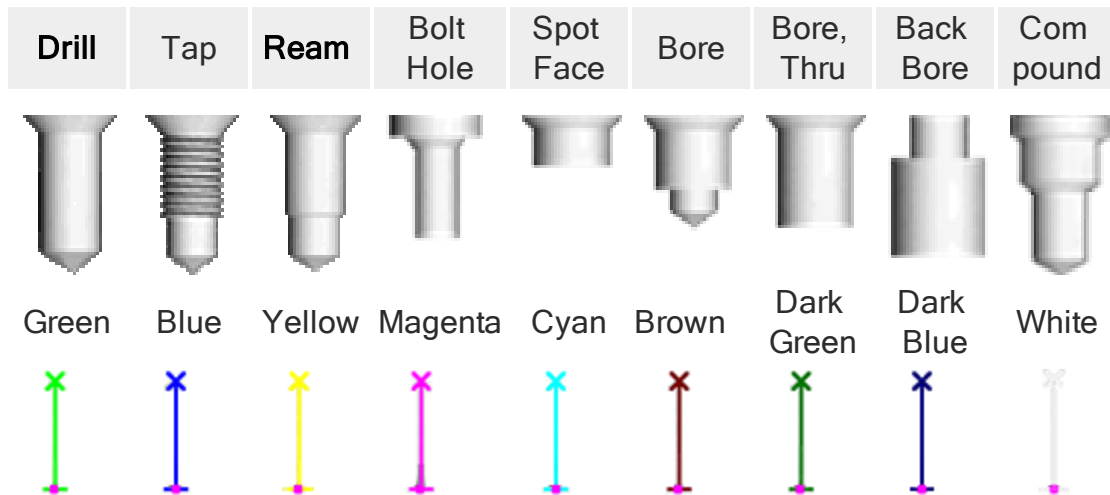
Item	Color	Meaning
Workspace	White	Wire frame view of the Part Stock and coordinate system markers
	Green	Profile Grid

Item	Color	Meaning
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.

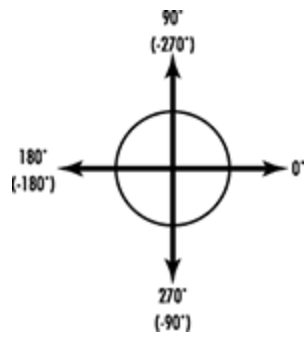
Key	Function	Key	Function
+	addition	-	subtraction
*	multiplication	x	multiplication
/	division	=	equals
m	convert inch to metric (x25.4)	i	convert metric to inch (/25.4)
s	sine	c	cosine
t	tangent	a	arc tangent
r	square root	e	scientific notation (41e-2 = 0.41)

## Interrogate

When a text box contains a cursor, you can **Alt+click** a point or other element (including toolpath) to interrogate its value and place it into the text box.

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**.

Please note that angular values entered in text boxes follow the standard Cartesian coordinate system, as shown. 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.

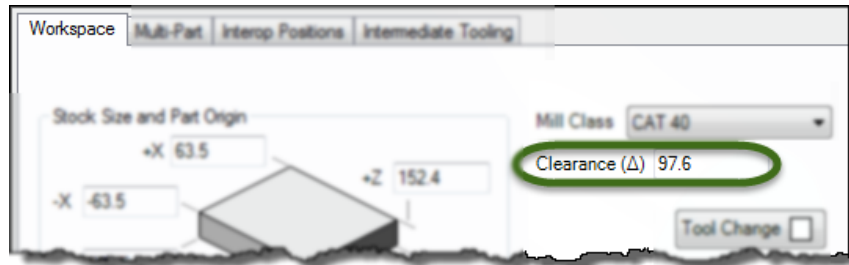
**Caveats:** Interop moves generated by Clearance Volume contain 5-axis simultaneous moves; thus it is best if the control has TCP capabilities, and it is unsuitable for machines that have indexing rotary axes or rotary axes that must be clamped between moves.

### 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 ( $\Delta$ ) 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

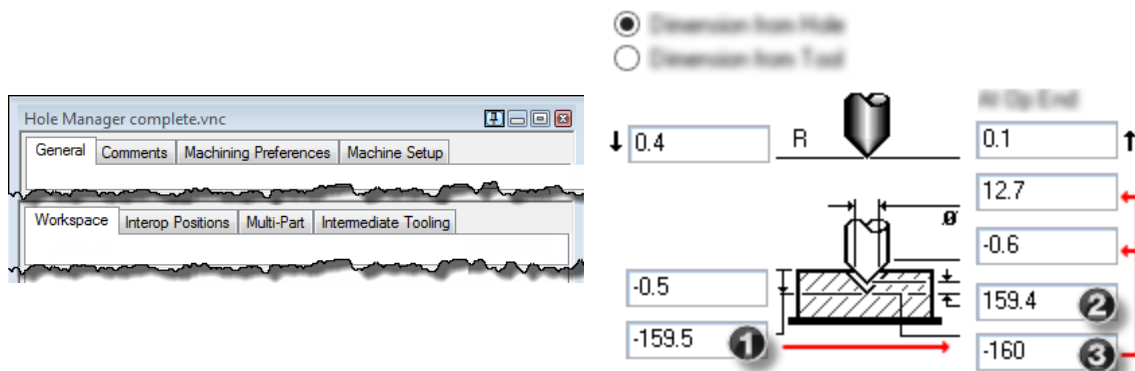
**Screen text.** Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

**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.

Faint green borders that outline areas within a graphic usually signify an image map. In online help or a PDF viewer, you can click a green-bordered area to follow the link.

## Links to Online Resources

Link	URL	Action / Description
<a href="#">Go</a>	<a href="http://www.GibbsCAM.com">http://www.GibbsCAM.com</a>	Opens the main website for GibbsCAM.
<a href="#">Go</a>	<a href="https://online.gibbscam.com">https://online.gibbscam.com</a>	Opens a restricted website containing materials available for download. Requires a GibbsCAM Online Services account; to set up an account, contact GibbsCAM Support.
<a href="#">Go</a>	<a href="https://store.GibbsCAM.com">https://store.GibbsCAM.com</a>	Opens the website for the GibbsCAM Student Store.
<a href="#">Go</a>	<a href="https://macros.gibbscam.com">https://macros.gibbscam.com</a>	Opens a wiki containing documentation and examples of GibbsCAM macros. Requires a GibbsCAM account.
<a href="#">Go</a>	<a href="http://kb01.GibbsCAM.com">http://kb01.GibbsCAM.com</a>	Opens a Knowledge Base article, <b>Contour Operations Using Thread Mill Tools</b> , that explains in detail the correct way to program Contour processes using Thread Mill tools.
<a href="#">Go</a>	<a href="mailto:Support@gibbscam.com">mailto:Support@gibbscam.com</a>	Runs your email client to create a new message addressed to the CAMBRIO Technical Support department for GibbsCAM.
<a href="#">Go</a>	<a href="mailto:Registration@gibbscam.com">mailto:Registration@gibbscam.com</a>	Runs your email client to create a new message addressed to the CAMBRIO Registration department for GibbsCAM.
<a href="#">Go</a>	<a href="mailto:Sales@gibbscam.com">mailto:Sales@gibbscam.com</a>	Runs your email client to create a new message addressed to the CAMBRIO Sales department for GibbsCAM.
<a href="#">Go</a>	<a href="http://www.autodesk.com/inventor">http://www.autodesk.com/inventor</a>	Opens an external website that provides more information on Autodesk Inventor products.
<a href="#">Go</a>	<a href="http://www.celeritive.com">http://www.celeritive.com</a>	Opens an external website that provides more information on VoluMill Ultra High-Performance Toolpath (UHPT) from Celeritive Technologies.
<a href="#">Go</a>	<a href="http://www.predator-software.com">http://www.predator-software.com</a>	Opens an external website that provides more information on a CNC editor and a virtual CNC viewer from Predator Software, Inc.

# Index

---

## %

% of Body's Chord Height 27

---

## 2

2016 PDFs 99

2D Rotate 72

---

## 3

3D Real Time Cut Part Rendering 137

---

## 5

508MT (Willemin)  
and Clearance Volume 189

---

## A

About GibbsCAM 102

Absolute 2d Rotate 72

Absolute or Incremental, in post 45

Absolute Translate 73

Activate Dialog on RMB 17

Active Content 105

Additive Preferences 51

Air geometry 76

Alloy Group 53  
material 54

Alloys 54

Ambient 18

Ambient, Material Properties 18

Angle, 2D Rotate 72

---

Angular Resolution 27

anonymous usage statistics 33

Apply Value to Selected  
Managers 93

Approximate Arcs 28

Arc segments 77

Attribute Manager  
menu item under Features 85

Auto Tool Size 15

Axis block  
about 180  
pop-up 184  
showing/hiding 20

Axis Limits 147

Axis Marker in Sim 20

---

## B

Background Tab, Appearance  
Settings 20

Balloons 100

Bitmap 153

Block Numbers 134

Bodies (Graphics) 24

Bodies, Select 61

Body Chord Height 27

Bolt Table 38

Bookmarks, online help 104

---

## C

Calculate  
Feed 57  
RPM 57

Cascade Dialogs 87

Change CS (HVD) 71, 78  
Change CS (XYZ) 71, 78  
Change Intersection, Geometry  
    Expert 63  
Check  
    Body Validity 81  
    Face Validity 81  
    Op Gouges 82  
    Self-Intersect Loops 80  
    Trimmed Surface Edges 82  
    Trimmed Surface Polyline 82  
Check for Updates 101  
Check Out, NLO seats 34  
Chord Height 24, 27, 77  
Circles, Select 60  
    All Circles 60  
Circular Threads 27  
Clear Bodies' History 83  
Clearance ( $\Delta$ ) 190  
Clearance Volume 189  
Closest Hole Next, sort 76  
Collision events 146-147  
Color Button 23  
Colors Appearance Setting 18  
Comment 54-55  
Comment, select by 62  
Common Appearance Settings 17  
Communication dialog 13  
Communication Set-Up 47  
    Protocols 47  
        Change 48  
        Remove 48  
Communications 13  
Condition 54  
Configurations, system display option 17  
Contents, Help 98  
Contents, online help 103  
Context Sensitive Help 99  
Contouring 36  
Coordinate Systems  
    Creating and Modifying Options 43

CPR  
    Current Display 139  
    Next Operation 139  
    Play 139  
    Speed Control 139  
CPR/Sim Appearance Settings 19  
CRC 36  
CRC Type 35  
Create Facet Body 153  
CS Preferences  
    Geometry References 43  
    Out of Plane Rotation 44  
    Self Definition 43  
    Set New CS Visible 44  
    Toolpath References 44  
Current CS, Select 62  
Current Display box 139  
Current Interface Level 31  
Curves, Select 61  
Customizing the User Interface 159  
Cut Part Rendering 15  
Cut Type 55  
    material 54  
Cut Width 36  
Cut XY Depth 54  
CutDATA 52-53  
Cutdata Path 40  
Cutter Radius Compensation 35  
Cutting Options, Op/Tool Sim 26

---

## D

data collection 33  
Date and Time, in post 45  
Default Color 23  
Delete Row 64  
Depth, select by 62  
Diagnosing problems in solids 80  
Dialogs Follow Application 30  
Dialogs Roll-up 30  
Dialogs, control over 86

---

Digitizer Calibration 35  
Dimensions, Select 61  
Dimensions, Show 68  
Display Lists 21  
Display Settings option 16  
Do It 71  
Don't Preload Subspindles 154  
Drag Rectangle  
    Appearance Settings 17  
Draw Axis 20  
Draw Points 68  
Drawing  
    Print 13  
Drill Hole 84  
Driver Info... 23  
Driver, OpenGL Settings 22  
Duplicate 74  
Duplicate And 74  
    2d Rotate 74  
    Force Depth 74  
    Mirror 74  
    Scale 74  
    Translate 74

---

## E

Edges 24  
Edges, Select 62  
Edit Appearance Settings... 16  
Edit Material 54  
Edit menu 58  
Enable Caption 29  
Enable OpenGL Acceleration 17  
Enable toolbar transparency 31  
Engraving fonts 40  
Extend Lines 68  
Extended Characters, in post 45

---

## F

Facet Body Opacity 22  
Facets 24  
Family, Material 53  
Favorites, online help 104  
Feature Manager  
    menu item under Features 85  
Features menu 84  
Feed Status, Toggle 76  
File Extension 41  
File Extensions 133  
File Launch Behavior 39  
Finish Bore 36  
Finish Profile 36  
floating toolbars  
    transparency 31  
Floor Angle Tolerance 32  
Force Depth 71  
Format Code 45  
Format Name, in post 45  
Freehand rotation and translation 184  
From Tool Edge 36  
FromPos 73  
Front View 66

---

## G

G-code 36  
G-Code Editor Choices, in  
    Preferences 49  
G-Code Editor Preferences 49  
G-Code Output 45  
GCStatsAdmin.exe 33  
Gen 2 Engine 81-82  
Geometry Expert 58, 64, 83  
Geometry, moving 184  
Geometry, Show 68

---

Getting Started 99  
Gouge Tolerance 28  
Gradient Background 20  
Grid Brightness 24

---

## H

Halo Thickness (Color Appearance Settings) 19  
Halo Transparency (Color Appearance Settings) 19  
Hardness 54  
Hardware OpenGL 22  
Help Button 99  
Help menu 98  
high degree spline curves 25  
History 83  
Holder Class Menu Items 38  
Hole Data 38  
Hole Manager  
    menu item under Features 84  
Hole Wizard 38  
    menu item under Features 84  
Home View 66

---

## I

IDs 101  
Ignore Lathe Max Feed Moves 38  
Ignore Lathe Maximum Feedrate Moves 38  
Import 12  
Import Material 55  
Inches to mm 73  
Index, online help 103  
Insert Comments, Posting 134  
Insert Row 63  
Interface  
    Level 2 24  
    Preferences 31

---

interop moves  
    Clearance Volume 189  
interpolation  
    using Clearance Volume 190  
Invert Selection 60  
IPR 55  
Isometric View 66

---

## J

Java Runtime Environment 105

---

## L

Labels 62, 68, 75  
Launch New Application on File 39  
Length in Characters, in post 46  
Length in Feet, in post 46  
Length in Meters, in post 46  
Level 1 Interface 31  
Level 2 Interface 31  
License Options 34  
Lights Appearance Setting 18  
Line segments 77  
Lines drawn for high degree curves 25  
Lines, Select 61

---

## M

Machine Definition Document (MDD)  
    defined 12  
Machine Sim  
    preferences for Display settings 16  
Machine Sim Path 40  
Machine Space  
    Clearance Volume 190  
Machining Face Check 81  
Machining Preferences 35  
Macros menu 95  
main menu  
    Edit 58

- Features 84
- Macros 95
- Modify 71
- Plug-Ins 94
- Solids 80
- View 65
- WEDM (Wire EDM) 95
- Managers
  - accessing 88
  - customizing headers 92
  - data types 88
  - editing data 90
  - right-click context menus 91
- Material
  - Alloy Group 54
  - changing 54
  - importing 55
- Material Database 52
- Material List 54
- Material Properties 18-19
  - Color (Appearance Settings) 18
- Material, part 51
- Material.txt and CutDATA.txt 40
- Materials
  - Cut Type 54
- Materials Database file 40
- Maximum Feedrate
  - ignoring in Lathe 38
- MDD
  - defined 12
- MDD Path 40
- Minimize, Posting 134
- Mirror 71
- MM to inches 73
- MMPR 55
- Modify menu 71
- Mouse Drag, Select 60
- Mouse Position 67
- Move Part Origin 79
- MTM
  - posting 133
- Multi-Level Sort 91
- Multi-Pass Stitch 81

---

## N

- Name, select by 63
- NC program 93
- Network Licensing Option 34
- New Part Interface Level 31
- Next Operation
  - Render 139
- Non-Cutting simulation 145
- Number of Parts, in post 45
- NURBS 77, 81

---

## O

- Offset 36
- On Item, help 99
- Online Help 98
- Op Color Mode 145
- Op Manager
  - accessing 88
  - customizing headers 92
  - data types 88
  - editing data 90
  - right-click context menus 91
- Op Sim
  - Cutting Options 26
  - Display Options 23
  - Lighting 18
  - Performance 26
  - preferences for Display settings 16
- Op Sim display
  - Transparency 20
- Open Part Interface Preference 31
- OpenGL 21
- Operation Comment, in post 45
- Operation Manager
  - accessing 88
  - customizing headers 92
  - data types 88
  - editing data 90
  - right-click context menus 91
- Operation Number and Type, in post 45
- Optional Stops, Posting 134
- Options button 16

---

## P

P-Buffer Open GL 22  
Palettes, lock 31  
Panning the Part 126  
Part Space  
    Clearance Volume 190  
Partial Updates 21  
Paths  
    for VMMs, MDDs, and Machine Sim 40  
Performance, Op/Tool Sim 26  
Performance, Rendering 21  
Plane Opacity 18  
Play (CPR) 139  
Play button 139  
Plug-Ins Guide 95  
Plug-Ins menu 94  
Pocketing, Offset 36  
Points, Select 61  
Pop-up axis block 184  
    (illustrated) 184  
Post  
    Selection 133  
Post File Overwrite 39  
Post Processing Comments 44  
Post Processor button 131  
Post Processor dialog 131  
Posted Output 45  
posting  
    MTM considerations 133  
Pre-selection Properties 19  
Preferences 14, 155  
Previous View 66  
Prim 1 146  
Prim 2 146  
Print  
    Black on White 24  
    GE to Text File 83  
    Post 13  
    Screen 24

Printing 156  
Printing Preferences 24  
Process Manager 87  
    accessing 88  
    customizing headers 92  
    data types 88  
    editing data 90  
    right-click context menus 91  
Program Comment, in post 45  
Program Name, in post 45  
Program Number 134  
pseudoholder display 25

---

## R

Radial 78  
Rapid Geometry 76  
Rebuild Solid 80  
Recent Files 13  
Record Video  
    button in Simulation palette 140  
Redraw 66  
Reload Simulation 154  
Remove Unneeded Topology 81  
Render 152  
    Play 139  
Rendering colors 145  
Rendering Options 21  
Reverse All Rows 64  
Reverse Arc 71, 76  
Reverse pattern, sort 76  
Rotating the Part 126  
Rotation  
    freehand 184  
Rough Bore 36

---

## S

S pattern sort 75  
Save  
    A Copy 11  
Save Size 39



- Scale 73
  - scrollwheel 125-126
  - Search, online help 104
  - Segment Spline 77
  - Select
    - Circles by Full Diameter Range 60
    - Circles by Full Radius 60
    - Lines, All 61
    - Lines, Feeds 61
    - Lines, Rapids 61
    - Mouse Drag 60
  - Select machine assembly 150
  - Selected Op to
    - Lightwork File 83
    - Screen Points 82
    - Splines 83
    - Text File 83
  - Selected Ops, Posting 133
  - Self-Intersect Loops 80
  - Semi-transparent Drag Rectangle 17
  - Sequence From, Posting 134
  - Set Font Directory 40
  - Set Op Start/Stop #s 152
  - SFPM 55
  - Sheet Side, Toggle 78
  - Sheets, Select 61
  - Shininess 19
  - Shininess, Material Properties 19
  - Show
    - Stock & Origin 68
  - Show Clearance Volume
    - Customization 190
  - Show Op 152
  - Show Time 152
  - Shrink Wrap 79
  - Shrink Wrap Visible 79
  - Shrinkage 77
  - Side View 66
  - Sim
    - Display Options 23
  - Sim tab 19
  - Simplify 81
  - Skip Pecks 153
  - Skip unselected Op 148
  - Slider 27
  - SMPM 55
  - Soft Shadows 22
  - Software OpenGL 22
  - Solid Edges (Color Appearance Settings) 19
  - Solids menu 80
  - Sort, points and circles 75
  - Special Functions, in post 45
  - Specular 18
  - Specular, Material Properties 18
  - Speed Control slider 139
  - Splines 77
  - Start At Op 152
  - Start Maximized 29
  - Starting Program Number, Posting 134
  - Starting Tool State, in post 45
  - Statistics for Op/Tool Sim 28
  - statistics on usage 33
  - Step Forward (CPR) 139
  - Steps Per Update 21
  - Stitch Utilities 81
  - STL 153
  - Stop (CPR) 139, 144
  - Stop Before Load/Unload 152
  - Stop Before Op 152
  - Sub Number, in post 46
  - Suggest 57
  - System, Appearance Settings 21
- 
- T**
- Tablet 35
  - Tap Table 38
  - Tapered Tools 36
  - Targeting pixel tolerance 31

Test Frame Rate 23  
Text, Select 61  
Tick Delay 101  
Toggle Wall/Air 76  
Tolerance, of collision settings 28  
Tool  
    Material 55  
    Size, relating to material 54  
Tool Center 35-36  
Tool Comment, in post 45  
Tool Edge 36  
Tool Manager  
    accessing 88  
    customizing headers 92  
    data types 88  
    editing data 90  
    right-click context menus 91  
Tool Motion on Target Body 145  
Tool Number and Type, in post 45  
Tool Sim  
    Cutting Options 26  
    Display Options 23  
    Lighting 18  
    Performance 26  
    preferences for Display settings 16  
Tool Sim display  
    Transparency 20  
toolbars  
    transparency 31  
Toolpath Display 35  
Tools sub-menu 80  
Tooltips 100  
Top View 66  
ToPos 73  
Trace From Run 149  
Trace Operation 149  
Trackball  
    Customizing 159  
Translate 74  
Translation  
    freehand 184  
Transparency, Material 20  
Transparent Dialogs 30

turning toolholders  
    previewing 25

---

## U

Unwrap Geometry 78  
Unzoom 66  
Unzoom on View Change 17  
Updates  
    one-time check 101  
usage statistics 33  
Use Op Start 152  
Use Op Stop 152  
utilities  
    GCStatsAdmin.exe 33

---

## V

Vertex Arrays 21  
Video, outputting 140  
View Animation Speed 18  
View menu 65  
Visible WGs 71  
VMM Path 40

---

## W

Wall Angle Tolerance 32  
Wall geometry 76  
Walls From Selected Edges, Select 62  
WEDM menu 95  
WG Manager  
    accessing 88  
    customizing headers 92  
    data types 88  
    editing data 90  
    right-click context menus 91  
Willemin 508MT  
    and Clearance Volume 189  
Window Size 29  
Window Snap Positioning 30  
Wire Drawing 24

---

Wire EDM menu 95

Wire Frame 24

Workgroup Comment, in post 45

Workgroup Manager

- accessing 88

- customizing headers 92

- data types 88

- editing data 90

- right-click context menus 91

Wrap Geometry 78

---

## Z

Zoom 67

Zooming the part 126