GIBBSCAM | 2027 CAM for Production Machining

Version 2027 : September 2026

Getting Started

GIBBSCAM

Contents

Introduction 5
Welcome 5
About The System 5
About Getting Started 5

GibbsCAM On The Web 6

Interface 7
About the Software 7

Quick Start 9
Basic Steps for Using the System 9
1. Create or Open a Part File 9
2. Make or Modify a Model 10
3. Create Operations 10
4. Render the Part 11
5. Post the Part 11

Using The System 13
Launching the System 13
Opening or Creating a Part 13
Opening Parts 14
Creating New Part Files 15
Importing Part Models 16
Setting Up A Part - The Document Control Dialog 16
Top half of DCD dialog 18
Bottom half of DCD dialog 21
Default Type and MDD Parts 23
Geometry Creation 24
Geometry Palette 24
Creating Geometry 25
Connecting Geometry 26
Workgroups 27
Level 1 Interface 27
Level 2 Interface 27
Workgroup List 28

Workgroup dialog Right Mouse Menu 28
Coordinate Systems 30
Coordinate System List 30
Coordinate System Palette 31
Creation of 3D Bodies 31
Creating Solids Example 32
Workgroups and Coordinate Systems in Solids 33
Tool Creation 33
Tool Creation Dialogs 34
Tool Tile List 38
Dragging Multiple Tools 39
Tool List Context Menu 39
Tool Manager 43
Process Creation 43
Creating a Process 44
Applying the Same Machining Function to Multiple Processes 45
Clearing the Process List 45
Process Tile List 46
Multiple Process Programming 46
Process List Context Menus 47
Process Manager 48
Operations 48
Defining the Cut Shape 50
Process becomes Operation 51
Operation Tile List 52
Context Menu (Right-Mouse Menu) 52
Operation Tile Stacking 53
Operation List Context Menu 53
Operation Manager 55
Operation Data 55
Operation Modifiers 58
Using Operation Modifiers 58
Modifying Operations 59
Sorting Operations 60
Multi-Level Sort 61
Printing the Toolpath 62
Cut Part Rendering 62
Post Processing 62
Posting Step-By-Step 63

Communications 67
Protocols 67
Communicating with a CNC 68

Conventions 70
Text 70
Graphics 70

Links To Online Resources 71

Appendix 72
Glossary 72

Introduction

Welcome

This *Getting Started* guide provides you with an overview of GibbsCAM and its use and interface. A lot of general information about the software is discussed in this document. As such, we highly recommend that you review this material before moving on to more specific topics such as Geometry Creation, Advanced CS, SolidSurfacer or MTM.
Note: The capabilities and user interface described in this and other guides apply to GibbsCAM Industrial Edition with all product options licensed and active. GibbsCAM Viewer, GibbsCAM Operator and GibbsCAM Student Edition provide a subset of the full functionality.

About The System

Please be advised that GibbsCAM security is activated via the Internet. If you do not have Internet access available, you may activate security via e-mail or regular mail.
Please allow adequate time to have the license file sent to you.
Please contact your reseller for support.

About Getting Started

This document is designed to give you both a quick start to using GibbsCAM and also a more detailed overview of the system in general. Whether you want to jump right into using the software or read the manuals cover to cover, this book can be of great help.
Among the contents of the book are:

- Interface : Explains all of the common elements of the system. It does not matter whether you are a new user or a seasoned power user, this section will help you get acquainted with GibbsCAM and its many changes. It will help you understand the software and might answer the question, “What am I looking at?”
- Quick Start which introduces the process or steps to using GibbsCAM. This can be thought of as an overview of using GibbsCAM.
- Using the System which goes into greater detail than the Overview does.: Goes through the process of making a part from start to finish. You will still want to read the individual product guides for the greatest understanding of a particular product, but this section will give you a head start.
- Appendix: Includes a Glossary of terms used in GibbsCAM and in the industry. You might want to look through the glossary to learn whether we are speaking the same language. The Appendix also includes answers to frequently asked questions.

GibbsCAM on the Web

For more information and help, find us on the World Wide Web.

https://online.gibbscam.com
Official GibbsCAM website

https://www.gibbscam.com/support
GibbsCAM technical support

https://forums.gibbscam.com
GibbsCAM discussion forums

http://www.facebook.com/gibbscam
GibbsCAM's Facebook page

https://www.youtube.com/@gibbscam
Product videos and information

http://twitter.com/gibbscam
Follow us on X (formerly called Twitter)

Other useful contacts:

http://www.autodesk.com/inventor
External website that provides more information on Autodesk Inventor products.

http://www.Volumill.com
External website that provides more information on VoluMill and VoluTurn Ultra High-Performance Toolpath (UHPT) from Celeritive Technologies.

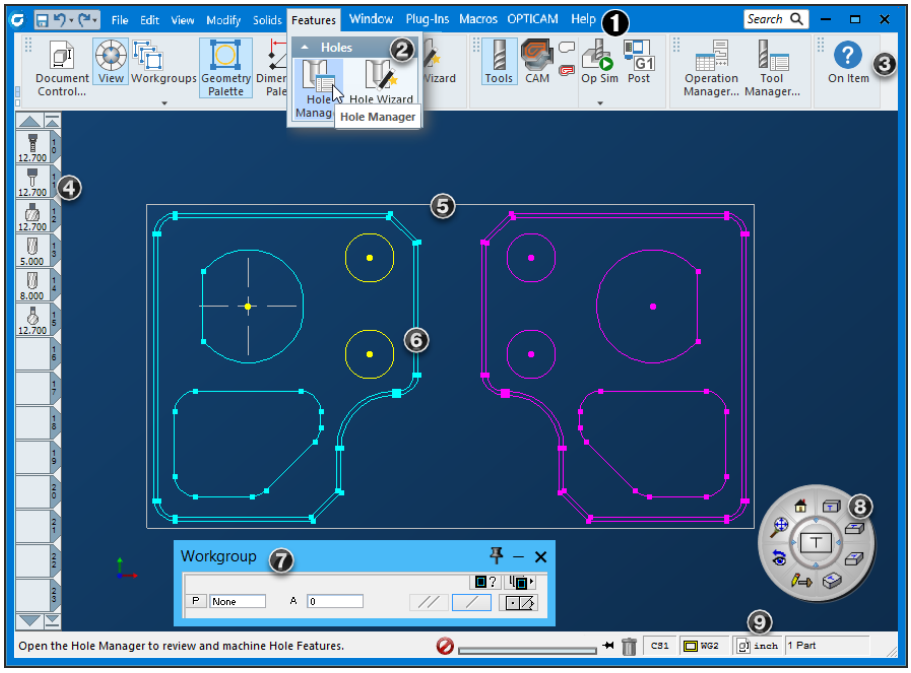
https://www.camtek.de/opticam
External website that provides more information on OPTICAM EDM from Camtek GmbH.

http://www.predator-software.com
External website that provides more information on a CNC editor and a virtual CNC viewer from Predator Software, Inc.

Interface

About the Software

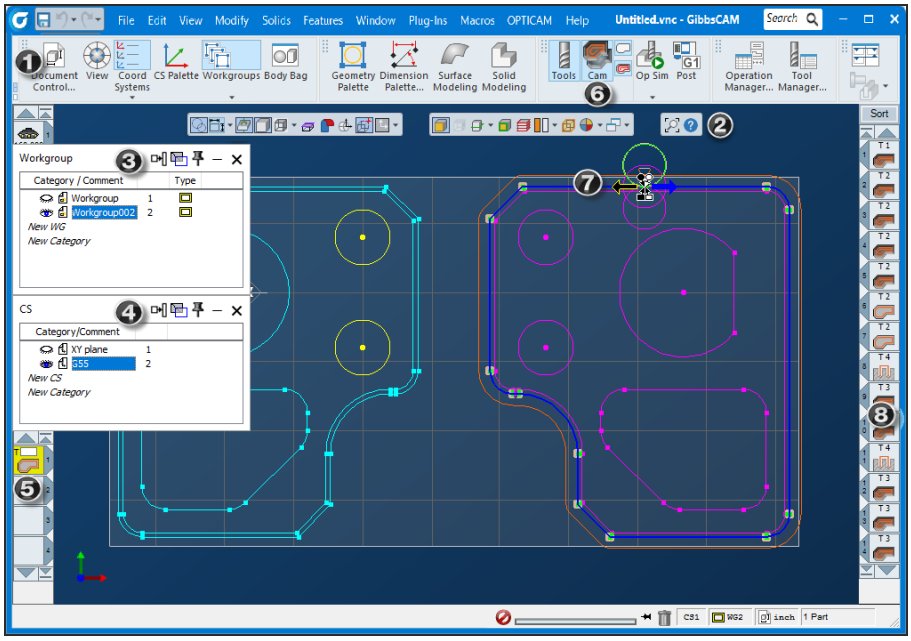
GibbsCAM supports Windows 11. This means, among other things, we follow user account control guidelines, support x64 Versions of Windows and support concurrent user sessions. GibbsCAM no longer supports 32-Bit Windows.
OpenGL is integrated into the software. OpenGL is a 3D library that provides the system with fast hardware accelerated rendering for high quality rendering, true color display, material properties, advanced light sources, user-definable colors, animated view transitions, shadows, transparent planes and user-defined OpenGL configurations. All of this translates to improved display quality and more user control and customization.
The GibbsCAM software has two interface levels. Level 1 Interface is deliberately clear and uncluttered but still displays the Main menu, Tool, Process and Operation Lists and toolpath.



1. Main Menu
2. Pull-down menu
3. Command Toolbar
4. Tool List
5. Workspace
6. Geometry
7. Palette with WG info
8. View Control palette
9. Trashcan

Level 1 Interface

The Level 2 interface is the most advanced interface. It is recommended that most users work in this interface level as it is optimized for accessibility and power. Every capability of the system is available from the Level 2 interface. Level 1 contains subsets of the features in level 2.



1. Command Toolbar
2. Floating Toolbar
3. Workgroup List
4. Coordinate System List
5. Process List
6. Machining
7. Toolpath
8. Operations List

Level 2 Interface, with some elements also available in Level 1

Quick Start

GibbsCAM software is easy to use. Here, we provide a quick overview of the process from part creation to completion. Although GibbsCAM imposes no set sequence to creating a part, some things precede others. For example, you must have a file to work on before you can make a part; a Tool list must be defined before toolpath can be generated. However, once you have done something in the system, you can change something (like a tool or stock condition) and effortlessly update the entire part. Here are general guidelines for creating a part.

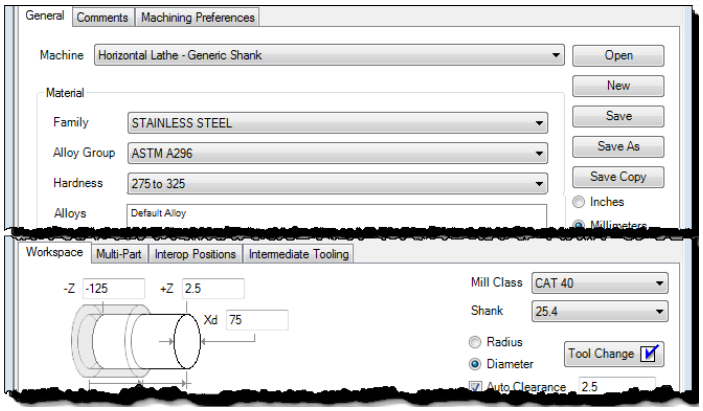
Basic Steps for Using the System

There are five basic steps to using GibbsCAM.

- 1. Create or Open a Part File
- 2. Make or Modify a Model on page 10
- 3. Create Operations on page 10
- 4. Render the Part on page 11
- 5. Post the Part on page 11

1. Create or Open a Part File

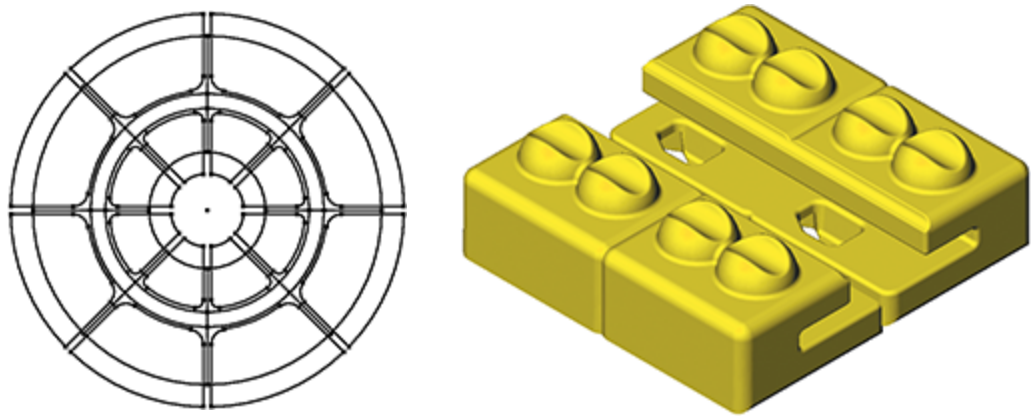
You can either create a new part file or open an existing model. This is often done with the **Document Control** dialog (DCD), whose icon is in the Command Toolbar at the top left of the workspace. The DCD provides controls for file management, part measurements, comments and preferences, stock setup, and the like. For more information, see "Setting Up A Part - The Document Control Dialog " on page 16.



DCD (Document Control dialog) for a Turning part

2. Make or Modify a Model

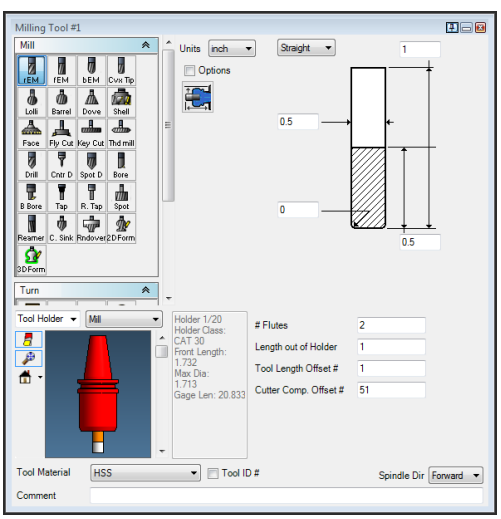
Next, you need to make a model of your part, or to modify an existing model. Modeling is accomplished with the Geometry Creation Palette, with Solid Modeling, or a combination of the two. For more information see the *Common Reference* and *Geometry Creation* guides.

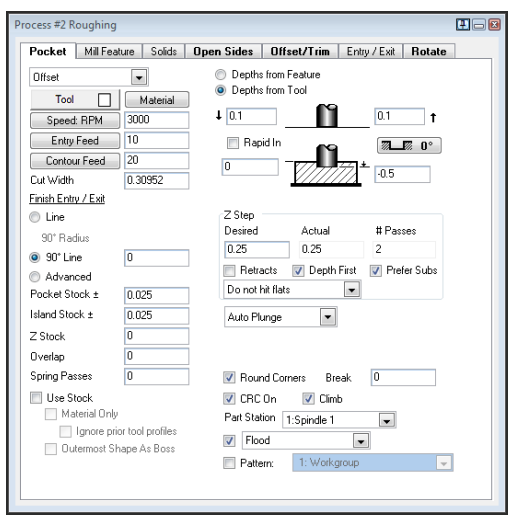


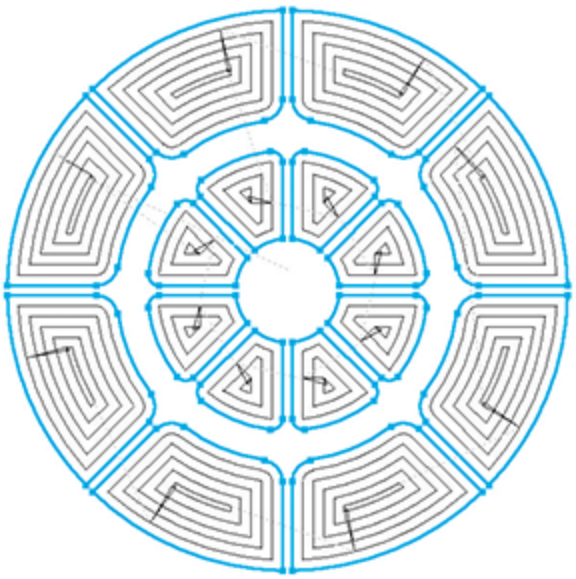
Part Models: Geometry and a Solid

3. Create Operations

Once a model is created, you apply machining functions to the part. This includes setting the available tools , defining processes, and creating operations. Creating tools can be done at any time when a part file is open. Processes and operations are created after tools are defined. For more information, see the *Common Reference*, *Mill*, and *Turning* guides.



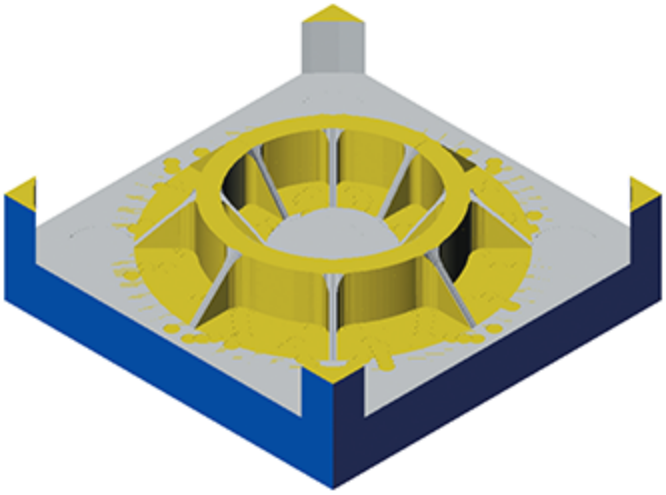




A Tool, a Process, and Resulting Toolpath

4. Render the Part

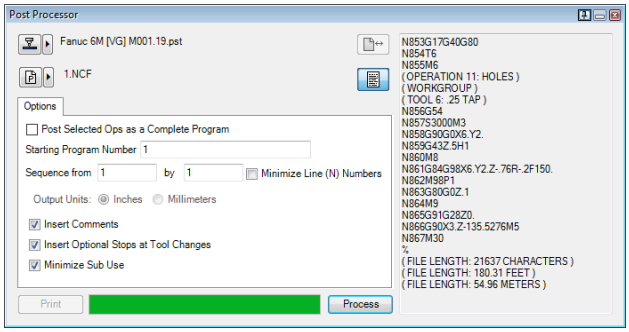
Once operations have been defined for machining a part, the part should be rendered. This provides a visual check of the part to ensure that the results are as expected.
Many errors can be caught using Cut Part Rendering (CPR) or Simulation. It can also be very useful to run the rendering as you create operations, rather than after you are finished and are about to post the part. For more information, see the *Common Reference* guide.



Cut Part Rendering (CPR) Is Run on a Part

5. Post the Part

The last step is to generate the output from the part file and create NC-code. This is called Post Processing. For more information, see “Posting Step-By-Step ” on page 63.



Posting a Part

Using the System

The following sections provide a more detailed description of your GibbsCAM software. This includes
Launching the System, next
Opening or Creating a Part
Geometry Creation,
Tool Creation,
Creating a Process ,
Operations,
Post Processing.

Launching the System

GibbsCAM products can be launched in several ways.

- Launch the application through a desktop shortcut.

  The installer creates a desktop shortcut. This is the easiest way to start the software.

- Use the Start menu to launch the application.

  Click the Start menu, go to All Programs, then the GibbsCAM folder, then GibbsCAM v2027.
  Click the mouse button again to launch the program.

- Double-click a part file.

  Double-clicking a part file will also launch the system.

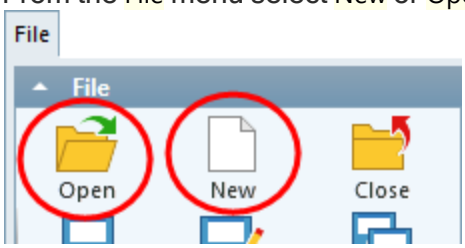
- Windows Search Bar.

This is perhaps the easiest method if you are unable to locate the application shortcut. Open the Start menu and start typing the application name, for example “gi”, and a list of files and programs instantly appear that begin with those letters allowing you to select the exact item you are searching for.

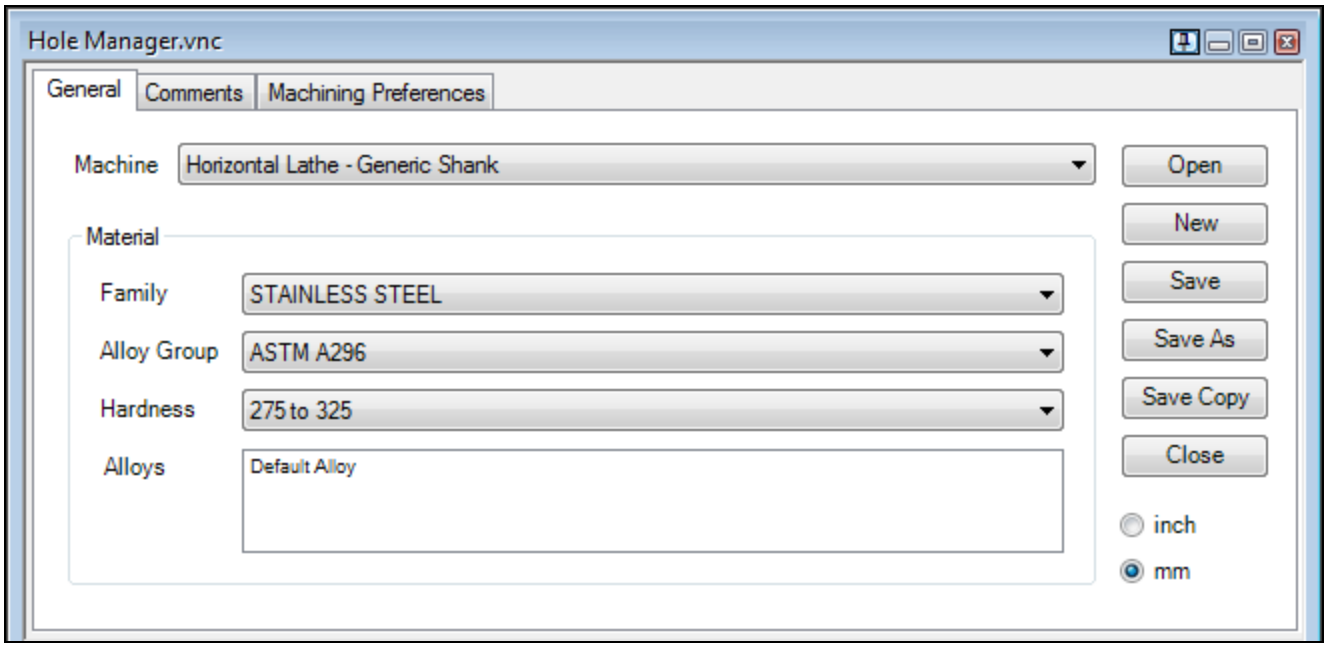
Opening or Creating a Part

There are four ways to create a new part or open an existing part.

1. From the File menu select New or Open.



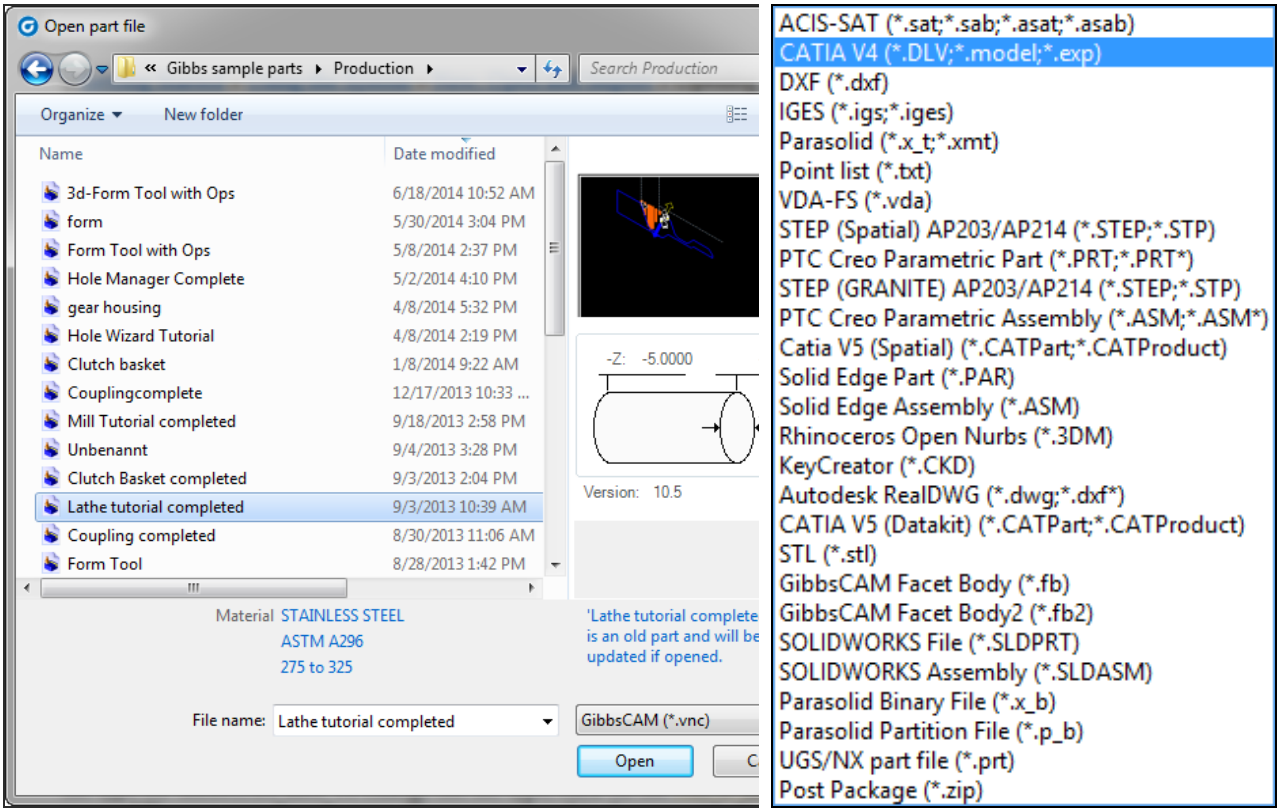
2. Open the Document Control dialog by clicking on the Document button in the Command Toolbar.
   The top section of the Document Control dialog has buttons for creating a new part (New) or opening an existing part (Open).



3. Press `Ctrl+N` or `Ctrl+O` on the keyboard.
   Pressing `Ctrl+N` will create a new part. Pressing `Ctrl+O` will bring up the Open dialog. Learning the system's keyboard shortcuts can be a great time saver.
4. Drag and drop a file icon onto the application window or a desktop shortcut.

Opening Parts

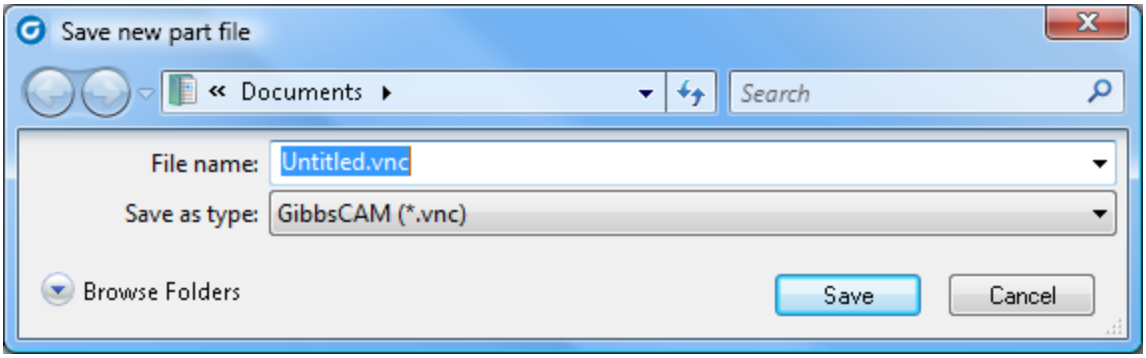
When the Open command is selected, the Open part file dialog will appear. The initial location is "My Documents" or the directory that was last used when opening a part. The dialog displays a picture of selected VNC files and provides some basic information about the part, such as its size and material.



The Open dialog will only display files of the type you tell it to look for. By default this setting is VNC files, the native file type for GibbsCAM files. To change the file type, click on the Files of type pull-down menu. This will display the types of files you may directly open. Please note that this changes with the options you have purchased.
Optional ways of opening a part are: `double-clicking` a part file and `dragging` and dropping in the workspace or a shortcut icon.

Creating New Part Files

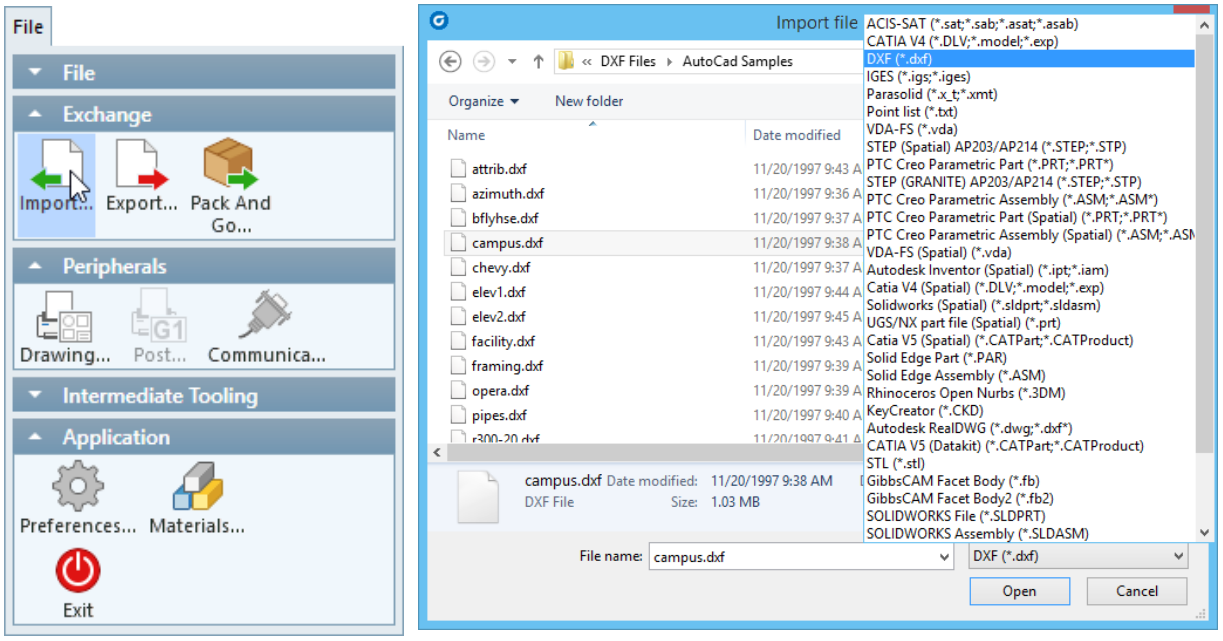
When one of the three methods of creating a new part has been selected, a dialog will come up. This allows you to give the new file a name and select where it should be saved. Enter the name of the part in the File Name text box and navigate to where the file should be saved. Click on Save.



Now that a new part file has been created, the part must be set up. The units of measurement, stock size, type of part and clearances must be set. This is accomplished through the **Document**

Control dialog (DCD). For more information on the DCD, see Setting Up A Part - The Document Control Dialog .

Importing Part Models



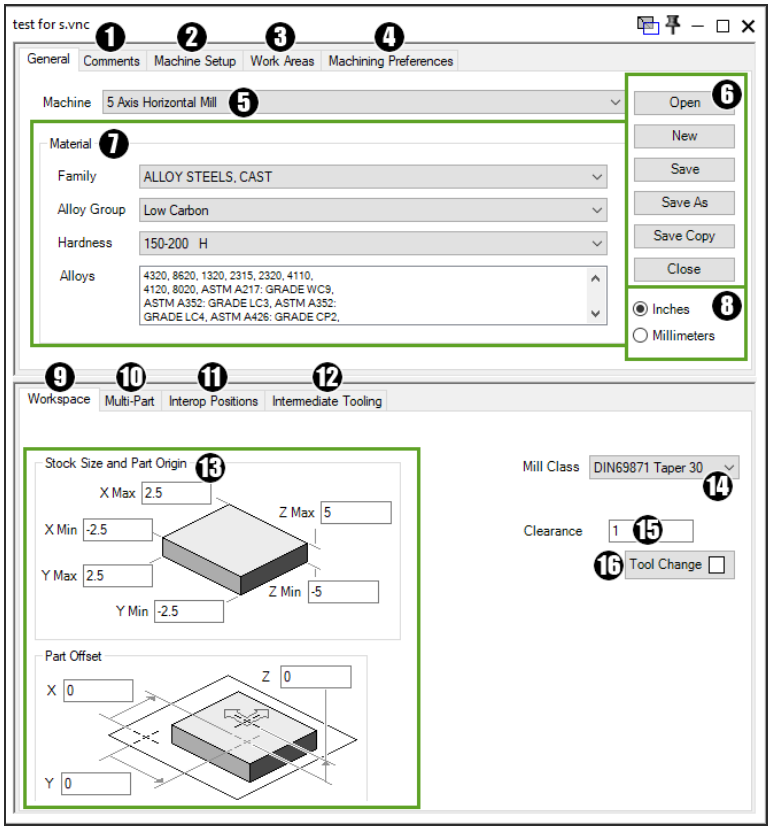
Importing allows an existing saved part to be added to a current part. The current part file may be new, with no tools or geometry or the part may be an existing part full of geometry, solids, tools and operations. Using the Import function is a good way to merge two similar parts or add fixtures and jigs.

To import a model into the part file click on the File > Import... option in the main Menu. The Import file dialog appears, allowing navigation to the file being imported. Choose the directory that the file is in and change the Files type choice to the type of file that is being imported. When the file is shown in the list, double-click it or choose Open. This will bring the existing part and its settings into the Workspace.
The Import dialog will only display files of the type you tell it to look for. To change the file type, click on the pull-down menu. This will display the types of files you may directly import. Please note that this changes with the options you have purchased.

Setting Up A Part - The Document Control Dialog

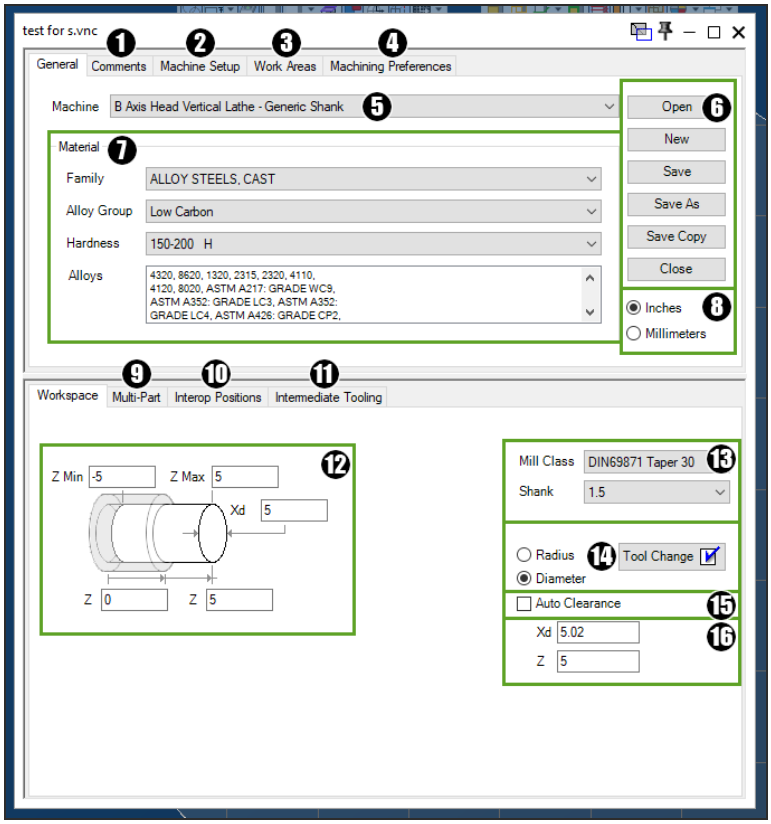
Setting up a part refers to defining the units of measurement and size of the part, the type of machine that will cut the part, the material the part will be made from and setting the part's clearances. All of this data is entered in or selected from the **Document Control** dialog (DCD). The DCD is accessed by clicking on the Document button in the Commands palette.

The lower part of the dialog differs according to machine type and other factors. It is discussed in more detail in the guides for *Mill* and *Turning*.



1. Comments Tab
2. Machine Setup tab: see the *Mill* guide.
3. Work Areas
4. Machining Preferences Tab
5. Machine Type
6. File Control buttons
7. Materials: see the *Common Reference* guide.
8. Unit of Measurement
9. Workspace tab: see the *Mill* guide.
10. Multi-Part tab: see the *Mill* or *TMS* guide.
11. Interop Positions ; see below.
12. Intermediate Tooling tab: see the *Common Reference* guide.
13. Stock size and Origin
14. Tool Holder Basics
15. Clearance Plane
16. Indicates whether the Tool Change checkbox is selected

Document Control dialog for a generic 4-axis vertical mill



1. Comments Tab
2. Machine Setup
3. Work Areas
4. Machining Preferences tab: see the *Common Reference* guide.
5. Machine Type
6. File Control buttons
7. Materials: see the *Common Reference* guide.
8. Unit of Measurement
9. Multi-Part tab; see the *Mill* or *TMS* guide.
10. Interop Positions
11. Intermediate Tooling tab: see the *Common Reference* guide.
12. Workspace tab: see the *Turning* guide.
13. Tool Holder Basics
14. Radius or Diameter display
15. Clearance Plane
16. Fixed Clearance positions

Document Control dialog for a generic horizontal lathe

Top half of DCD dialog

General Tab

Machine Type

Click to open the Machine pull-down menu which lists all the available types of machines. This may include 2- and 3-axis lathes and 3-, 4- or 5-axis mills, both horizontal and vertical. The actual contents of the menu will vary depending on the options installed. Select the type of machine that will be used to cut your part.

File Control buttons

This set of buttons provides access to creating a new part, opening and closing an existing part, and saving the current part. The part may be saved under a different name or location by selecting Save As. A copy of the part may also be saved using the Save Copy button. Save Copy will append “copy” to the file name so the original and copy can be saved in the same folder. You can use Save Copy to save a part to an older version of GibbsCAM. Select the version you wish to save the part in, from the Save As Type pull-down menu.

Part Material

The material the part will be cut from may be selected here. A database of materials and recommended cut speeds and feeds for each entry can be created or edited here. There is also the option of purchasing CutDATA™ material library. For more information on Materials, see the *Common Reference* guide.

Unit of Measurement

Select whether the part will be defined in inches or millimeters. This setting may be changed at any time but any existing geometry, tools and operations will have to be updated to match the change. Please note that solids will automatically scale to match the part. This does not affect posted data as post processors are either inch or metric and will convert values as needed. Also, solids and sheets are unaffected by changing the measurement units. Tools of either unit can be created.

Comments Tab

Any text entered as a part comment will be shown in the part preview of the Open dialog and will appear in posted output. An unlimited number of plaintext Unicode characters can be entered. The user is responsible for ensuring that this field contains only post-compatible characters.

Machining Preferences Tab

Spline Machining Tolerance

This is the curve height accuracy used when machining splines/curves or when using 2D Form or 3D Form tools. Curves are machined as a series of small straight lines. This value is the maximum amount these lines may deviate from the true curve.
Important: The value for Spline Machining Tolerance is used by 2D Form tools and 3D Form tools, which typically use free-form curves (spline geometry) in their construction.

Entry Line Approach

Used when specifying an entry line and arc for Contouring or Pocketing. Tangent Entry Line to Arc keeps the line tangent to arc. Normal entry line to Arc keeps line perpendicular to arc - recommended if you prefer CRC from tool edge. Line length should be greater than or equal to tool radius.

Mill/Turning CRC Type

Select Toolpath type for Ops. In Tool Center, the toolpath is calculated to tool centerline, including any stock allowances. For Tool Edge, toolpath is to the tool edge, including stock allowance. For Finish Profile, toolpath is at finish geometry and does not include stock.

CRC Entry Line Validation

When checked, this will enable the validation check for the CRC entry line. The check will determine if the CRC entry line is greater than zero in length.

Omit Small Chord Arcs

When this checkbox is selected, toolpath arcs that are nearly identical (specifically, arcs whose chord height from a line with the same end point is less than 0.0001 inches) will be output as a single line move.

Oriented Toolpath

Affects operations that use the Multiple Orientations checkbox in the Linear Broaching process dialog. When part and tool rotate with respect to each other and the tool must move to arrive at the next broach position, you can specify when the tool rotates to the needed orientation for the next broach. For complete information, see the *Broaching* guide.

- Before the move:
  Is supported on all machines. Tool and/or part rotate to the correct orientation and then move to the correct position.

- With the move:
  Is the fastest of the options, but it requires a type of interpolation that is not supported on all machines. Tool and/or part move and rotate simultaneously.
- After the move:
  Is supported on all machines. Tool and/or part move to the correct position and then rotate to the correct orientation.

Use Global Settings for Solids

When checked, applies default tolerances to machining processes. These settings can be overridden for specific operations using the Advanced Settings option in the process dialogs.

Part Rough Tolerance

When Tolerance:Rough is selected in a process dialog, this value is used for the machining tolerance. This value is also used for the stock tolerance.

Part Finish Tolerance

When Tolerance:Finish is selected in a process dialog, this value is used for the machining tolerance. This value is also used for the stock tolerance.

Fixture Tolerance

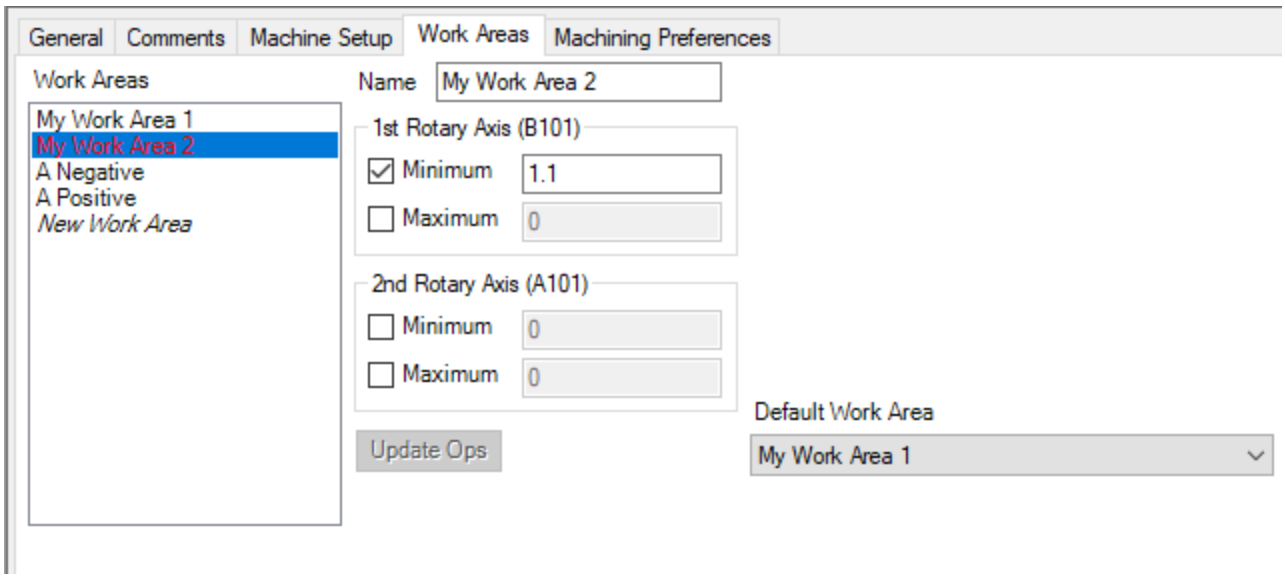
This value is the tolerance used if toolpath encounters a body designated as a fixture. The tolerance is applied to the clearance tolerance value.

Fixture Clearance

This is a clearance tolerance applied to fixtures encountered when creating toolpath.

Work Areas Tab

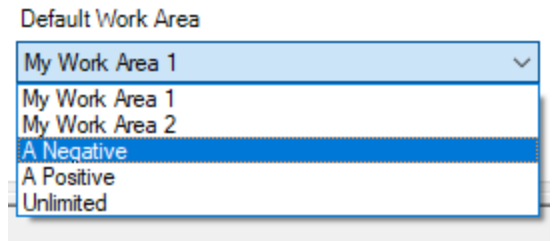
The Work Area does not override the machine hard limits, but rather, sets a preferable soft limit. The work area is designed to give the system a hint or suggestion on which solution to perform (when multiple solutions exist.)
The work area does not change the actual toolpath generated by the machining operation: just the way the tool approaches the work piece. The system will ignore work areas if they are outside the machines hard limits defined in the MDD.



When a machine has two or more rotary axes, the Document Control Dialog (DCD)'s upper half has a new tab, **Work Areas**. This lets you view work areas defined in your MDD and, for each

part, you can create work areas with user-specifiable values for such parameters as minimum and maximum angles for rotary axes. If you specify a default Work Area, it is used for all newly created operations.
Effect on Operations.If you change a part-defined Work Area, the Update Ops button becomes available, allowing you to update any operations that were assigned to that Work Area before your changes. If multiple identical Work Areas are defined, all but the first display red to inform you that these Work Areas cannot be matched.

In parts that have Work Areas defined, the **Op Data** dialog provides a Default Work Area dropdown.



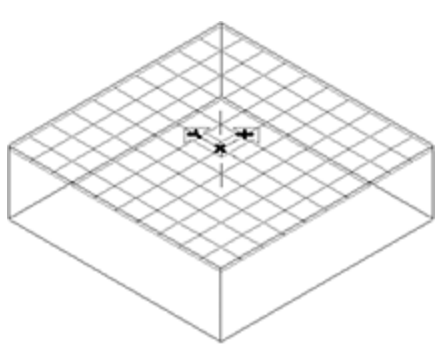
Machine Setup

This option is discussed in more detail in the *Mill* guide and *Turning* guide for B-Axis lathes.

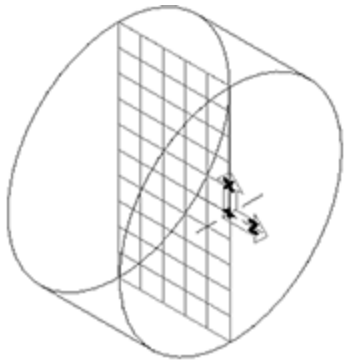
Bottom half of DCD dialog

Workspace (Mill or Lathe) Tab

For more details on this tab, see the *Mill* or *Turning* guides.



Mill stock



Lathe stock

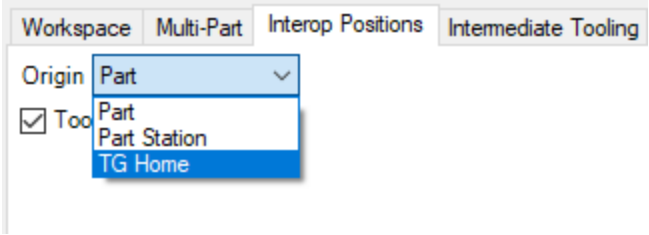
Multi-Part Tab

For more details on this, see the *Mill* or *TMS* guides.

Interop Positions

Interop Positions Settings - User Toolchanges

On machines supporting User Toolchanges, you can now select the origin for the toolchange on the **Interop Positions** tab of the DCD.



- Part is default behavior from before GC2023. It defines the toolchange location relative to the part origin. This is useful in many cases, such as close turret rotation on a typical lathe setup.
- Part Station defines toolchange location relative to the part station origin without the part offset.
- Machine (not available for Generic MDDs) defines toolchange location relative to the machine root, which lets toolchange occur in the same machine location regardless of the part setup.
- TG Home defines the toolchange location relative to the toolgroup home location. Note that selecting an alternate origin changes the meaning of the input coordinates but does not affect output by default. The postprocessor may also choose to change the output mode based on your origin selection; this will require a post modification

Interop Positions Settings - MDD

For any generic MDD, or for any custom MDD that specifies a Flow Axis Set (FAS) with an Interop Event Location whose axes are set to User, the **Interop Positions** page lets you decide whether or not to specify tool change positions for parkable axes.
Additional controls are offered, depending on settings in the MDD.

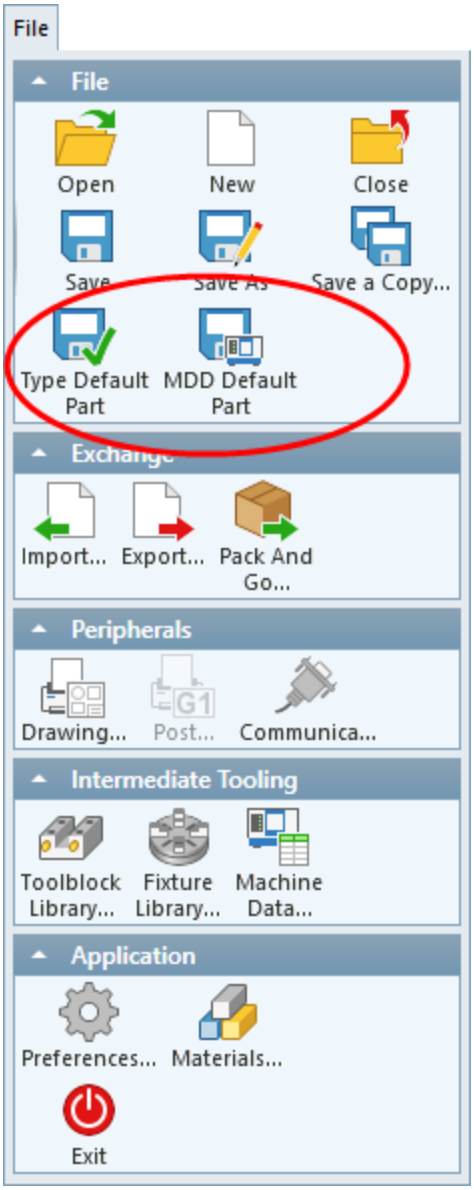
- If the MDD's **Machining Preferences** page has Show Tool Change Check Box selected, then the DCD's Interop Positions page will offer a Tool Change checkbox. Selecting this checkbox displays an Axis / Value table that allows you to specify the tool change position for each listed axis.
  Note that for a radial axis, such as the X in a simple XZ lathe, the value you supply is a radial tool change position, not a diameter value.
- If the MDD's **Machining Preferences** page has the Force Share User Axis Values checkbox selected, then the Axis / Value table will cause the specified values to be shared automatically across all matching axes in each Flow Axis Set (FAS).
- If the MDD does not force sharing of user axis values, then the end user has control:
  - If you select the Share User Axis Values checkbox, the values you specify are shared across all axes in every FAS, as noted above.
  - If you deselect this checkbox, then a pull-down menu appears that allows you to set user axis values for each FAS Interop Event Location.

In addition to specifying the position of the turret when tools are changed, the Document dialog provides the user with two options for handling part clearance: Auto Clearance or Fixed Clearance. The selection made will determine how the system will calculate positioning moves between operations.
This enables you to set tool change positions.

Intermediate Tooling Tab

Enables detailed setup of toolblocks and fixtures. In the *Common Reference* guide, see the section on Intermediate Tooling.

Default Type and MDD Parts



Default settings and prepared tool lists may be automatically applied when creating a new part. This includes information such as tools, processes, operations, geometry and custom stock. This capability works for Machine Types and MDD's (Machine Definition Documents). An MDD must be available for each machine that will be used within Gibbs. This is because each machine has different parameters and abilities.

To enable Defaults for machine types or MDD's, create a new part with the desired machine type or MDD with the tools, stock and other settings and choose File >

Type Default Part or MDD Default Part. You can override these defaults by resaving the default type or MDD. Once a default is created, each time a new file is created it will have the very same data as the default file.

Switching Between Defaults

The default settings are determined by the current MDD selection in the Document Control dialog. Close any open part files and choose a machine type from the MDD list and create a new part to apply the defaults for a different MDD.

Geometry Creation

For more comprehensive information about Geometry Creation, see the Geometry Creation Guide. The following provide a general overview of creating geometry within GibbsCAM.

Geometry Palette,
Connecting Geometry,
Workgroups,
Coordinate Systems.
Creation of 3D Bodies

Geometry Palette

To create and modify geometry, you use the Geometry Creation palette. The Geometry Creation palette has seven buttons that lead to sub-palettes for creating geometry, one button that connects and disconnects geometry and one button that leads to a dialog–the Geometry Expert. Each sub-palette contains a set of buttons that display a dialog or perform a function.

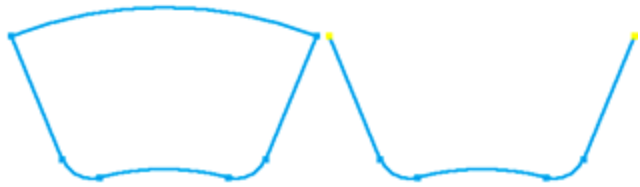
1. Points button
2. Lines button
3. Circles button
4. AutoShapes button
5. Splines button
6. Chamfer/Fillet
7. Geo From Solids
8. Connect/Disconnect
9. Geometry Expert

The Geometry Creation palette

Geometry is essential to machining a part. Without it, there is nothing to machine unless you have the SolidSurfacer option. Points are used for drilling, tapping and thread milling. Lines and circles are used for roughing, pocketing and contouring.
You can create geometry in four ways:

- Text input through a dialog,
- Copy and paste from existing geometry,
- Draw by hand with a mouse or a digitizing tablet. You can only create lines and points by hand, other features require dialog input.
- Imported from a file.

You can combine geometric features into complex shapes. Shapes can be “open” or “closed.” A closed shape is a shape that has no discernible end, such as a rectangle. An open shape has ends and does not connect to itself. An example is three sides of a rectangle or a line connected to an arc.

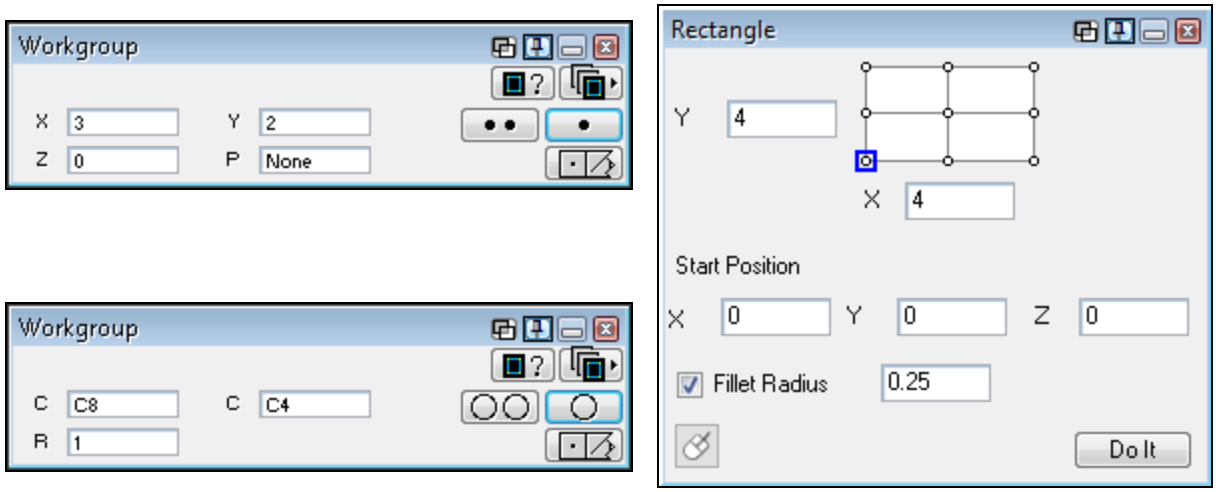


An example of a closed shape and an open shape

To connect two features, such as a line and a circle, a point must be created where the features intersect. The point is referred to as a connector. Points are also referred to as terminators, or terminating points in some cases. The points at the ends of an open shape are referred to as terminators.

Creating Geometry

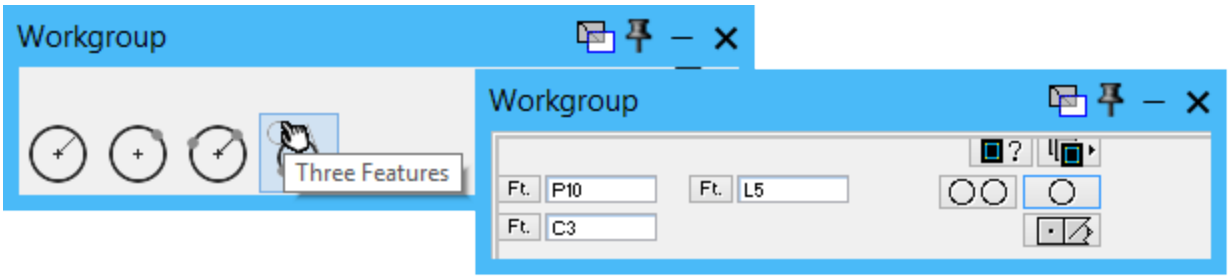
There are three basic types of dialog input for geometry. First is the entry of specific coordinates in text boxes. Second is the selection of a specific feature to be used as a reference, such as tangent to a circle or a point. Third is specifying the dimensions and location of a final shape, such as a rectangle.



The three basic types of geometry dialogs

Once a dialog has enough information to create the type of feature specified, the Do It or single feature button will become available. Click on the button or hit `Enter` on your keyboard to create the feature. Shape dialogs will remain open after a feature is created while the palette will return to the Geometry Creation palette unless the multiple feature button is clicked.

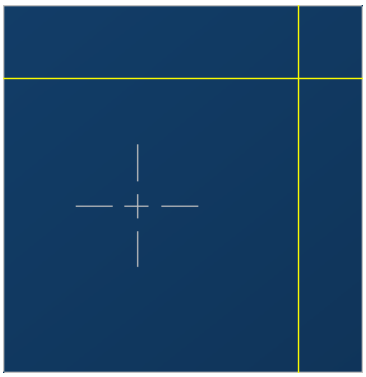
An unseen function of the Geometry Creation palette is its ability to create geometry without having data entered into dialogs. If geometry in the Workspace is selected and provides enough information to define a feature, clicking on that feature in the geometry creation palette will open the appropriate creation dialog. For example, selecting three points in the Workspace and clicking on the Circle button in the Geometry Creation dialog will open the Three Features Circle sub-palette with all of the data loaded and ready to be created.

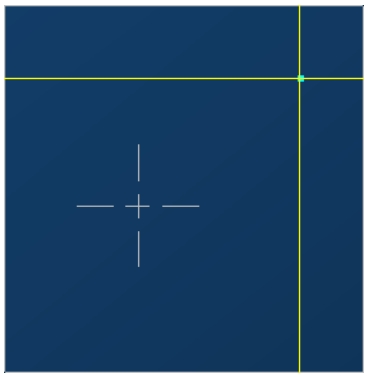


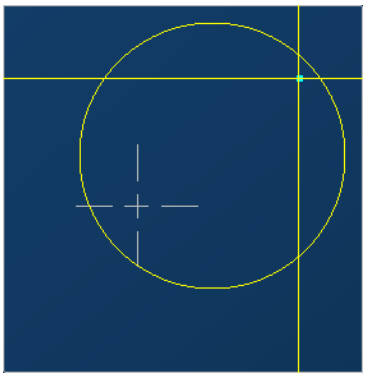
Connecting Geometry

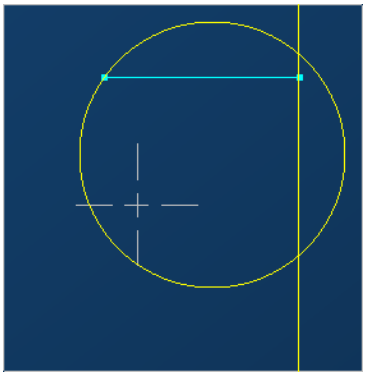
The following figure illustrates how geometry is connected.

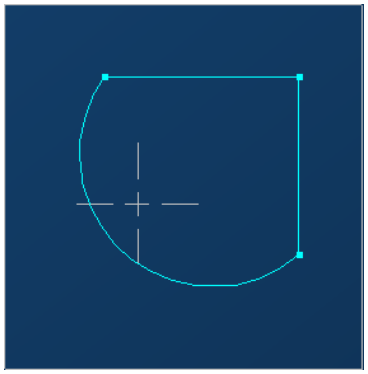
1. To begin with there are just two unconnected lines, one vertical, one horizontal. Both lines are unterminated.
2. A connecting point is then added by selecting both lines, then clicking on the Connect/Disconnect button. The lines are still unterminated but connected.
3. A circle is then added.
4. In the fourth image, the circle and horizontal line are selected then clicking the Connect/Disconnect button. A connecting point is added to connect the circle to the horizontal line, and the line is terminated.
5. The circle is then connected to the vertical line and a closed shape is created. A closed shape has no start or end point and effectively creates a loop of some shape.
6. In the sixth image, the vertical line is deleted and the connecting points are changed to terminating points by clicking on the Connect/Disconnect button. For more information see the *Geometry Creation* guide.

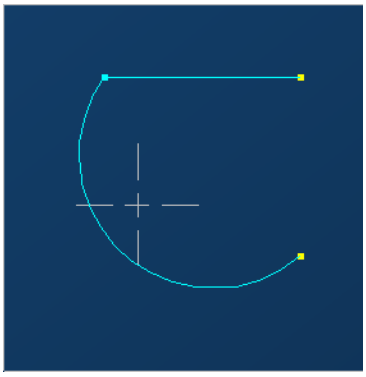

1


2


3


4


5


6

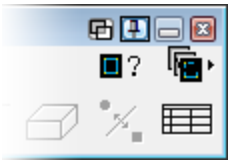
Example of Connecting and Terminating Geometry

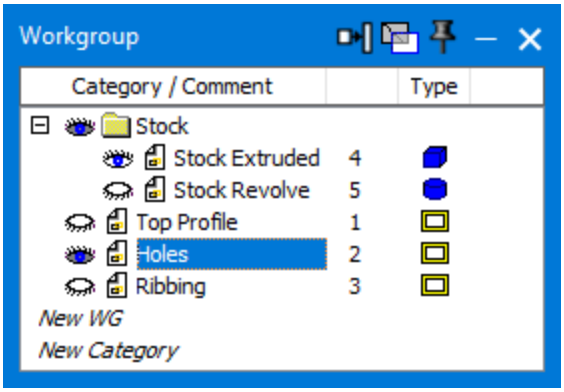
Workgroups

Workgroups are separate layers used to separate different groups of geometry, including custom stock. This can be of great help in keeping the Workspace uncluttered. While the geometry for more than one workgroup may be viewed, only the geometry in the current workgroup may be modified. All geometry created will be in the current workgroup. Geometry in background workgroups will be colored gray and cannot be selected.

Level 1 Interface

Workgroups are accessed through the Geometry Creation palette. There are two buttons on the top right-hand corner of the palette. The left button opens the workgroup list, and the right button brings up the workgroup dialog to quickly switch the current workgroup.

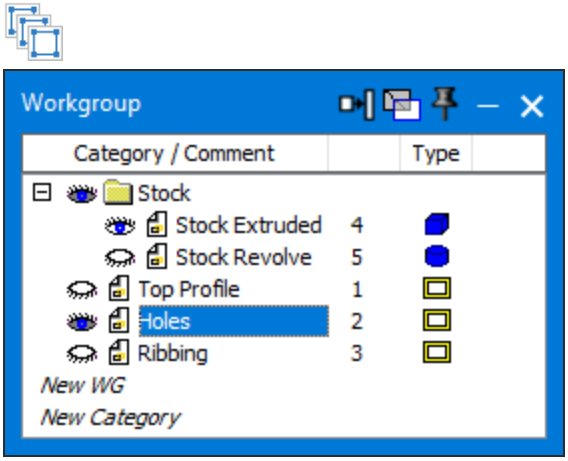


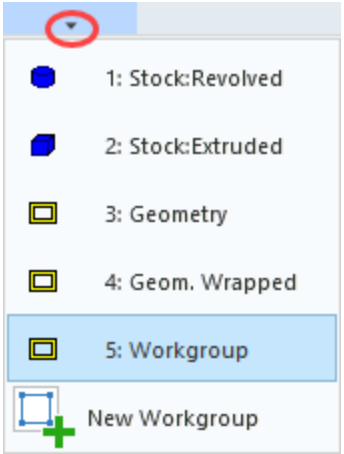


Level 2 Interface

Workgroups are accessed from the Command Toolbar. If you click the top of the Workgroup command the Workgroup dialog opens (Below left). Clicking the down arrow opens a quick selection list (Below right).

Workgroup List

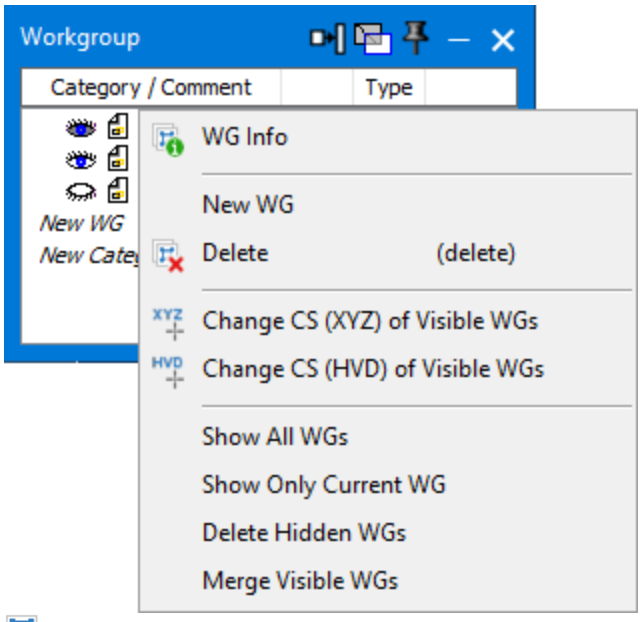




When a new part is created, there is only one Workgroup. Add more Workgroups by clicking `New WG`. The current Workgroup is highlighted in blue. Clicking a column title in the workgroup dialog will sort the Workgroup display.
Add a Workgroup category by clicking `New Category`. Drag and drop workgroups into the category as needed. Hide or show the contents of the category by clicking the `-` or `+` icon by the category folder.
The **Workgroup** dialog has five columns: Show/Hide Category; Workgroup number; Eye status display; (Show/hide Workgroup) Comment; and Type.

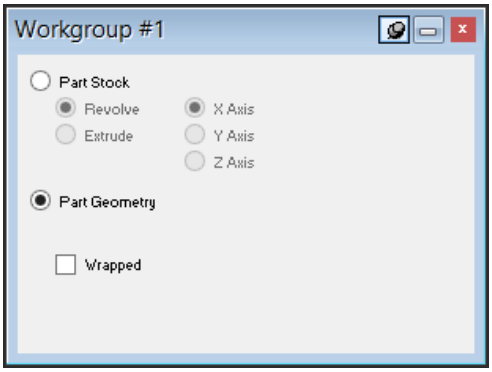
Workgroup dialog Right Mouse Menu

You can `right-click` the Workgroup title bar or a Workgroup list entry to open a context menu. This contains actions that are commonly used with workgroups:

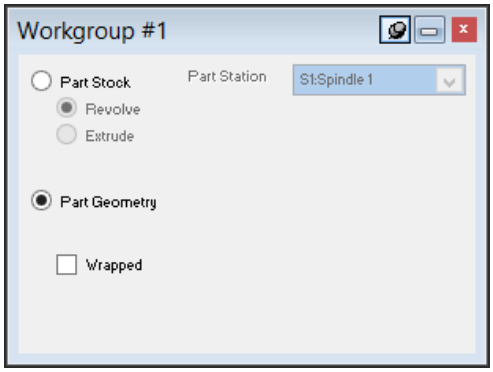


WG Info

The WG Info choice opens a dialog that allows you to specify the behavior of geometry in this workgroup:



Workgroup Info dialog for Mill part



Workgroup Info dialog for Turning part

Choose Part Stock to use the geometry as a stock shape, or choose Part Geometry to specify that geometry in the workgroup is used to define the part.

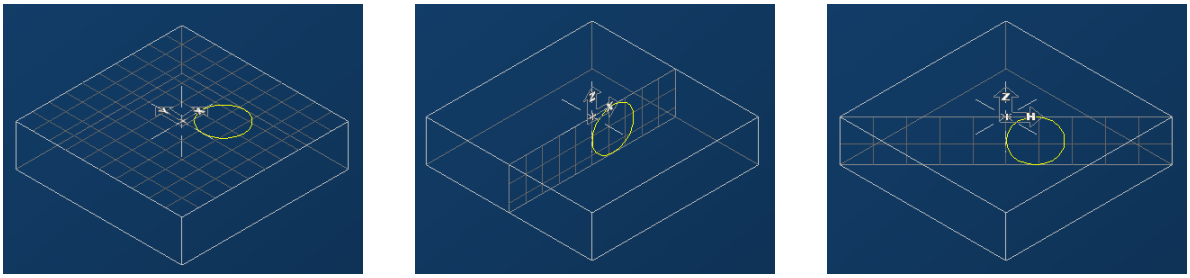
Part Stock

If Part Stock is selected, the geometry in the workgroup will be used for any calculations that need to look at the stock dimensions, such as Auto Clearance and Material Only. The geometry can be used as the starting stock condition in rendering and simulation, so long as it is a closed shape. When used in this way, the stock shape can also be used in machining operations to adjust the toolpath according to the amount of material to be removed based on the stock shape.

Extruded geometry in Turning workgroups is extruded along the depth.

Coordinate Systems

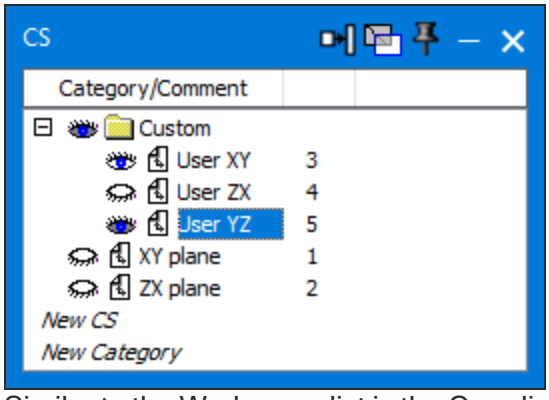
A coordinate system is a plane in space with an origin and three axes. Examples of different coordinate systems are the XY, XZ and YZ planes. In addition to these primary planes, there are a limitless amount of other planes that may be defined. Geometry can be defined using these other planes with the Advanced CS module. The following image is an example of three different coordinate systems. The circle in each coordinate system is at the same horizontal, vertical and depth position relative to each coordinate system.



Geometry is not contained in a coordinate system the way it is with workgroups. The coordinate system used to define geometry. It can be thought of as an attribute of the geometry and its orientation to the rest of the part. Geometry can be moved in and out of coordinate systems as needed.

Coordinate systems are used for 3D geometry creation, rotary part orientation for machining, multiple work fixture offsets, and as a basis for solid modeling. Fully understanding coordinate systems is vital to using the Advanced CS and SolidSurfacer modules. Coordinate systems are only available in the Level 2 interface.

Coordinate System List



Similar to the Workgroup list is the Coordinate System list. The Coordinate System list provides access to controlling the coordinate systems. When a new part is made there is only one coordinate system in the list: the XY plane for mill parts and the XZ plane for turning parts. Additional coordinate systems are created by clicking on New CS. The current coordinate system is highlighted. The “eyes” on the left side of the list show and hide the coordinate systems. The name of the CS may be changed by clicking on the entry in the Comment column.

Add a CS category by clicking New Category. Drag and drop a CS into the category as needed. Hide or show the contents of the category by clicking the - or + icon by the category folder.

The CS list also contains entries for specifying the spindle to which the CS is assigned (MTM only) and specifying the Work Fixture Offset to which the CS maps (requires Advanced CS). Clicking on column title will sort the CS list.

Coordinate System Palette

When a coordinate system is created, it is automatically made to be identical to the current coordinate system. The coordinate system palette allows you to redefine the new coordinate system's orientation, change its origin, toggle its depth axis or quickly align it to one of the primary planes.

Creation of 3D Bodies

Three dimensional bodies, either solid bodies or sheet bodies, may be used in addition to or in place of geometry for a part model. The process of solid modeling involves using graphically disjunct objects (solids and sheets) and combining, modifying and manipulating them to create the final part model. The process of solid modeling starts with the creation of a simple body referred to as an atomic or primitive body. Examples of an atomic body include a sphere, a cube, a revolved or extruded 2D shape, etc.
The basic operations, often referred to as Boolean operations, are to add, subtract or intersect these simple bodies to create a new, distinct body. Each time a Boolean operation is performed the result is always a single object. In this way, solid modeling allows the user to work with single objects rather than hundreds of surfaces.
Solid modeling capabilities are accessed through the Solid Modeling icon in the Command Toolbar. Clicking this button brings up a palette that contains the basic Boolean functions and buttons to open the two solids sub-palettes. The two sub-palettes are the Create Solid palette and the Advanced Solid Modeling palette.

1. Create Solid
2. Advanced Models
3. Slice
4. Swap
5. Replace
6. Add
7. Subtract
8. Intersection
9. Separate

The solids sub-palettes provide access to the creation of primitive solid bodies and the modification of primitive bodies. A primitive body is an original solid, created from a geometry profile or text input in a dialog describing the solid. Primitive bodies are the basis for creating solid models. The functions to create a primitive solid are found in the Create Solid palette.

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

The Advanced Solid Modeling palette provides access to non-basic Boolean functions that may be applied to bodies. This includes rounding corners and offsetting or shelling a body.

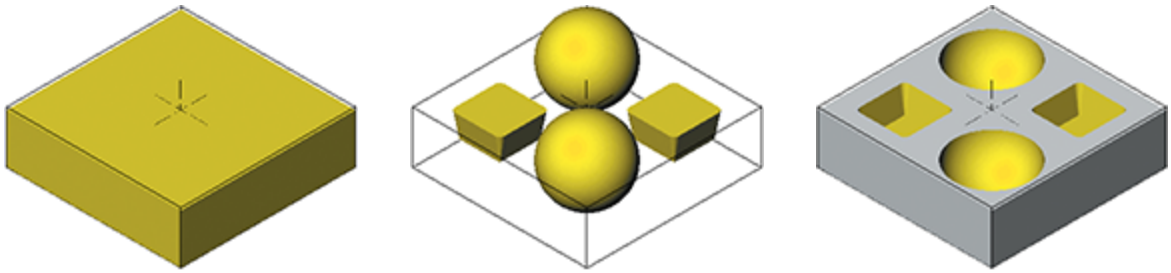
1. Offset/Shell
2. Round Corner
3. Unstitch Body/Heal
4. Draft

Surfaces can also be created by the 3D modeler. The modeler creates an object called a sheet. A sheet is similar to a solid body in that it is made up of faces. A face is a single surface. However, a sheet does not have any volume the way a solid does. A sheet is only made up of faces, whereas a solid is composed of the faces as well as the space enclosed by the faces. One way to think of it is that a sheet is similar to an infinitely thin gauge balloon while a solid body is more like a bowling ball.

1. Plane
2. Extrude Sheet
3. Revolve
4. Loft
5. Coons Patch
6. Sweep
7. Sheet From Face
8. Trim / Un-Trim
9. Stitch
10. Unstitch
11. Untrim & Extend Surface

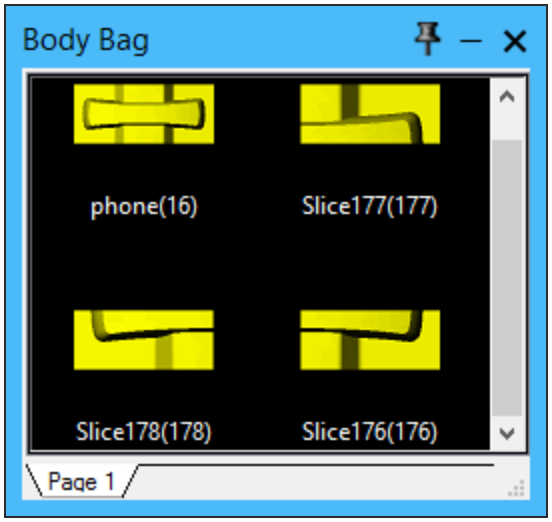
Creating Solids Example

The following three images show a very simple example of making a solid beyond primitive bodies. In the first image we have a simple cube made from the Cuboid button in the Create Solid palette. The second image shows four primitive bodies, two spheres and two tapered extrusions. The spheres were created from the Sphere button (also found in the Create Solid palette). The tapered extrusions were created from the Extrude button in the Create Solid palette along with selected geometry. The selected geometry provided the filleted corner shape to apply the extrusion to. The third image shows the result of subtracting the spheres and extrusions from the cuboid. The Subtraction button found in the Solid Modeling palette was used for this operation.



Example of making a solid model beyond primitive bodies.

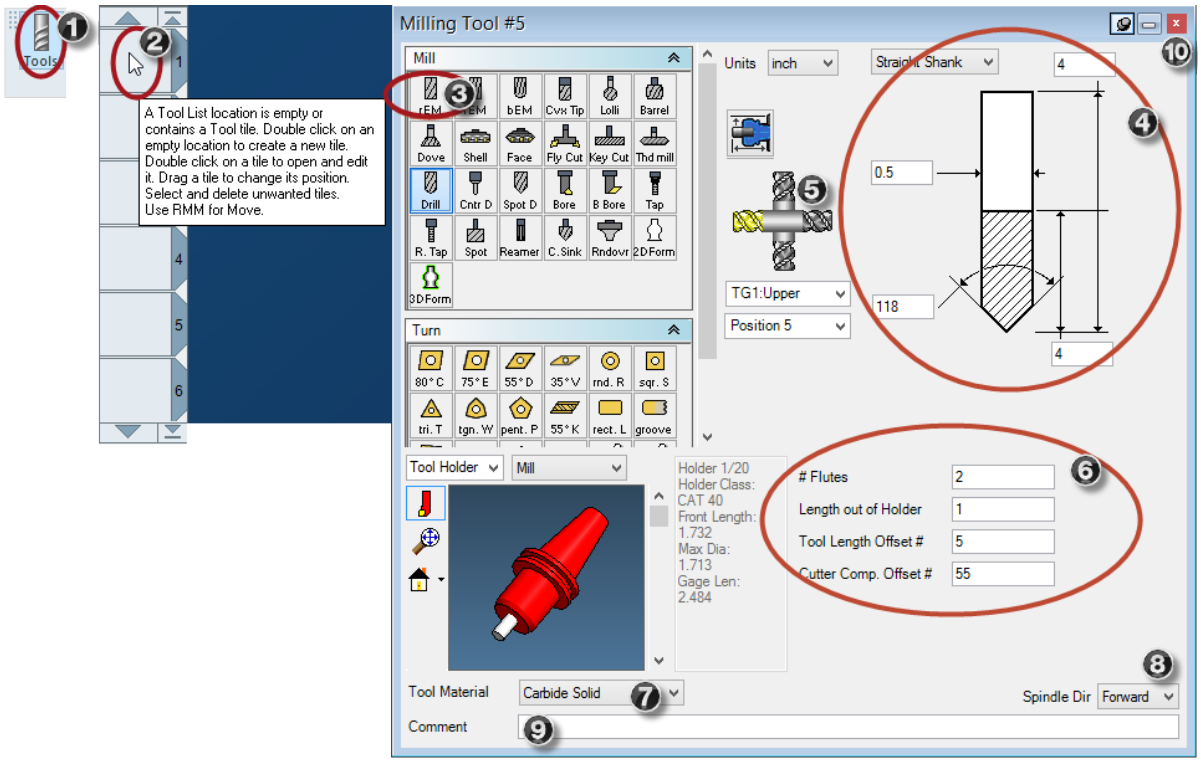
Workgroups and Coordinate Systems in Solids



Solids and sheets are not contained in workgroups. They are either drawn in the Workspace or placed in the Body Bag. The Body Bag is a small window where bodies are stored to minimize clutter. Bodies are assigned a coordinate system based on the current CS when the body was created. Some of the modeling functions such as extrusions and revolved bodies are CS-specific, meaning that the current coordinate system is used to create the body. Other modeling functions, such as lofting, are not dependent on the current CS.

Tool Creation

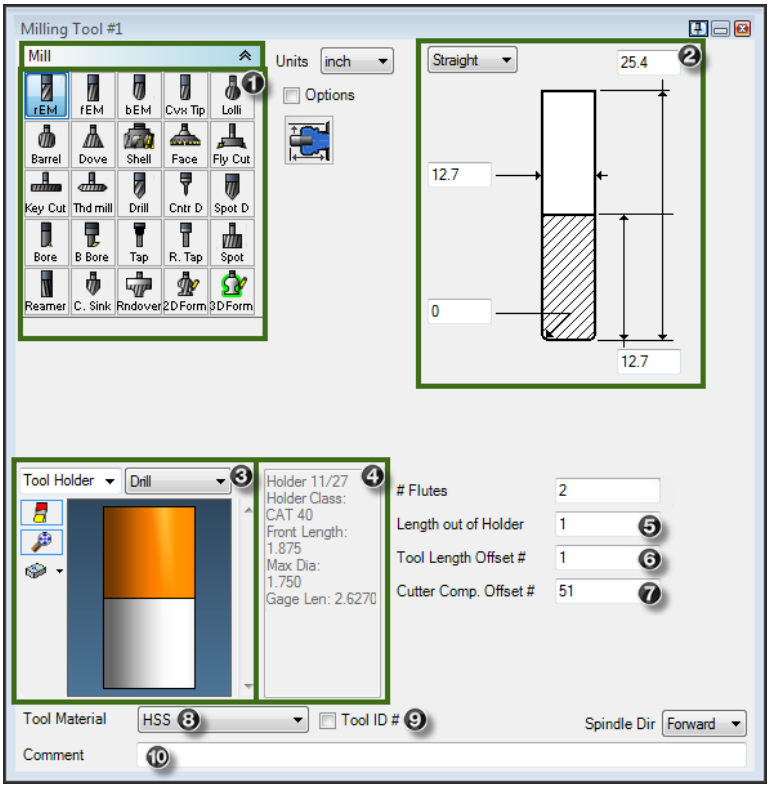
Quick How-To (Tool Creation)
Defining a tool is easy. Simply follow these steps.



1. Clicking the Tool List button in the Command palette will open the Tool List.
2. Double-click a tile in the Tool List.
3. Select the tool type from the matrix of buttons.
4. Define the tool's size through text entry or pull-down menus.
5. Specify tool orientation.
6. Specify Tool Stick out of Holder and Cutter Compensation.
7. Select the tool's material from the dropdown menu.
8. Select Spindle direction.
9. Add a comment if required.
10. Close the dialog or double-click on another tile (empty or full) to save the tool.

Tool Creation Dialogs

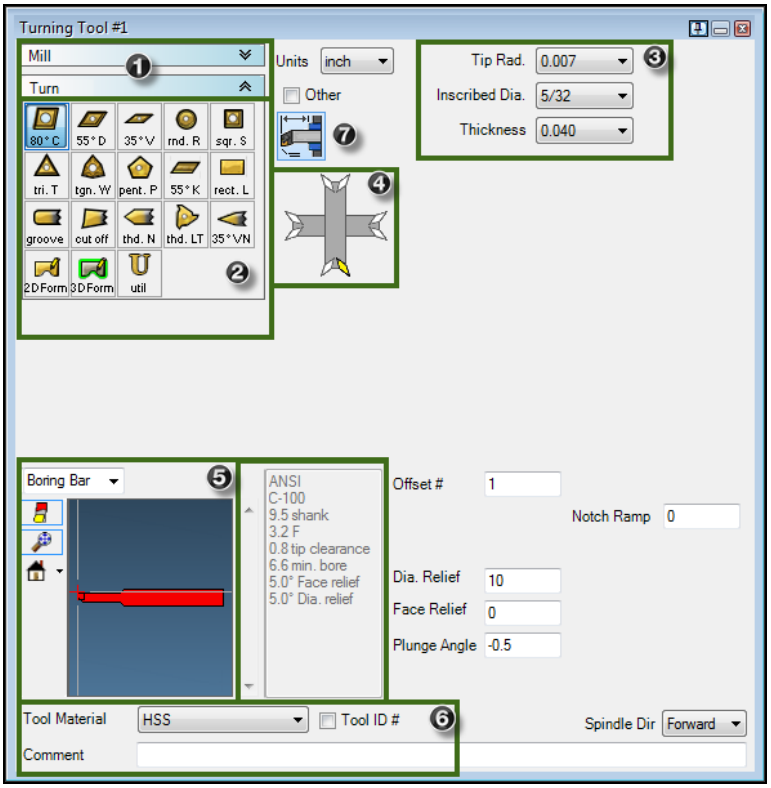
By `double-clicking` in an empty tile a new tool is created and a Tool dialog opens. A tool is then fully defined within the tool window. This includes, but is not limited to, the type of tool and its size and material. Only one Tool window may be open at a time. The Tool dialog can be re-sized by clicking and dragging on the margin.
All Tool Creation dialogs have common elements, including a matrix of tool types, a graphic of the tool, a method to specify the tool's dimensions and material, spindle direction and offset settings. Additionally, all of the Tool dialogs have a Comment box. This comment will be output in the finished code at the beginning of every operation that uses the tool. An example of each of the various Tool Creation dialogs follows.



1. Tool Type
2. Tool Diagram
3. Holder Diagram
4. Holder Specifications
5. Length out of holder
6. Tool length offset #
7. CRC Offset #
8. Tool Material
9. Tool ID #
10. Tool Comment

Mill Tool Creation dialog

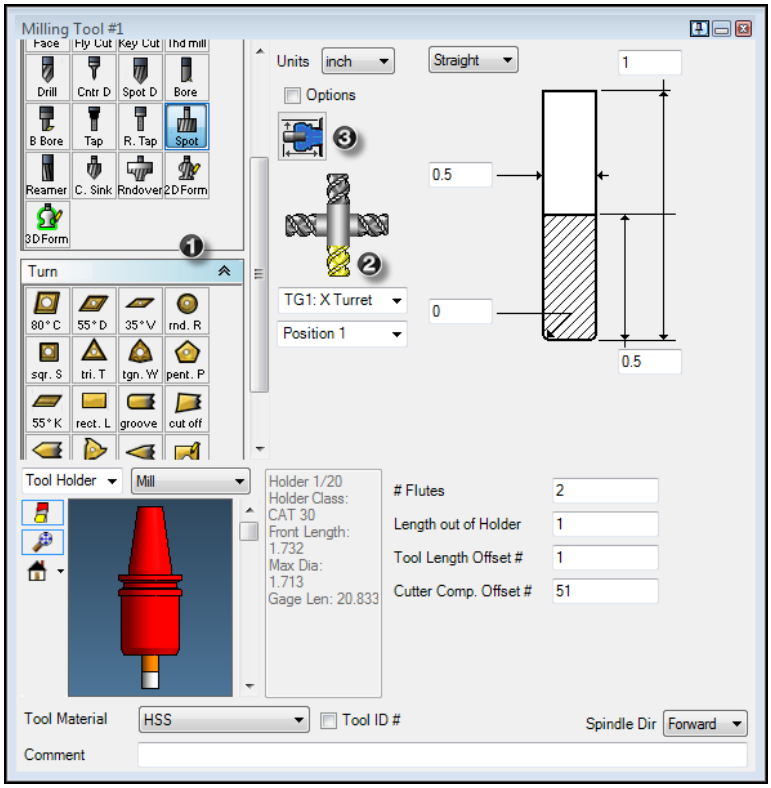
The Turning Tool Creation dialog has numerous options to describe an insert. In addition to the elements common to the Tool Creation dialogs, the Turning Tool dialog has a tab to switch back and forth between mill and turning tools, and a diagram to specify the insert's orientation. The Other checkbox allows the user to set up a custom insert. For more information, see the "Tool Creation" section of the *Turning* guide.



1. Tool Type
2. Insert Type
3. Insert Specifications
4. Insert Orientation
5. Tool Holder Definition
6. Tool Options
7. Tool Setup Data

Turning Tool Creation dialog

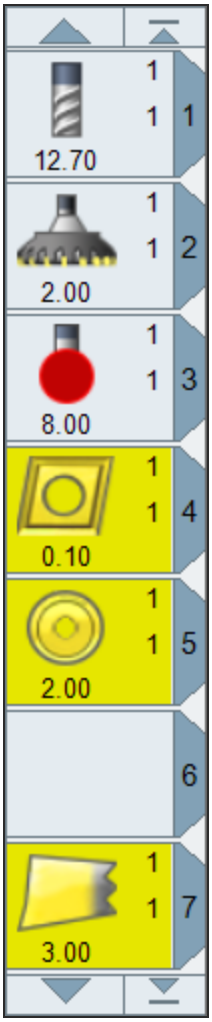
The Mill/Turn and Multi-Task Machining options slightly modify the Mill Tool dialog, adding several elements from the Turning Tool dialog and a Tool Orientation Diagram. The Tool Orientation Diagram allows the tool's axis of approach as well as the face to be cut to be specified.

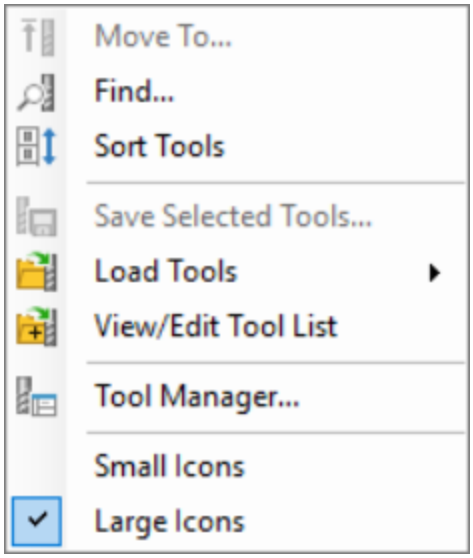


1. Tool Type
2. Tool Orientation
3. Tool Setup data

Mill/Turn Tool Creation dialog

Tool Tile List





Tool tile list and its right-cllick context menu

The tool tile list can contain a mixture of tool types, such as milling, turning, probing, and broaching. Each tool type has a unique graphic to help you quickly determine what type of tool it is. Double-clicking a tool tile opens the **Tool** dialog, showing the current parameter settings for that tool tile. Double-clicking another tool tile while the Tool dialog is open saves any changes, closes the dialog, and opens the Tool dialog for the new tool tile.
The tool tile list is accessed from the Tool button in the Command palette. The list can contain a maximum of 999 tool tiles per part. The tool tiles do not have to be contiguous; there can be empty spaces between tiles.

To Scroll Through a Tile List

- **Up**: Above the tile list, click to scroll up one tile or to scroll to the top.
- **Down**: Below the tile list, click to scroll down one tile or to scroll to the bottom.
- **Find**: Right-click anywhere in the list, choose **Find**, and enter a tile number.

- If you have a mousewheel, you can use it to scroll up or down individually, or Ctrl+mousewheel to scroll rapidly.

Note that scrolling has no effect on tiles' selection or deselection states.
Tool tiles can be reorganized in the list at any time, even after operations have been created, without reprocessing the operations. To reorganize the order of tools, drag the tool tile to an insertion point. The system will automatically adjust the operations to reflect the change in tool order and number.
Tool parameters can be modified at any point. However, if operations have been created using the tool, those operations do not incorporate the new parameters until you use Redo. For complete details, see the *Common Reference* guide.

Dragging Multiple Tools

You can select multiple tools and drag the group to the Process List. The tools appear in the Process List at the location you release the mouse button, whether the tiles are empty or already contain machining functions. Any existing tools will be replaced. You can also insert the tools between process tiles.
You can drag a tool or multiple tools to multiple process tiles.
To Select a Range of Tiles:

1. Select the first tile.
2. Hold down the Shift key and select the last tile in the range.

To Select Multiple Tiles:

1. Select the first tile.
2. Hold down the Ctrl key and select each tile you want to drag.

To Drag Multiple Tools to the Process List:

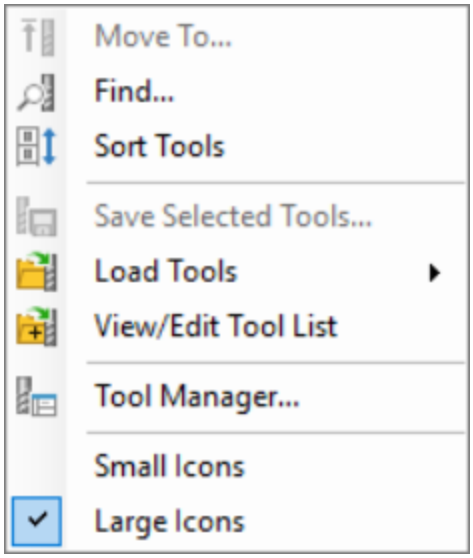
1. Select the tool tiles you want to drag. The tools are highlighted.
2. Drag the tools to the position in the Process List where you want the tools to appear and release the mouse button. The top tile in the group will be inserted where the hand icon is positioned and the mouse button released.

To Drag a Tool to Multiple Tiles in the Process List:

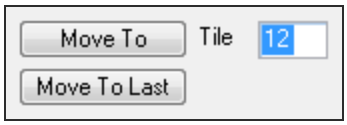
1. Select the process tiles you want to add the tool or tools. The process tiles are highlighted.
2. Select the tool or tools you want to use.
3. Hold down the ALT key, drag the tool or tools to the selected process tiles, and release the mouse button.

Tool List Context Menu

Right-click anywhere in the tool tile list to summon this menu.



Move To



The Move To menu item opens a dialog that lets you move a tile to a specific position by number or move a tile to the last position in the list.

Find

The Find menu item opens a dialog that lets you locate a specific tile by number or jump to the last tile. For operations, you can also search by tool number.

Sort tools

When selected, tools will be sorted by Tool Group and spaces will be inserted between tool groups.

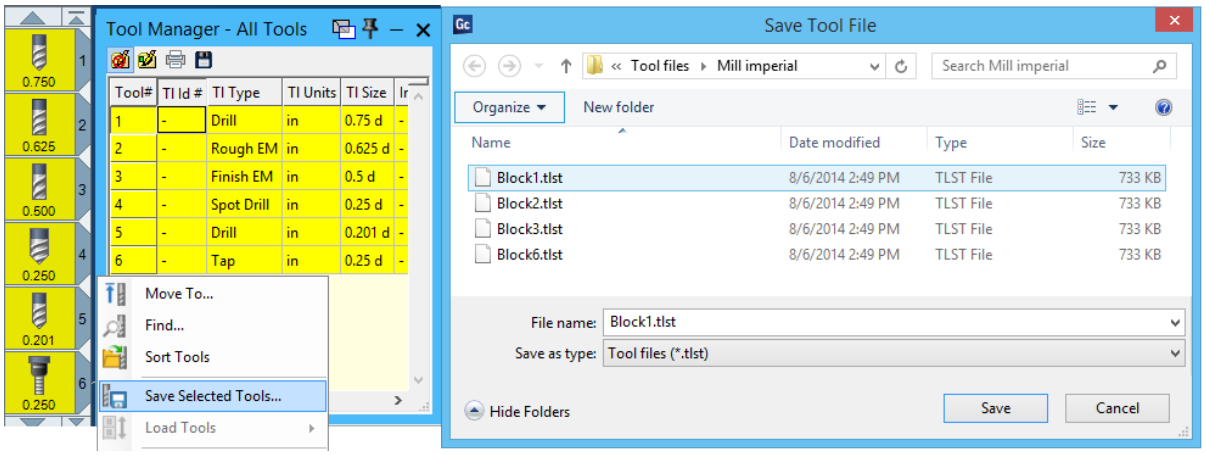
Save Selected Tools

This menu item opens a dialog that lets you save the selected (highlighted) tools as a `*.tlst` file.

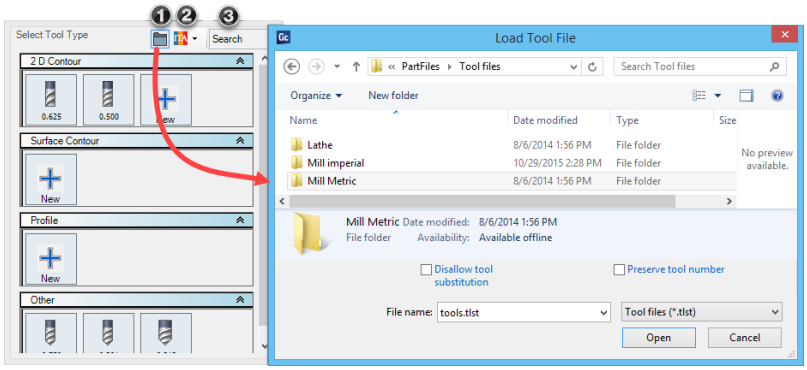
To Scroll Through a Tile List

- **Up**: Above the tile list, click to scroll up one tile or to scroll to the top.
- **Down**: Below the tile list, click to scroll down one tile or to scroll to the bottom.
- **Find**: Right-click anywhere in the list, choose **Find**, and enter a tile number.
- If you have a mousewheel, you can use it to scroll up or down individually, or Ctrl+mousewheel to scroll rapidly.

Note that scrolling has no effect on tiles' selection or deselection states.
Navigate to your desired directory to make it the default directory for saving and loading tool lists.
Enter a filename and click the Save button.
Note that you can also create tool lists (`*.tlst` files) from within the **Tool Manager** dialog.



Load Tools

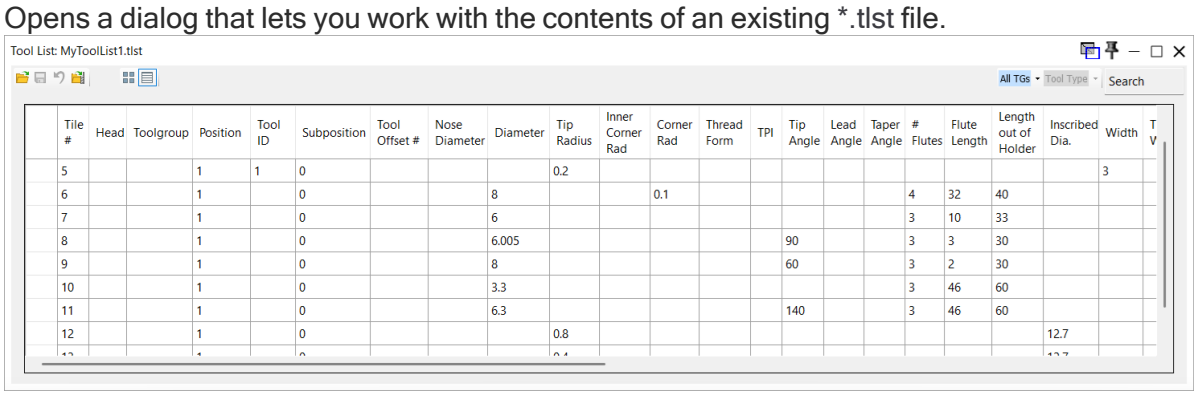


1. Load Tool file
2. Load tools from ISCAR Tool Advisor or other tool library
3. Search for Tool file

1. Tools can be loaded from a previously saved `*.tlst` file. This is a list previously saved using the Save Selected Tools menu item.
   Saved tool lists can be added to any part file using the Load tools command. Or, alternatively, tools can be loaded into the process creation tool selection dialog as shown above.
2. Tools can be loaded from an ISCAR Tool Advisor or similar `*.xml` file.
3. Use the search facility to search for specific tool sizes within the existing tool tiles. Simply start typing a tool size and select the exact size from the dropdown list.

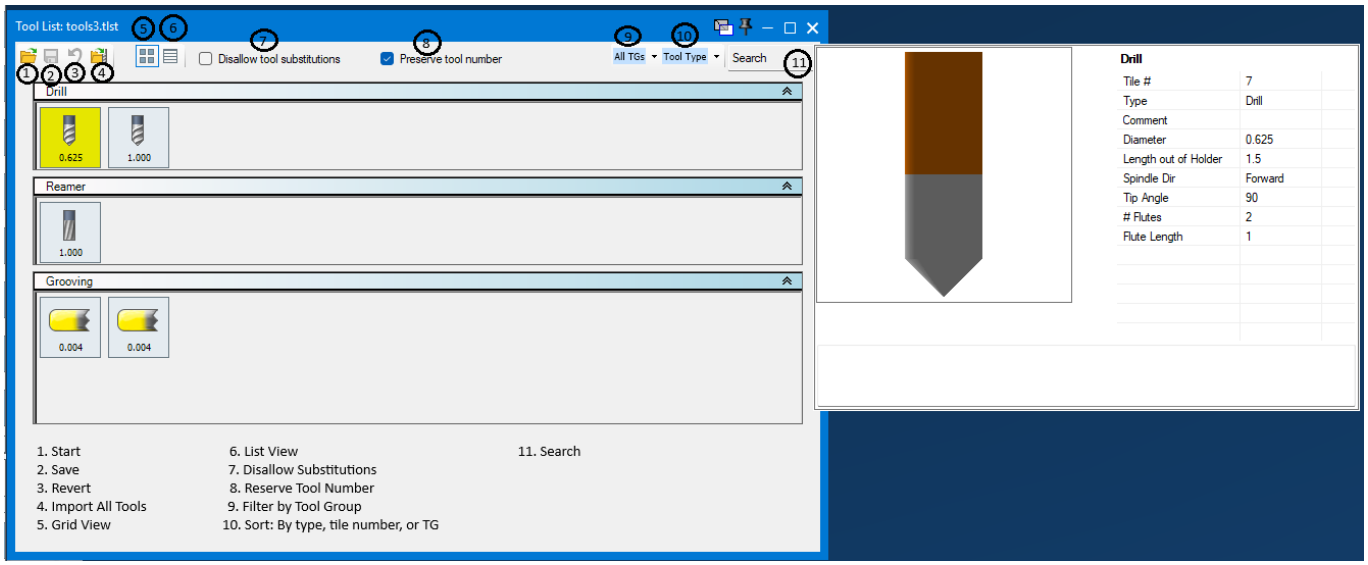
View/Edit Tool List

Opens a dialog that lets you work with the contents of an existing *.tlst file.



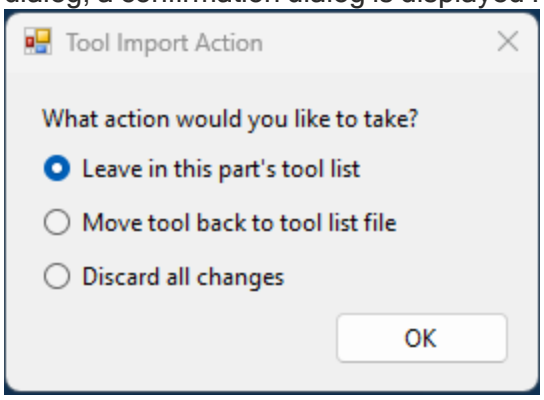
Tile #	Head	Toolgroup	Position	Tool ID	Subposition	Tool Offset #	Nose Diameter	Diameter	Tip Radius	Inner Corner Rad	Corner Rad	Thread Form	TPI	Tip Angle	Lead Angle	Taper Angle	# Flutes	Flute Length	Length out of Holder	Inscribed Dia.	Width
5			1	1	0				0.2												3
6			1		0			8			0.1						4	32	40		
7			1		0			6									3	10	33		
8			1		0			6.005						90			3	3	30		
9			1		0			8						60			3	2	30		
10			1		0			3.3									3	46	60		
11			1		0			6.3						140			3	46	60		
12			1		0				0.8											12.7	

This dialog can be toggled between spreadsheet form, shown above, and list form, shown below. It lets you:



- Hover over a tile in the dialog to show the Tool Preview in the hover window.
- Order tools by tool type (default), by tile number, or by toolgroup.
- Use the search bar to find a tool by text and/or by its most important dimension, optionally filtered by toolgroup.
- Delete a tool from the tool list.
- Drag a tool from the tool list to the part or vice versa.

Double-clicking any required tool in the list opens the Tool dialog, allowing tool parameters to be reviewed and updated as required. After completing the modifications and closing the Tool dialog, a confirmation dialog is displayed :



Tool Manager

For complete details, see the *Common Reference* guide.

Small/Large Icons

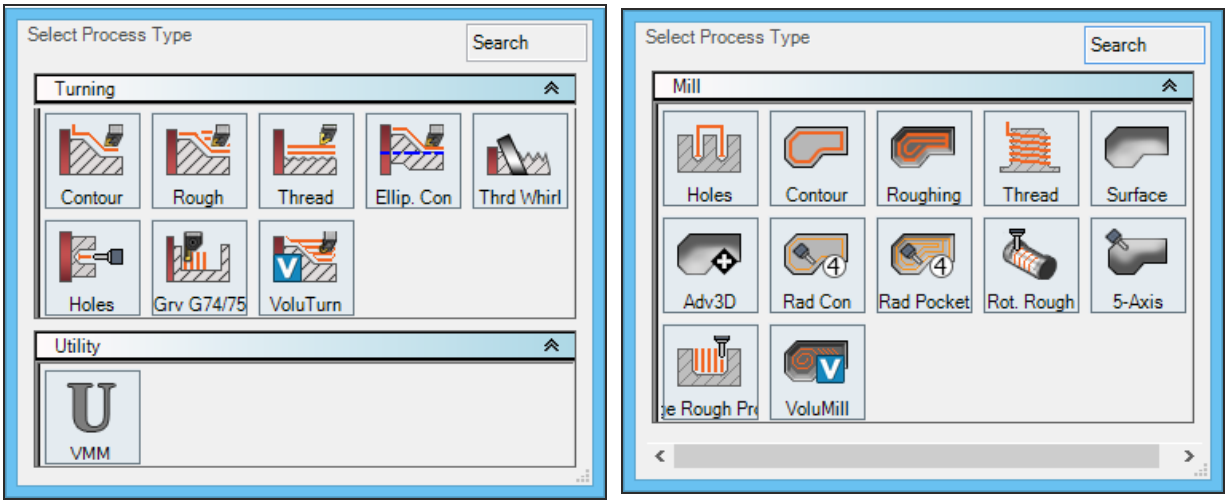
Choose the display size of all toolbar icons.

Tool Manager

The Tool Manager provides an expanded view of the Tool List in a tabular format. Each column contains a parameter type and each row contains a tool. The list is customizable and can be sorted, saved and printed.
For more details on the manager capabilities see the *Common Reference* guide.

Process Creation

To create a Process, you need to select a Process type and a Tool.
The machining Palette displayed will vary depending on the MDD you have specified in the DCD dialog.
A part that references a machine capable of Turning offers a Machining palette with Turning Contouring, Roughing, Threading, and Drilling tiles.
A part that references a machine capable of Milling offers a Machining palette with Mill Drilling, Contouring, Pocketing, and Threading tiles. If SolidSurfacer is installed on your system, the Mill machining palette also offers a Surfacing tile and local stock definition buttons. If you have other options, other controls also appear.

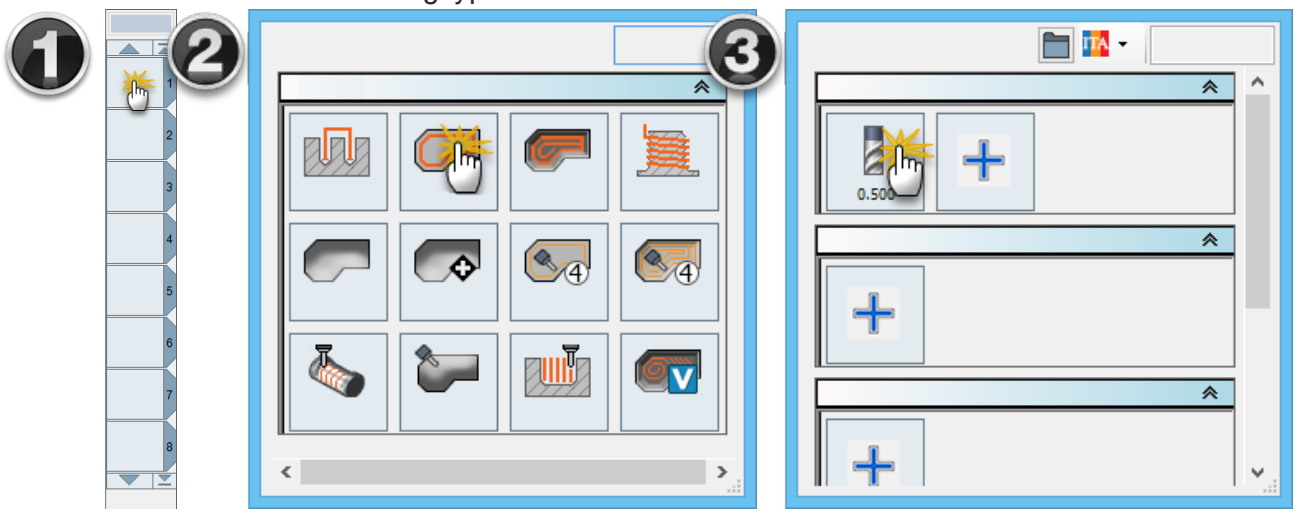


Turning Machining Palette and Mill Machining Palette

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.
When the machine is capable of both milling and turning operations, its Machining palette has two dropdown sections. These dropdowns display the Turning and Mill Machining options available for the MDD in use, providing access to both types of machining in one palette.

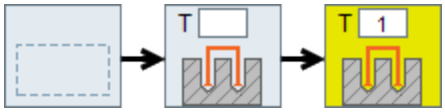
Creating a Process

A process is defined by a tool and a machining function. To create a process, double-click a Process Tile. Select a process Type from the options available for your machine. Then, select a Tool to use. Hover over a tool icon to display more tool information to confirm you have the correct tool.
If you already know which Tool you wish to use, you can drag its tool tile onto an empty Process Tile and then choose the machining type.

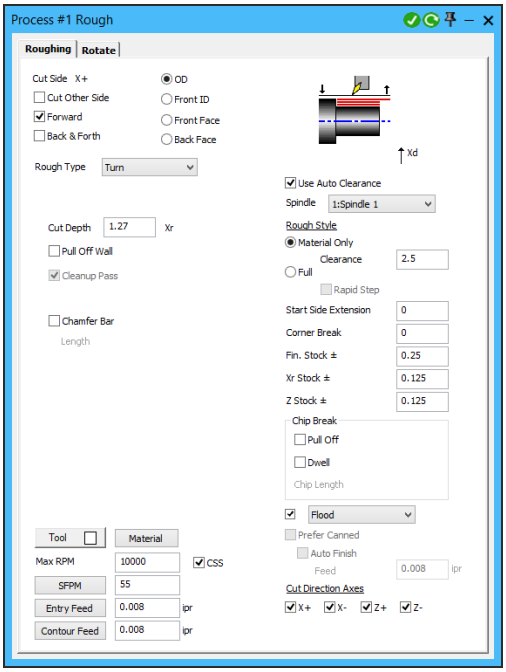


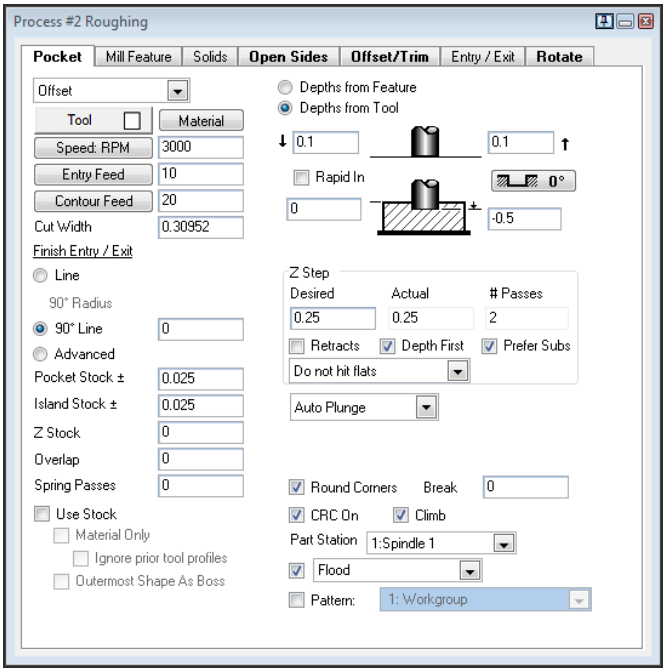
1. Double-click an empty Process tile.
2. Choose a Process type.
3. Choose a tool type.

The Process tile will now show the Machining type icon and the number of the tool selected.



Once these items are in place, the appropriate Process dialog will open. Clearance values, cut depth, speeds and other items are filled out in the Process dialog. The actual Process dialog will vary depending on the type of machining function chosen. Roughing, Contouring, Drilling, Threading and Surfacing Process dialogs are different from each other, with many common elements such as the Entry/Exit Diagram. For detailed information see the *Mill* and *Turning* guides.





Lathe and Mill Process dialogs

Applying the Same Machining Function to Multiple Processes

You can quickly apply the same machining function to multiple processes: The machining function is applied to all selected process tiles, and a dialog opens for the first-selected process tile.
To apply the same machining function to multiple processes:

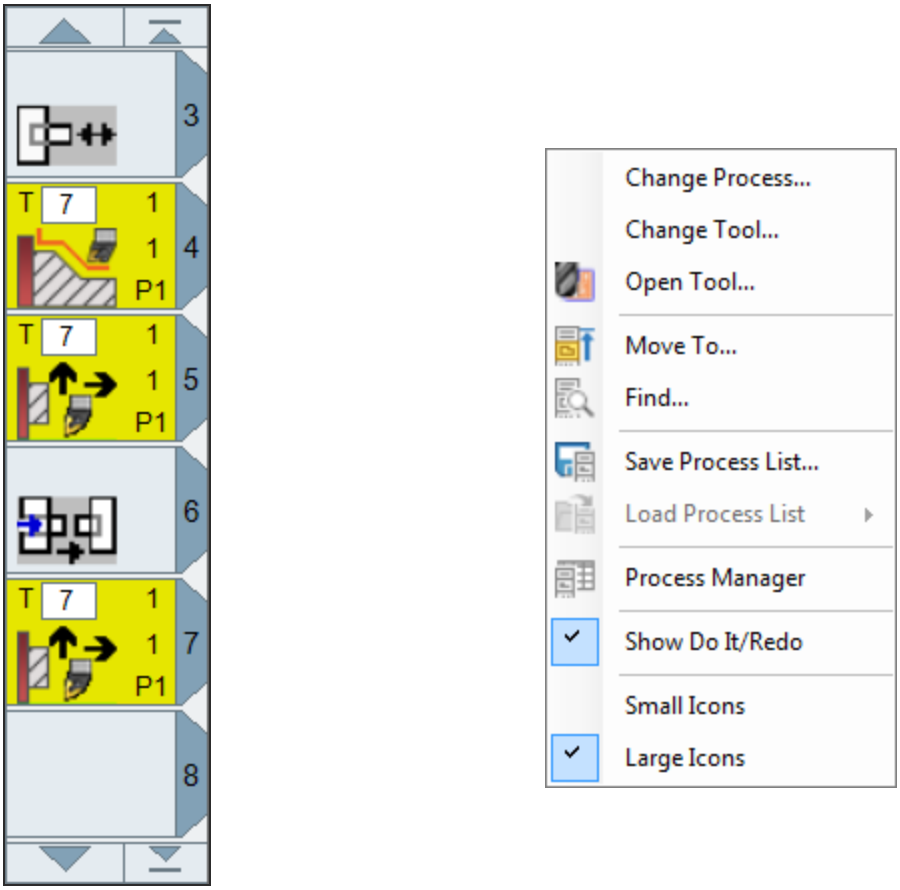
1. Select the process tiles you want to change.
2. ALT-double-click the machining tile you want to apply or ALT-drag the machining tile to the Process List.

Clearing the Process List

You should delete or modify Process tiles that you used to create previous operations and before you create a new set of operations. When you click the Do It or Redo button, all of the Process tiles in the Process list are used to create operations, even if operations have already been created using those tiles. You should delete the Process tiles after you create the operations to ensure that you do not unintentionally create duplicate operations. Clicking the Clear button

deletes all the Processes in the Process list.

Process Tile List



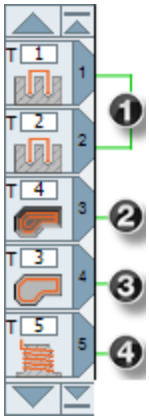
Process tile list and its right-cllick context menu

The process tile list contains a maximum of 99 process tiles. Process tiles do not have to be contiguous; there can be empty spaces between tiles. Multiple processes can be created in the same list. Any combination of machining functions can be grouped as needed.
Once operations are created, the processes can be thrown away, because the information from the process is stored in the operation.

Multiple Process Programming

If the Process list contains more than one Process tile when you click the

button, multiple operations are created. All these operations use the same geometry. When the finished operations are placed in the Operations List, they are “linked.” When you double-click one operation, all operations created with the same Process list are selected. This enables you to change the geometry more easily. Linked operations can be moved around in the Process list without losing the link. A typical use for multiple Processes is roughing and finishing the same shape with the same or different tools.



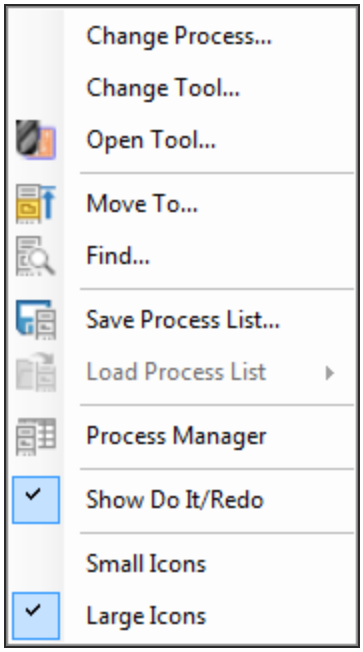
1. Drilling Processes
2. Roughing Process
3. Contouring Process
4. Threading Process

If multiple Process tiles of the same type are used in the same Process list, each tile is applied to the selected geometry. For example, you can use three drilling processes to drill, tap, and counter-bore the same points or circles. Three separate operations are created to perform each of these functions.

When multiple Process tiles of different types are used in the same Process list, the effect of each process changes, depending on the other processes in the Process list. If a drilling process is combined with a roughing or contouring process, the drilling process drills entry holes for the other process. If a contouring process is combined with a roughing process, the contouring process performs a finish pass around the wall of all selected pockets and bosses.

When multiple operations are created from the same Process list, they are linked. Double-clicking on a linked Operation tile selects all of the other operations created by the same Process list. The Process list is rebuilt and the geometry is reselected. The link is preserved even if the operations are reorganized in the Operations list.

Process List Context Menus



Right-click the process list and select the item you want from the menu. In addition to Find and Process Manager you can select:

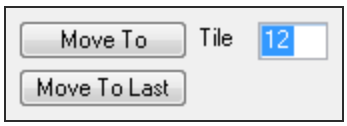
Change Process/Tool

Change the Process Type or Tool used for the current process.

Open Tool

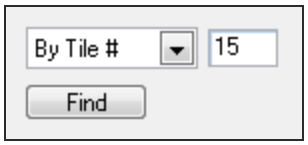
Opens the tool dialog of the current process.

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.

Save/ Load Process File

Process lists can be saved to a .prc2 file for reuse.

Process Manager:

See Tool Manager .

Small/Large Icons

Choose the display size of all toolbar icons.

Process Manager

The Process Manager provides an expanded view of the Process List in a tabular format. Each column contains a parameter type and each row contains a process. The list is customizable and can be sorted, saved and printed.

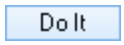
For more details on the manager capabilities see the *Common Reference* Guide.

Operations

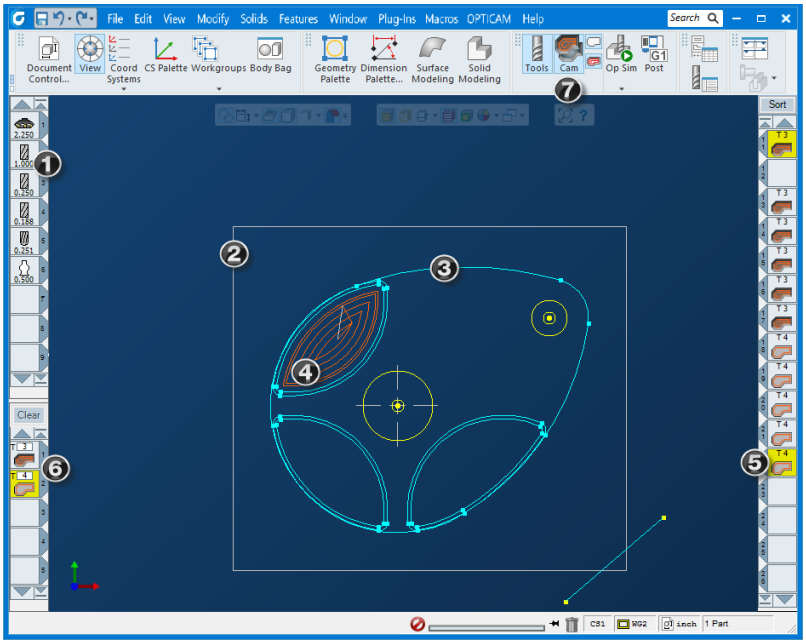
A single operation consists of toolpath, clearances, tool information, feeds & speeds, and coolant choices. It is a visualization of G-code and will be used as the source of the G-code sent to a CNC.

Operations are made from a GibbsCAM process. A process is the combination of a tool and a machining function (roughing, contouring, drilling) applied to geometry or solids. Processes are used to specify all of the operation's settings.

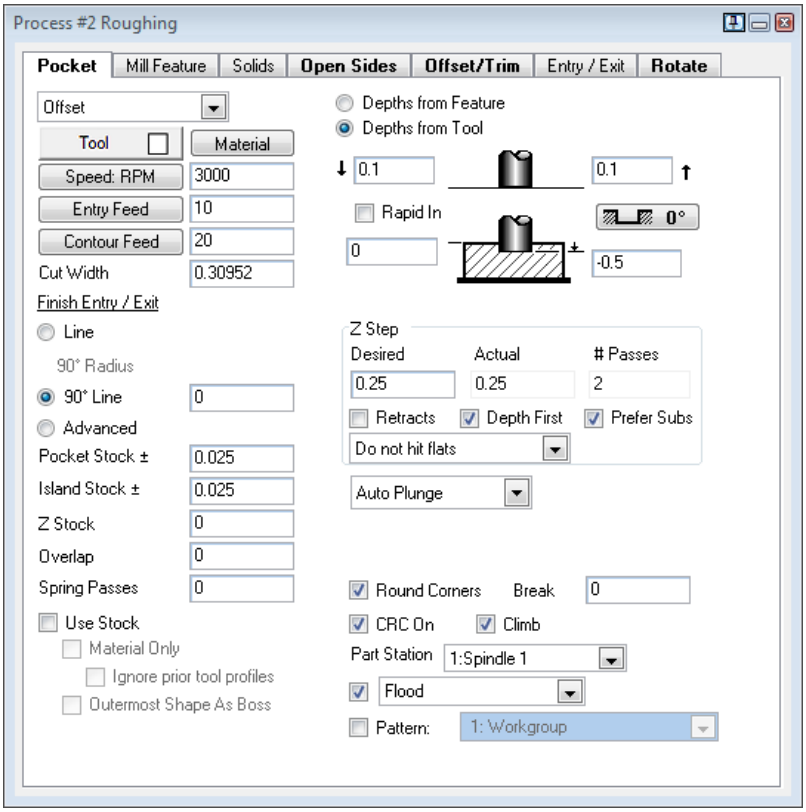
1. The first step to creating operations is to have the part model defined along with a list of tools to cut the part.
2. You then click on the Cam button in the Command Toolbar to open the Process and Operation Lists.

3. Double-click a process tile to create a Process. This will open a window where you can choose a process type and then a suitable tool. Finally, a dialog will open to set the operation's parameters.
4. You then select the geometry or solid that is to be machined and click the Do It 

button above the process tile list. This will create the toolpath for an operation. Repeat this process until the part is complete.
5. Finally run Cut Part Rendering to check the part.



1. Tools
2. Stock
3. Geometry
4. Toolpath
5. Operations
6. Processes
7. CAM Palette



Parameters

The components of operations

Defining the Cut Shape

After you complete the Process dialog, you must define a cut shape. The cut shape is the geometry to which the process is applied.

For contouring and roughing functions, you must select geometry to use as the cut shape for the operation. For roughing operations, you must select at least one closed shape for pocketing or stock shape for face milling. For engraving operations, you can select multiple shapes for the cut shape.

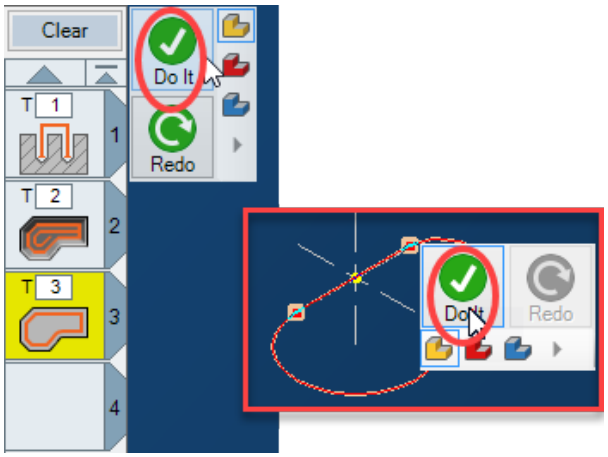
For contouring operations where you are machining a single shape, the cut shape is defined with Machining Markers.

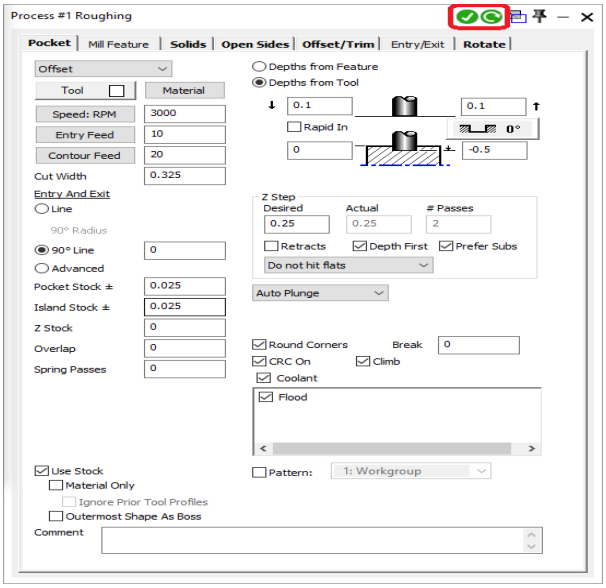
Machining Markers indicate the portions of the geometry to use as the cut shape when creating a toolpath. The markers appear when you select the geometry. Drag the machining markers to the locations you want on the geometry. Machining Markers are not used for drilling and threading functions.

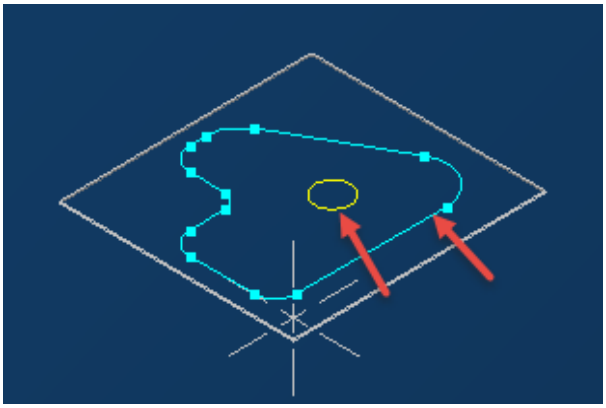
For more detailed information see the "Machining Markers" section in your Mill or Turning Guide.

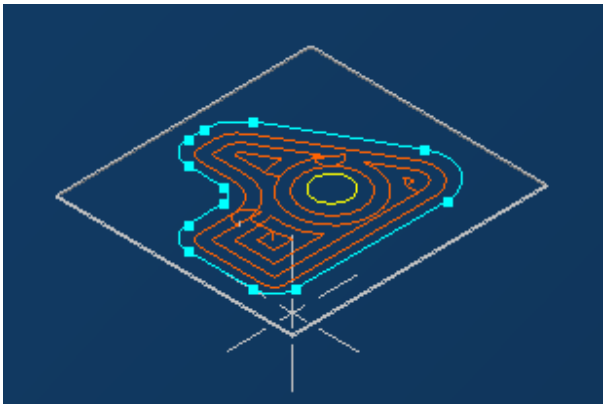
Process becomes Operation

An operation is automatically created by entering the desired parameters in the Process dialog, selecting the geometry or solid to be machined and then clicking the Do It button.
(The Do It button is available in the workspace in a translucent dialog, from the Title bar of the Process dialog and also by right-clicking anywhere in the workspace.)
This will generate one or more operations and toolpath.







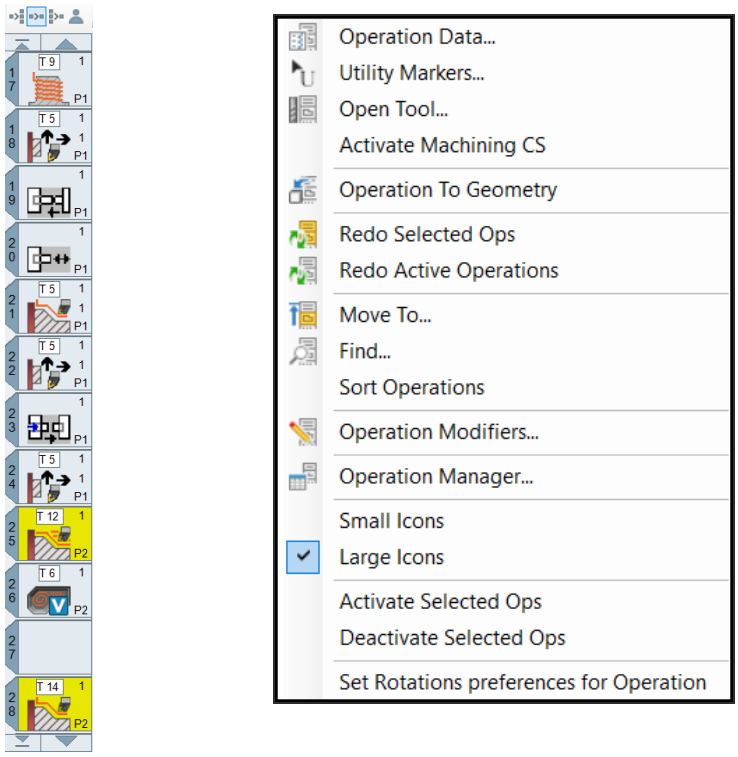


A process, selected geometry, and the results of clicking the Do It button.

Clicking the Do It button applies the processes defined in the list to the selected geometry, creating as many operations as needed. Operation tiles are created and appear in the Operations List. A pocket might contain an entry hole process followed by roughing, finishing and possibly back boring in one process group.
If any completed Operation tiles are highlighted in yellow, the Redo button is also available. If you make changes to the information in the Process list, click the Redo button to replace the operations in the list with the new operations.

Operations contain the finished toolpaths. A toolpath consists of the actual moves the tool will make. The toolpath is based on the cut shape. The post processor uses the operation data to generate G-code.
Be sure to deselect the Operations so that you do not make unintentional changes. To deselect Operation tiles, select an empty tile location or an insertion point between locations.

Operation Tile List



Operation tile list and its right-cllick context menu

The software will use the list of operations to generate the final program that is sent to the CNC machine. Operation tiles cannot be moved away from the operation tile list, but they can be sorted and reordered. To edit an operation's process information, double-click the operation to load its processes to the process tile list. Clicking the Redo button will update the changes.
A part can contain a maximum of 16,000 operations. Operation tiles do not have to be contiguous; there can be empty spaces between tiles.

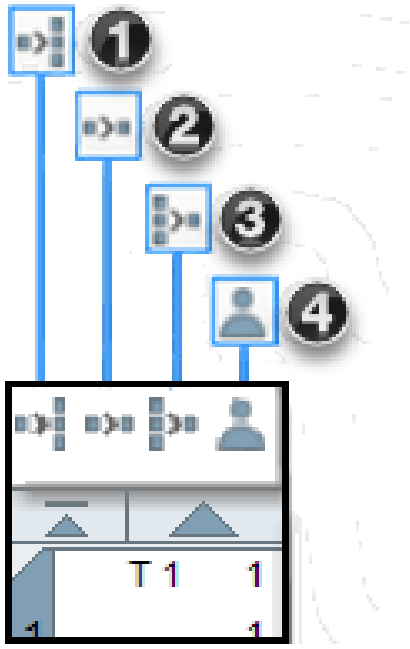
Context Menu (Right-Mouse Menu)

The operation tile list has a context menu, accessed by `right-clicking` an operation tile. The context menu offers commands that can be applied to an operation or its toolpath, as well as the usual Find and Move options.
You can also make changes to the actual toolpath. This includes changing speeds at specific points in the toolpath, accessing the tool used for the operation, converting the toolpath to geometry, and locking data in the operation. For complete details, see the *Common Reference* guide.

You can also lock data and change specific information about an operation using the **Operation Data** dialog. See Operation Data.

Operation Tile Stacking

The operation tile list offers four choices for organizing operation tiles:

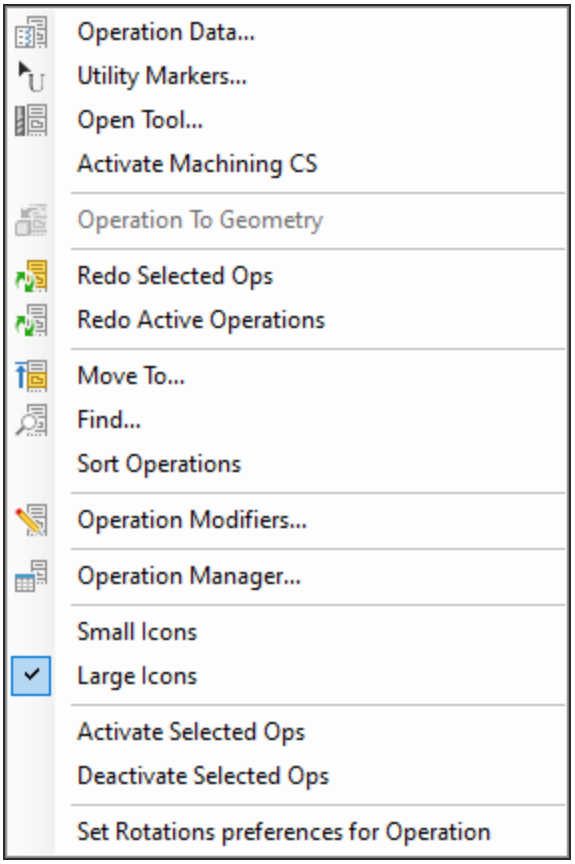


1. Individual: Unstacked.
2. Process Mode: Consecutive operations are stacked together if they originated from a single process and all belong to the same flow.
3. Process List: Consecutive operations are stacked together if they originated from a group of processes to machine the same geometry–and all 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.

For more detailed information on tile stacking, see the *Common Reference* guide.

Operation List Context Menu

The Operation List provides the following context menu for finding, managing, and modifying operations :



Operation tile context menu

`Right-click` anywhere in the operation list and select the functions you want from the menu. You can select the following:

Operation Data:

Displays the Operation Data dialog. See Operation Data.

Utility Markers:

Displays the Utility Markers dialog and the toolpath for the current operation. You can edit various position-dependent toolpath data. See "Utility Markers" in the *Mill* or *Turning* guides.

Open Tool:

Displays the tool dialog associated with that operation.

Operation To Geometry:

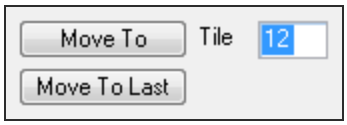
Converts the highlighted operation's toolpath to geometry. This geometry can then be modified as needed and a center cut contour operation can be applied to it. This applies to contouring, roughing and surfacing operations. This may be useful for avoiding clamps and fixtures or editing a pocket or profile's toolpath for individual preferences.

Redo Selected Operations:

Selecting this option regenerates the toolpath for any selected (highlighted) operation.

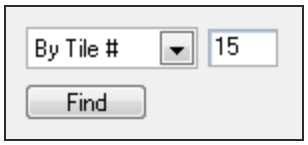
Redo All Operations:
Selecting this option regenerates the toolpath for all operations.

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 Operations:
Reorganizes all operations in the Operations List by tool number and creation order, from lowest to highest tool number. See "Sorting Operations" on page 60.

> 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.

Operation Manager:
Displays the Operation Manager .

Small/Large Icons
Choose the display size of all toolbar icons.

Operation Manager

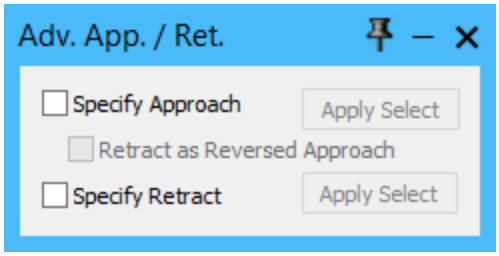
The Operation Manager provides an expanded view of the Operation List in a tabular format. Each column contains a parameter type and each row contains an operation. The list is customizable and can be sorted, saved, and printed.
For more details on the manager capabilities see the *Common Reference* guide.

Operation Data

You can edit various operation specifications in the Operation Data dialog. Right-click an operation in the operation tile list to do this. You can change and lock values in the Operation Data dialog when multiple operations are created from a single Process tile. You can change certain specifications for one operation without changing them for all the operations that were created with the same Process tile. In Milling, for example, if multiple pockets are machined from one Process tile, a separate operation is created for each pocket. If an Entry or Exit Clearance

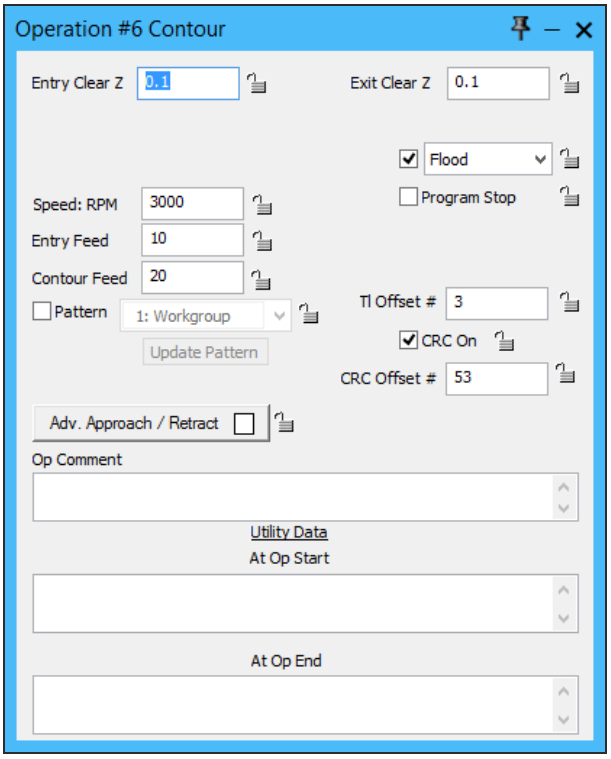
Plane needs to be changed to avoid a clamp between two of the pockets, you can change the clearance values in the Operation Data dialog for one of the operations. The clearance values of the other roughing operations are not affected.
You can specify a custom path for tool approach and retract. This allows you to determine tool locations to avoid interference between tool and stock for such operations as back boring, milling with right-angle heads, and ID or OD turning.

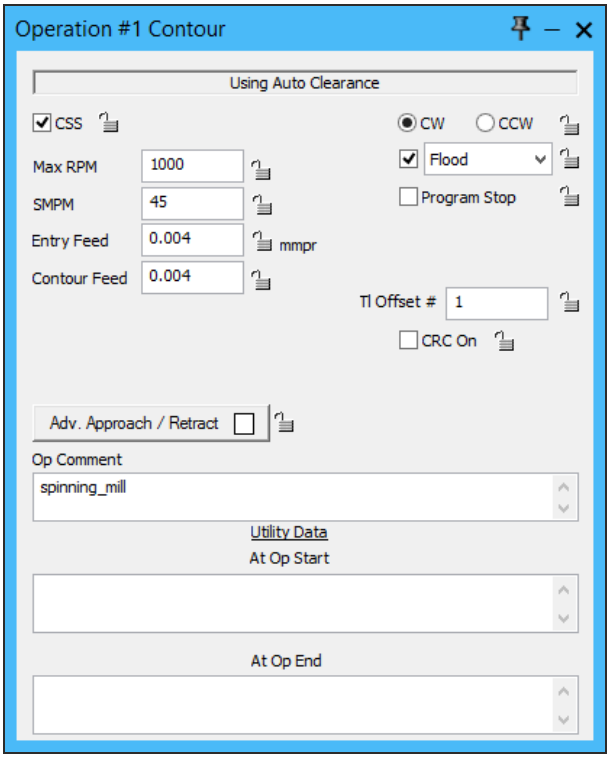
Please Note: This feature requires a post upgrade. To request a post upgrade, contact your Reseller or the Gibbs Post Department.



Checking the Adv. Approach/Retract checkbox reveals the Advanced Approach Retract dialog, allowing you to select a custom approach and/or a custom retract.

You can also specify that the retract path reverses the approach path. When you click either of the Select buttons, a dialog prompts you to define a path by selecting it in the workspace. Clicking Apply sets that path as the custom approach or retract, before the tool moves to the operations' Entry Clearance value.





Mill dialog | Turning dialog

The Operation Data dialog also allows for overrides of certain items such as speeds and entry/exit settings. More importantly is text that can be output in the G-code that is operation-specific and the ability to lock values by clicking on the padlock icons. If the part is opened on

another machine with different speed and feed data, the original settings will not be unintentionally overwritten. Unlocking these settings will allow changes to the operation. For additional information, see *Mill* and *Turning* guides.
Mori Seiki (and others) have the capability for performing turning operations that use a mill-like tool that spins. In order to mark an operation as being a mill op but with the part turning (for turning machines, millturns, and MTM), enter `spinning_mill` in the Op Comment section of the Operation Data dialog. This will also ensure that the part will be rendered correctly and is also a requirement for Post Processing.

Lock button:

Locked items ( ) retain the values entered in this dialog even if the operation is reprocessed. Unlocked items ( ) return to their original values if the operation is reprocessed. Changes that affect the toolpath appear in the toolpath drawing and the rendered image. The information in the

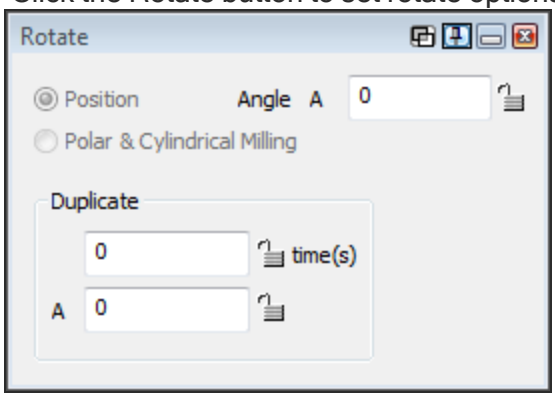
process tile that created the operation is modified to reflect the changes made in this dialog. If an operation contains one or more locked values, a small lock symbol appears on the Operation Tile.

To lock or unlock a value:

Click the graphical button next to the right of the control to toggle its state between “locked”( ) or “unlocked”( ).

Rotate button:

Click the Rotate button to set rotate options.



Program Stop:

The Program Stop checkbox is only available in the Operation Data dialog. If selected, a program stop command is added at the end of the operation in the posted output. The default position is off.

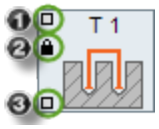
Op Comment:

Information entered in the Op Comment text box appears in the posted output before the selected operation it refers to. You can also enter Utility Data in this dialog.

At Op Start and At Op End:

You can use these text boxes to pass custom commands that trigger actions inside the post processor. The list of custom commands and their actions should be supplied to you with the documentation for your custom post.
Also, in these text boxes, any text string enclosed between simple straight quotes (either "..." or '...') will appear as a text string in the posted output. Other types of quotemarks, such as “ ” ‘ ’ « » 「 」 『 』, do NOT act as text delimiters. In the posted output, each such quoted text string is inserted on a new line.
If utility data is entered in the Utility Data At Op Start text box, then a small square appears on the Operation tile in the upper left hand corner. If utility data is entered in the Utility Data At Op End text

box, then a small square appears in the lower left hand corner of the Operation tile. If operations contain locked values, then a small lock symbol appears on the Operation tile.



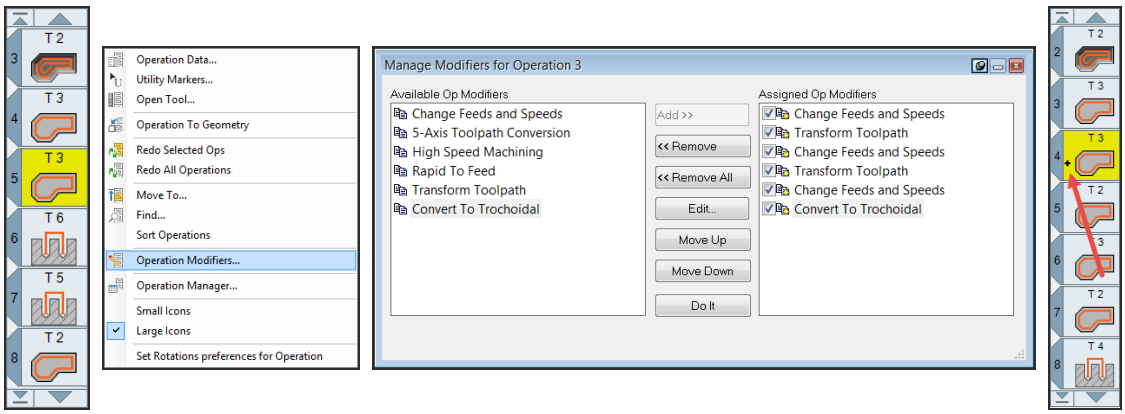
1. Contains At Op Start data
2. Contains locked values
3. Contains At Op End data

Operation Modifiers

When you change an operation by means of a plug-in, such as Transform Toolpath or HSM > Change Feeds and Speeds, the change is lost when the operation is regenerated. Operation Modifiers are slightly different, because they let you preserve the instructions you provide for changing toolpath. Each time the operation is regenerated by Redo, all operation modifiers are re-applied.

Using Operation Modifiers

Right-click an operation and choose Operation Modifiers, the Manage Modifiers dialog presents a list of operation modifiers. The left pane shows all modifiers available for the operation type; the right pane shows all that have been assigned to the selected operation.
Each time you add a modifier, its modifier dialog appears. The parameter settings for most modifier dialogs are the same as for the corresponding plug-in, except for the omission of controls for selecting operations or creating new operations. You can add multiple instances of some modifiers, such as Change Feeds and Speeds. Others, such as 5-Axis Toolpath Conversion, can be added only once.
In the list of assigned modifiers, you can change the order in which they are applied, and you can disable or enable one or more modifiers. Click the Edit button (or `double-click` the modifier) to view or edit parameter settings.
When you click Do It, the operation is modified: The operation tile is marked with a plus sign (+) in its lower left corner, and the corresponding toolpath is regenerated.
You can combine multiple Operation Modifiers, but you need to exercise caution applying two (or more) modifiers that both change toolpath. The general guideline is: *Use common sense*. Some specific caveats are noted below.



Notes for Using Operation Modifiers

- For detailed information on any particular operation modifier, see the *Plug-Ins* guide. Operation modifiers are like plug-ins in most respects, except that they persist and are re-applied each time the operation is regenerated.
- Operation modifier data is stored with each operation, ***not*** with the process or process group. For example, if a process group is associated with three operations, applying a modifier to the second operation has no effect on the first or third.
- You cannot save modifiers independently of the operation. You cannot copy or transfer modifiers from one operation to another.
- Modifiers are applied sequentially, without inter-modifier communication: The ending state of modifier *n* is the starting state of modifier *n+1*. Therefore, the toolpath could change significantly if the same set of modifiers were to be stacked in a different order.
- Clicking CAM palette button Redo preserves existing modifiers, but clicking CAM palette button Do It does not; it creates one or more new operations without copying modifiers.
- Do not combine a conversion modifier (such as 5-Axis Toolpath Conversion or Convert to Trochoidal) with any other modifier, except to *follow* it with Change Feeds and Speeds or Transform Toolpath.

Modifying Operations

Changes can be made to Operations in the following ways:

To recreate operation Processes:

Double-click the Operation tile in the Operations List. The Process tiles are recreated. Geometry is selected and any machining markers are repositioned as they were when the operation was processed. Any operations created from the same Process List as the selected operation are selected.

To replace selected operations:

1. In the Operation List, double-click the operation you want to replace.
   The operation tiles associated with the operation highlight and the corresponding processes appear in the Process List.
2. In the process List, double-click the tile for the process you want to change.
   The Process dialog appears.
3. Type the information or select the option you want to change.
4. Click the Redo button. Selected operations are replaced with the modified operations.

This is useful when multiple operations are created from a single Process tile, because you can make changes to one operation without changing the other operations created from the same process. For more information, see Operation Tile List.

To change an operation:

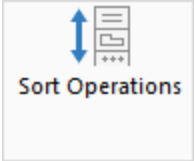
1. Right-click the Operation tile and select Operation Data.
2. Change or lock values, such as clearance planes and feeds and speeds.

3. Close the Operations dialog

To reprocess all operations in a part file:

From the Edit menu, select Redo All Ops . The system recalculates all of the toolpaths, and positioning moves based on the new order of operations.

To reorganize operations by tool number and creation order:

Select Sort 

. The operations sort by tool number and creation order.

You can also manually rearrange the Operation List by moving tiles to different locations in the list.

The order of machining in the finished NC program corresponds to the order of the operations in the Operation List. Therefore, the order of Operation tiles in the Operation List is very important. Efficient use of multiple process programming may produce operations in a less-than-optimal machining order. You can reorganize the Operation List to create a more optimal machining order. However, consider the following when reordering operations. When using the Auto Clearance option or the Material Only option, the system takes into account the material conditions when it creates the positioning moves and toolpath for each operation.

Changing the order of operations can change the initial material conditions for existing operations. If you change the order of operations, add operations, or remove operations from the list, be sure to check the toolpaths and positioning moves. Render the part to check for tool interference, unnecessary moves, or incorrect positioning moves. If adjustments are necessary, you must reprocess the operations.

Reprocessing all operations in a part file is very easy using the Redo All Ops item under the Edit menu. When the operations are reprocessed, the system recalculates all of the toolpaths and positioning moves based on the new order of operations.

Sorting Operations

To sort operations:

Right-click a tile in the Operations List or a row in the Operation report and select Sort Operations. The Sort Operations function reorganizes all operations in the Operations List. Sorting first looks at CS's and splits the Operations within the CS block. It then reorganizes the operations by tool number and creation order, from lowest to highest tool number. Next it looks at the following block of Operations on the same CS, etc. 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.

Before Sort

Op#	Op Type	TI Type	CS
1	Contour	Center Drill	1. XY plane
2	Pocket	Bore	1. XY plane
3	Pocket	Finish EM	2. G55
4	Pocket	Finish EM	2. G55
5	Contour	Center Drill	1. XY plane
6	Contour	Ball EM	2. G55
7	Contour	Ball EM	2. G55
8	Pocket	Convex Tip	2. G55
9	Contour	Drill	1. XY plane
10	Drill	Rough EM	1. XY plane
11	Pocket	Tap	1. XY plane
12	Contour	Center Drill	1. XY plane
13	Contour	Ball EM	2. G55
14	Contour	Ball EM	2. G55
15	Pocket	Bore	1. XY plane
16	Pocket	Face	1. XY plane
17	Pocket	Finish EM	1. XY plane
18	Pocket	Finish EM	1. XY plane
19	Pocket	Finish EM	1. XY plane
20	Contour	Finish EM	2. G55
21	Drill	Finish EM	2. G55
22	Pocket	Convex Tip	2. G55
23	Contour	Center Drill	1. XY plane
24	Contour	Center Drill	1. XY plane
25	Pocket	Barrel	2. G55
26	Contour	Ball EM	2. G55

After Sort

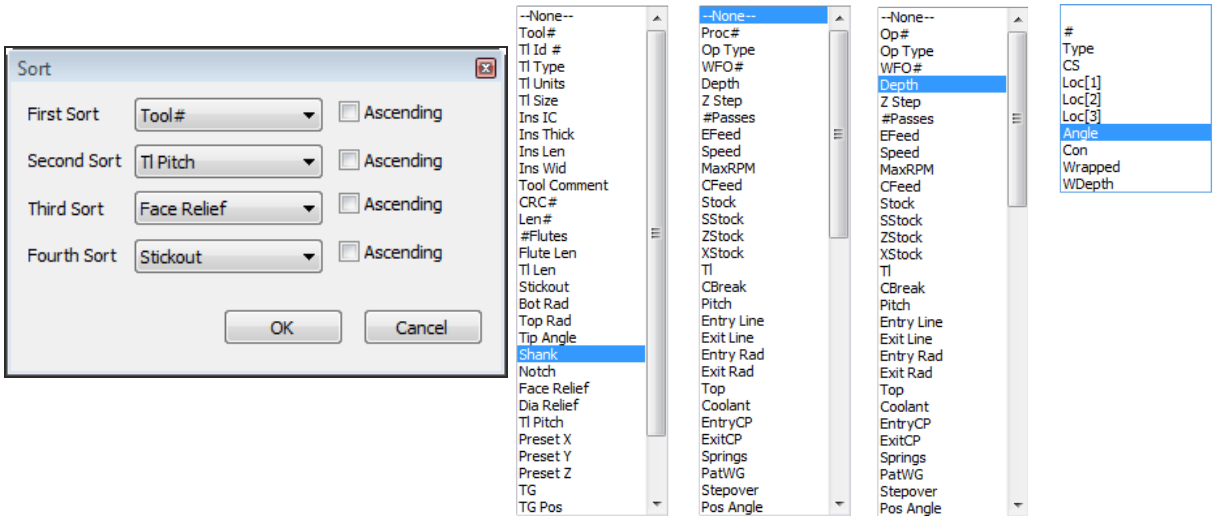
Op#	Op Type	TI Type	CS
1	Pocket	Bore	1. XY plane
2	Contour	Center Drill	1. XY plane
3	Pocket	Finish EM	2. G55
4	Pocket	Finish EM	2. G55
5	Contour	Center Drill	1. XY plane
6	Pocket	Convex Tip	2. G55
7	Contour	Ball EM	2. G55
8	Contour	Ball EM	2. G55
9	Contour	Drill	1. XY plane
10	Drill	Rough EM	1. XY plane
11	Pocket	Tap	1. XY plane
12	Contour	Center Drill	1. XY plane
13	Contour	Ball EM	2. G55
14	Contour	Ball EM	2. G55
15	Pocket	Face	1. XY plane
16	Pocket	Finish EM	1. XY plane
17	Pocket	Finish EM	1. XY plane
18	Pocket	Finish EM	1. XY plane
19	Pocket	Bore	1. XY plane
20	Contour	Finish EM	2. G55
21	Drill	Finish EM	2. G55
22	Pocket	Convex Tip	2. G55
23	Contour	Center Drill	1. XY plane
24	Contour	Center Drill	1. XY plane
25	Pocket	Barrel	2. G55
26	Contour	Ball EM	2. G55

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.

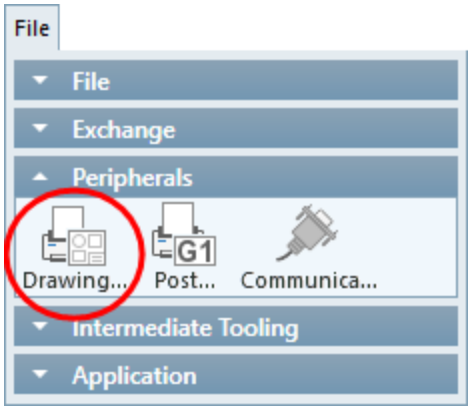
> ⓘ Please be aware that sorting operations can be risky. For example, you could potentially tell the system to tap a hole before it is drilled. Please be sure to review the results of the sort to ensure you get the results you want.

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.



Printing the Toolpath



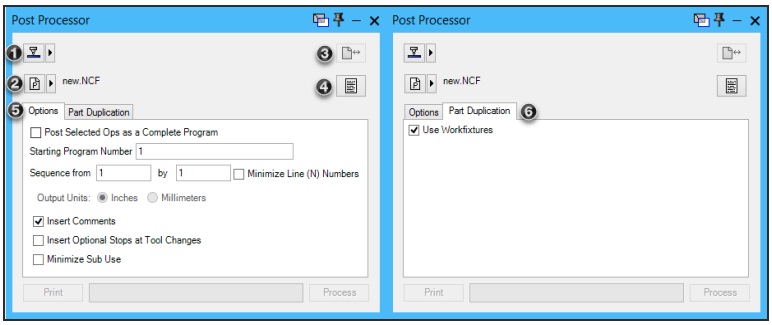
After an operation has been created, the resulting toolpath can be printed. The Printing Preferences section of the Preferences>Display Tab specifies how the system will handle the background color and line contrast. You can choose to print black and white, full color or color on a white background. When the desired toolpath is on the screen, choose Drawing from the Peripherals sub-menu in the File Main menu dropdown.

Cut Part Rendering

Cut Part Rendering is the process of running a visual inspection of the operations you've created. Rendering steps through each operation, displaying the movement made by each tool as it cuts the part. Once you have one or more operations you can render the part for a visual inspection. This can be very helpful in catching any errors in the toolpath. Rendering is accessed by clicking the Sim button in the Command Toolbar. For more information on CPR see the section on Rendering in the *Common Reference* guide.

Post Processing

After operations to machine the part are created and verified, the file needs to be post processed. Post Processing creates a text file (NC Program) that can be transferred to the machine control, from a part file (a VNC file). Post Processors specific to particular machine controls are used to create the text file from the VNC file. This is all accomplished in the Post Processor dialog. The Post Processor dialog is accessed by clicking on the Post Processor button in the Commands palette.
Posting a part is very easy. Once the dialog is open, simply select a Post Processor to use, give the posted G-Code a name, set any parameters desired and click on Process. Once the G-code is written, you can send the output to the control.



1. Post Processor Selection
2. Program Name
3. Communications
4. Text Window
5. Output
6. Multiple Parts

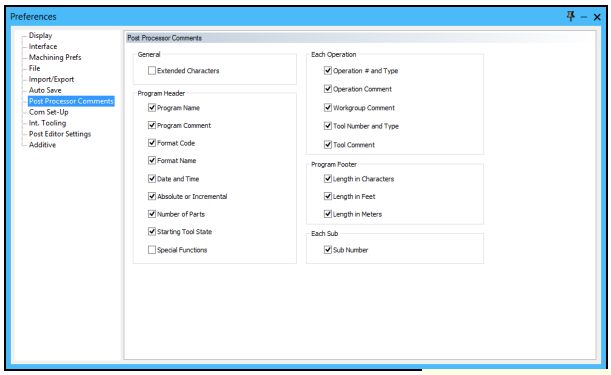
The Post Processor dialog

Use the File > Preferences > G-Code Editor Settings screen to set which Editor you will be using to edit your output files. For more details, see the *Common Reference* guide.

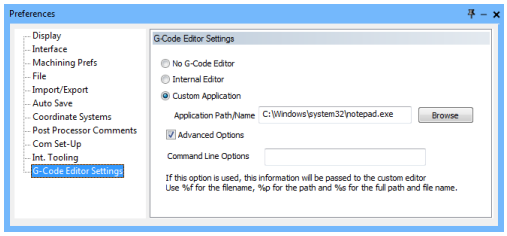
Posting Step-By-Step

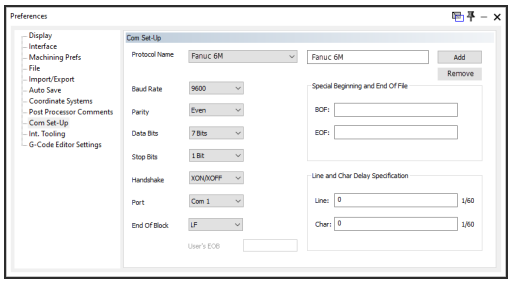
After operations to machine the part are created and verified, the file needs to be post processed. Post Processing creates a text file (NC Program) that can be transferred to the machine control from a part file (a VNC file). Post Processors specific to particular machine controls are used to create the text file from the VNC file. This is all accomplished in the **Post Processor** dialog. The Post Processor dialog is accessed by clicking on the Post Processor command button in the Commands palette.
Posting a part is very easy. Once the dialog is open, simply select a Post Processor to use, give the posted G-Code a name, set any parameters desired and click on Process. Once the G-code is written, you can send the output to the control.
The following will walk you through Post Processing a part file.

1. The first time you use a version of GibbsCAM, you should set your posting preferences. The Post Processor Comments preference tab allows you 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. This dialog can be found in the File menu under Preferences. This only needs to be done the first time you use a version, but you can change it for each program if that is your choice.

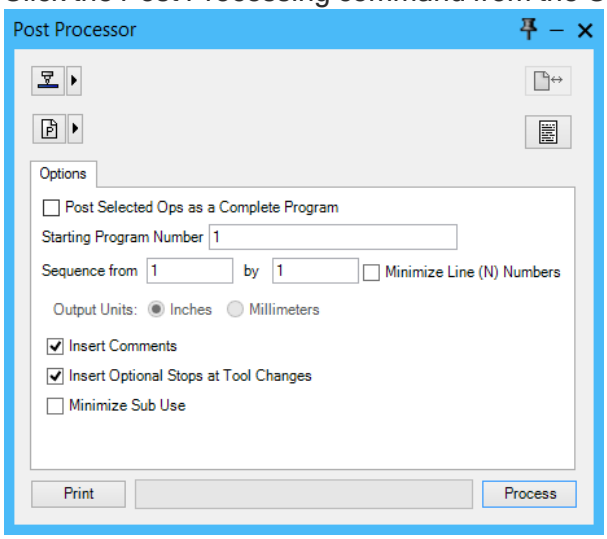


You should also at this time set your G-Code Editor Settings and Com Set-Up Preferences.

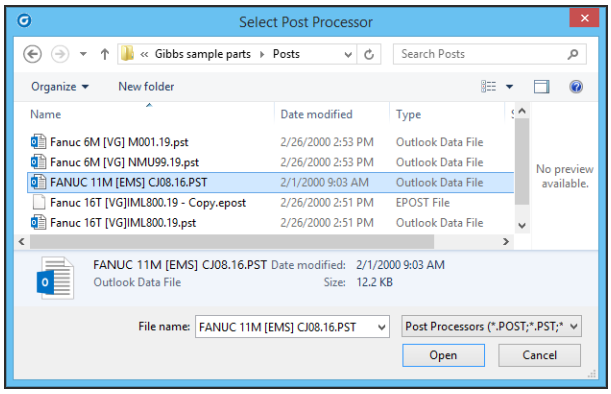




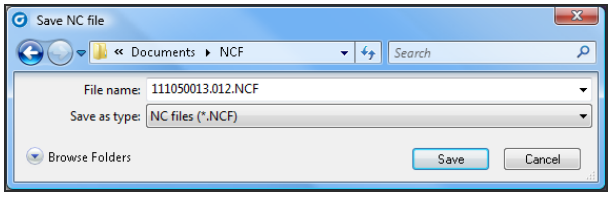
2. Click the Post Processing command from the Command Toolbar .



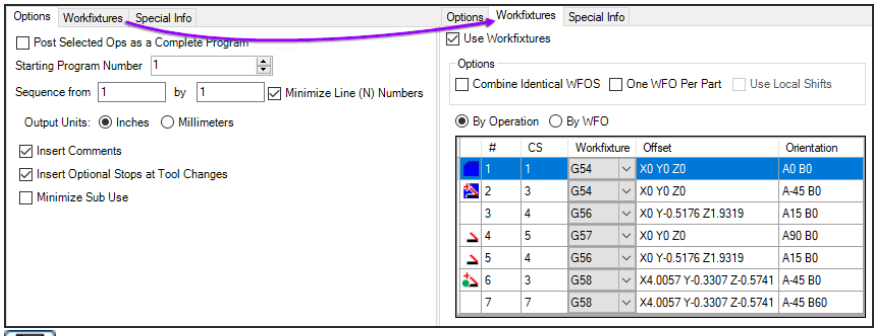
3. Select a Post Processor.
Clicking on the Post Processor Selection button will open the **Select Post Processor** dialog. Navigate to where you keep your Post Processors, select the file and click Open. This defines the post selection for the part.



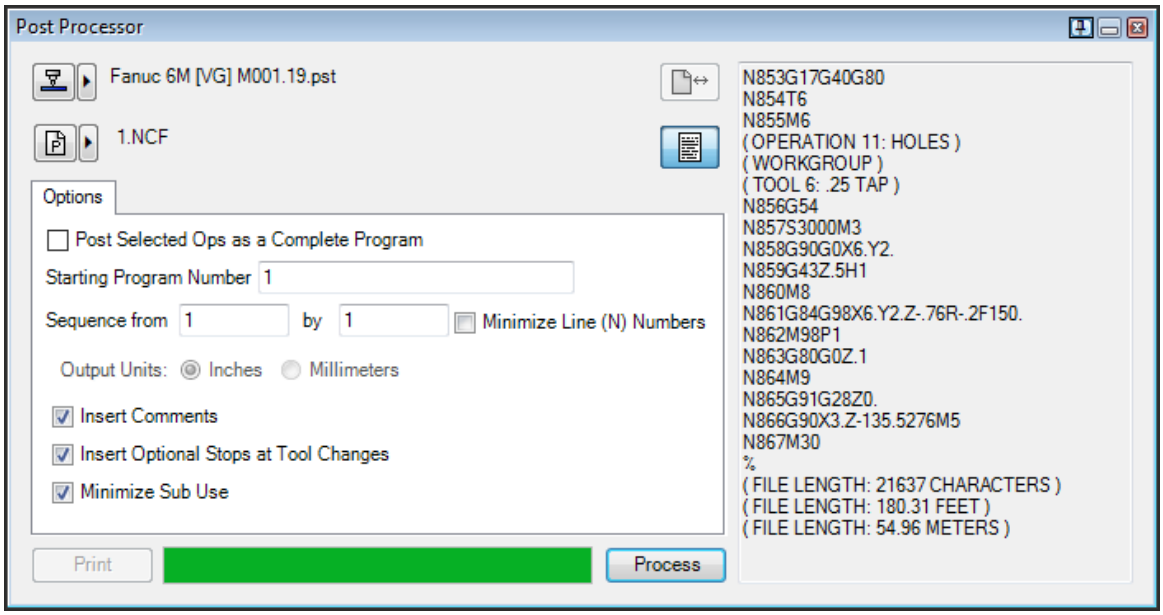
4. Name the program.
Clicking on the Program Name button will open the **Save NC File** dialog. Navigate to where you keep your posted output files, name the file and click Save. This creates an .NCF file for the code that is to be generated.



5. Set any desired output options.
There are a number of options you may set in the Options and Part Duplication sections of the **Post Processor** dialog. For more details see the *Mill* Guide.



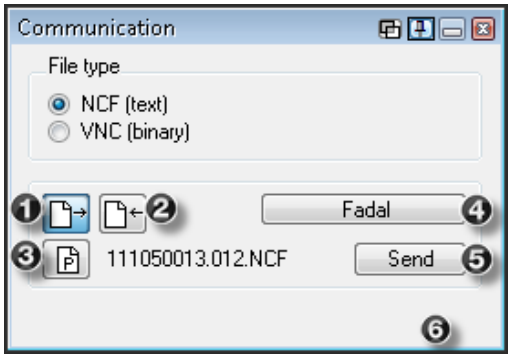
6. Open the Text window.
If you have not specified a Post Editor to use for the output from the Preferences menu and you want to view the code, open the Text window by clicking the Text window button. This opens a fly-out window to the side which is designed to view the posted code as it is being generated.



7. Process Click Process.
Clicking the Process button will generate the code which will be sent to the control. If you have selected a Custom Application from the Preferences menu, the code will be displayed in the application after it is generated. This may take a few minutes for large programs, so do not be surprised if the application does not open immediately.

8. Save Save the file.
Once the code is generated, save the file, either by clicking Save or by using the Save option from the File menu of the Post Editor application you have selected.

9. Send the file to the control.
Once the code is generated you may send the NCF file to the control. Click the File > Communication command to open the Communication window. The name of the NCF file you have saved should be visible next to the Program button. Click Send to send the posted output to the control.



1. Send File
2. Receive File
3. Program Selection
4. Control Selection
5. Send/Receive/Stop
6. Progress Bar

Communications

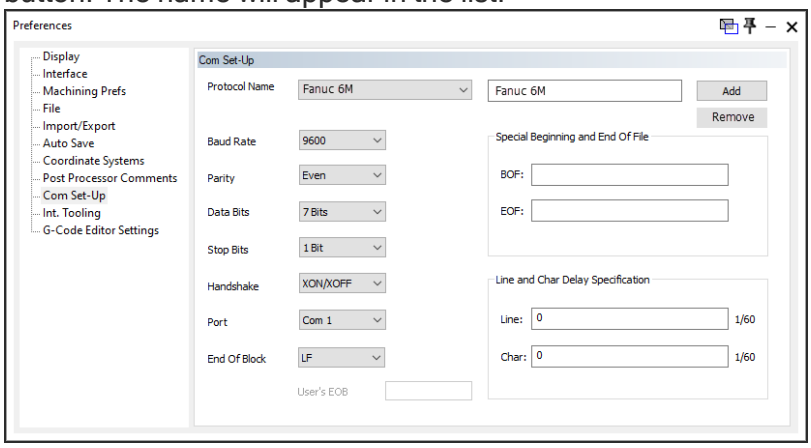
The system contains integrated communications. Third party communications packages can also be used to communicate with CNC machines. Before data can be sent to the CNC machine, the communication parameters need to be set up. To access the Com Set-Up tab choose File > Preferences. This dialog is used to set up communication protocols needed for sending a file to a control or receiving a file from the control. Different controls have different protocols (parameters). Refer to the machine control manual for the necessary protocol specifications.

- Protocols
- Communicating with a CNC

Protocols

Adding

To add a new protocol, type a new name and change the settings for the machine. Click the Add button. The name will appear in the list.



Changing

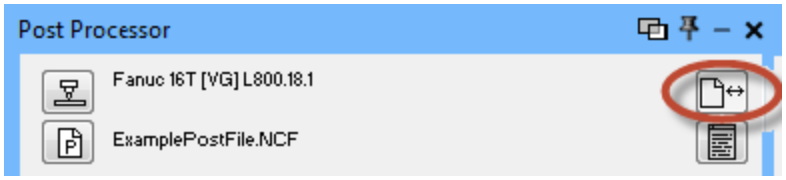
To change a protocol, select it from the protocol list and modify the information. The changes are automatically saved.

Removing

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

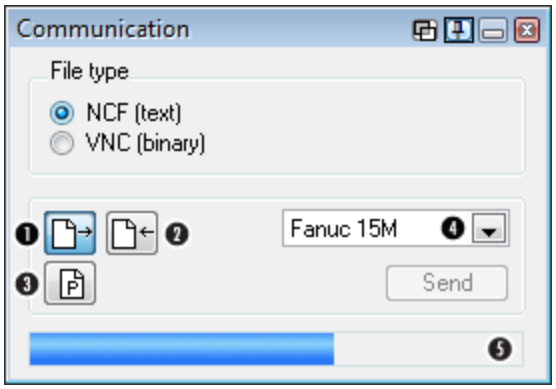
Communicating with a CNC

To send a file to or receive a file from a CNC machine, click the Communication button in the Post Processor dialog.



Communication dialog

The File > Communication dialog is used to send or receive files to and from a CNC. You may define the File type for the transmission, either a post (ASCII) or part file (binary).



1. Send Mode
2. Receive Mode
3. Program Name
4. Protocol Menu
5. Progress Bar

Sending a File to the Control

To send a file to the control: Ensure the Send Mode is on, choose the protocol, choose the file with the Program Name button and click Send. Click the Stop button to cancel. The Progress Bar indicates the status of the transmission.

Receiving a File from the Control

To receive a file from the control, open the Communication dialog from the Post Processor dialog or from the File menu. The Receive Mode button should be depressed. Choose the correct protocol from the Protocol pop-up menu. The name of the received program is specified by clicking on the Program Name button. When everything is set correctly, click on the Receive button, and then send the program from the control. If the Text Window is open from the Post Processor dialog, the program will scroll by as it is received.

> It is recommended that edited NCF files received back from the machine control be saved under a different name than the original NCF file that was initially sent to the control. That way if the original VNC file is reprocessed, it won't affect the edited NCF file. For example, a part file named SAMPLE.VNC is post processed and a text file named SAMPLE.NCF is created. Changes are made to the program at the machine control and the new file containing those changes is sent back and received at the offline computer, but under the name SAMPLE1.NCF. If SAMPLE.VNC is reprocessed again at a later date, it won't destroy the SAMPLE1.NCF file that contains the changes that were made at the machine.

Sending Other .NCF Files

Any text file that matches the extension set in the Post text box in the File Extensions Preference can be sent to the control. Refer to the Post Processing chapter for more information on output file extensions.

Conventions

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

Text

`Screen text`. Text with `this appearance` indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.
`Keystroke/Mouse`. Text with `this appearance` indicates a keystroke or mouse action, such as `Ctrl+C` or `right-click`.
`Code`. Text with `this appearance` indicates computer code, such as lines in a macro or a block of G-code.

Graphics

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

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

Links to Online Resources

Please contact your reseller for support.

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

Appendix

Glossary

This glossary is intended to provide clarifications on terms and their usage throughout the GibbsCAM guides. Some of the terms may, at first glance, appear common to the industry, but many have specific meaning to GibbsCAM products.

2.5D Solids	The 2.5D Solids module allows you to import and modify or create solid models. The models can be pocketed and contoured without the use of geometry.
Advanced CS	The Advanced CS module extends the capabilities of the Production Mill module. Advanced CS provides for the creation of multiple coordinate systems for creating and machining geometry. Capabilities include 3D geometry, 4th-axis and 5th-axis rotary positioning, tombstone machining, and/or work fixture offsets.
Body	A body is a term used for solid and sheet objects created by the system Each body is a single object composed of faces and the area enclosed by the faces. Bodies are used as the building blocks for creating part models.
Closed Shape	A closed shape is a set of geometry that does not have a discernible start or end point. The shape is a loop.
Coordinate System	A coordinate system (CS) is a plane in space with an origin and three axes. The axes are the horizontal, vertical, and depth axes. The standard coordinate system is the XY plane (milling) or the ZX plane (turning). The XY plane's axes are the X (horizontal) axis, the Y (vertical) axis, and the Z (depth) axis. Other standard coordinate systems are the XZ and YZ planes. Non-standard CSs are defined using horizontal, vertical, and depth axes, labeled as HVD.
Edge	An edge is a curve or line between two faces. A body must have two faces connected at every edge.
Face	A face is one surface of a body or sheet. Faces are surfaces that have knowledge of the surfaces that surround them. For example, one side of a cube would be considered a face. Each face is bounded by loops. A simple face is surrounded by one loop.
Geometry Expert	Geometry Expert is a means of creating connected geometry through an interface that is similar to a spreadsheet.
Half Point	Term referring to entering a value for a single axis (X, Y, Z, etc.) when creating geometry using Geometry Expert. Geometry Expert may only need the dimension of a single axis to create a feature due to existing tangencies.

List	GibbsCAMsoftware presents three lists: Tool List, Process List, and Operation List. The Tool List contains all of the tools used to machine the current part. The Process List contains temporary data that is used to define operations. The Operation List contains final operations that machine a part.
Loop	A loop is a series of connected edges that outline a face.
Machining Markers	Machining Markers are used when creating mill contour or turning operations. The Machining Markers specify start and end points, climb or conventional cutting, and the side of the geometry to cut.
MDD (Machining Definition Document)	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. Each item in the Document Control dialog's Machine Type menu is a separate MDD. MDDs include 3-axis, 4-axis, or 5-axis mills, 2-axis or 3-axis lathes, Wire EDM machines, and others.
Modeling	Modeling is the process of defining a part's shape and dimensions on a computer. Common types of modeling include wireframe modeling (both 2D and 3D), surface modeling, and solid modeling. Solid Modeling is the method of defining a part as a solid object rather than as a wireframe or collection of surfaces.
Multi-Task Machining	The Multi-Task Machining (MTM) module extends the capabilities of the Production Turning module. MTM supports programming machines with multiple spindles and multiple turrets.
Open Shape	An open shape is a set of geometry that is not fully connected. There is a definite start point and end point to the shape. The open shape may or may not be terminated.
Origin	The origin is the center of a coordinate system. The origin's location is H0, V0, D0. It is where the axes meet.
Production	Production refers to basic 2D GibbsCAM functionality, generally on a module basis. For example, both the Mill and Turning modules are Production modules. Additionally, the *Geometry Creation* guide would be referred to as a Production guide.
Sheet	A sheet is the term used for surfaces created or imported by the system. A sheet is an object composed of faces, but a sheet has no volume or thickness. It is only composed of its own faces.
Solids Import	Solids Import is a module that allows a user to import a solid model and

	perform simple modifications to the model, including geometry extraction. All machining operations are performed upon extracted geometry.
SolidSurfacer	SolidSurfacer is the segment of the GibbsCAM product that provides 3D solid model creation and the direct machining of 3D solids and sheets. The SolidSurfacer module requires the Advanced CS module.
Tile	A tile is an entry in a list. There are Tool tiles, Process tiles and Operation tiles. Double-click a tile to access its contents or set its parameters.
Utility Markers	Utility Markers are manual overrides of the feedrates and other facets of operations. Typically they are used to lower the feedrate when entering a corner and speed up again once leaving a corner.
Vertex	A vertex is the common endpoint of two edges.
Workgroup	A workgroup can be thought of as a layer that separates groups of geometry. This helps keep the Workspace uncluttered. Workgroups can also be used to define custom stock based on the geometry in them.