Version 14 : September 2020

Features

Proprietary Notice

Contents

Introduction

This guide describes the functionality and use of the Hole Wizard and Hole Manager features. You should have some familiarity with the *Getting Started*, *Geometry Creation*, and *Mill* guides before using this document. Additionally, some experience with one of the GibbsCAM solids packages is useful for an in-depth understanding of the Hole Manager. However, much of the functionality of these features is easy to understand simply by using the features and reading the introductory information in this guide.

You use the Hole Wizard to specify hole processes for a set of hole types. Wizards do not necessarily provide any new functions to the system but do offer a new and easier way of using existing features. The Hole Wizard provides you with a step-by-step guide through the hole creation process in four easy steps, described in Hole Wizard Basics.

The Hole Manager enables you to define and control points, circles, or hole features on a solid. Each point or feature can be defined as a particular type of hole. The Hole Wizard eases the process of generating G-code for machining holes. The Hole Manager section describes the various components, usage, interface and details of using the Hole Manager, Auto Feature Recognition (AFR) of holes in solids models, and group and feature lists. The Hole Manager is expressly for use with part models that have a large number of holes to be machined in a part. The Hole Manager groups similar holes to quickly define the features of these groups, and it uses the Hole Wizard to create the list of tools needed to machine the holes and create the operations.

The Features tutorial provides step-by-step instructions on using the Hole Wizard with and without AutoWiz, Hole Manager, Hole Features and the Feature Manager.

When do I use...?

You can use the drill process, Hole Wizard, or Hole Manager for drilling holes depending on the purpose and situation. To minimize confusion, the table below lists each drilling function along with a brief description and its intended application.

Function	Description & Application
Drill Process	Basic specification of what a single tool should do at each hole location.
Hole Wizard	Automated tool/process creation for basic multi-tool holes, based on Mitsubishi logic and user preferences. The Hole Wizard is not intended for all drilling processes or hole types; it simply attempts to complete an entire hole.
Hole Manager	Automatic hole definition on solids, enabling you to adjust parameters as necessary. Provides for hole groups and selection control for use by drill processes or Hole Wizard. You can apply processes and process groups to a Hole Manager selection.

Hole Wizard Basics

The Hole Wizard automates the process of making holes. The Hole Wizard can make a single drill hole or a pattern of tap holes using multiple tools on any number of holes. The Hole Wizard guides you through four easy steps to select the hole shape, define the hole size, position the holes, and build the processes and operations.

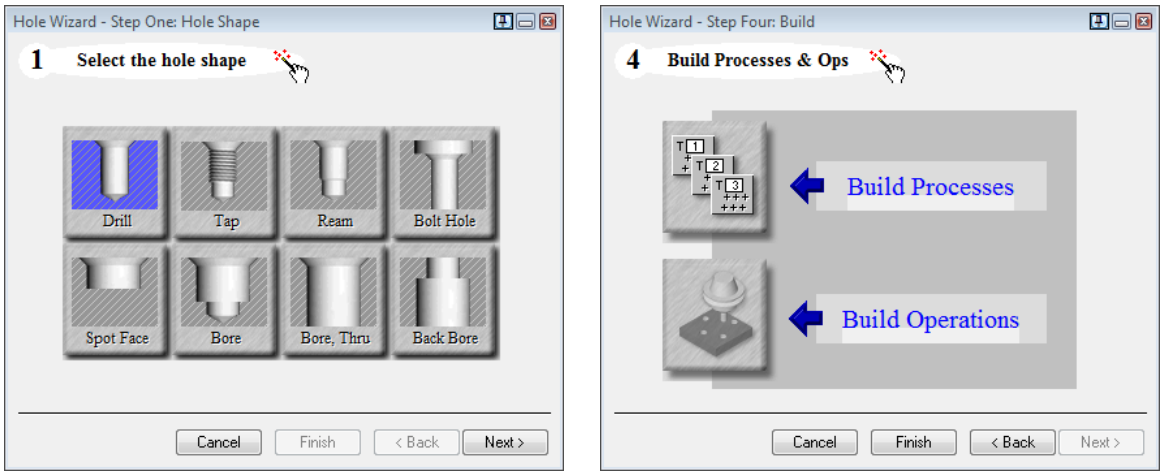


Figure 1: Two of the Hole Wizard steps

The Hole Wizard supports eight basic hole shapes including drill, tap, ream, bolt hole, spot face, bore, bore thru, and back bore. For each type of hole, the Hole Wizard automatically selects the required tools, determines and defines machining processes, then builds the operations to make the holes. All of this is done through your “Knowledge Base.” Using a bore hole as an example, the Hole Wizard automatically spots, chamfers, pre-drills rough bores, medium bores, and finish bores as needed. It will even mill bore and/or mill chamfer the hole, if a bore tool is not available.

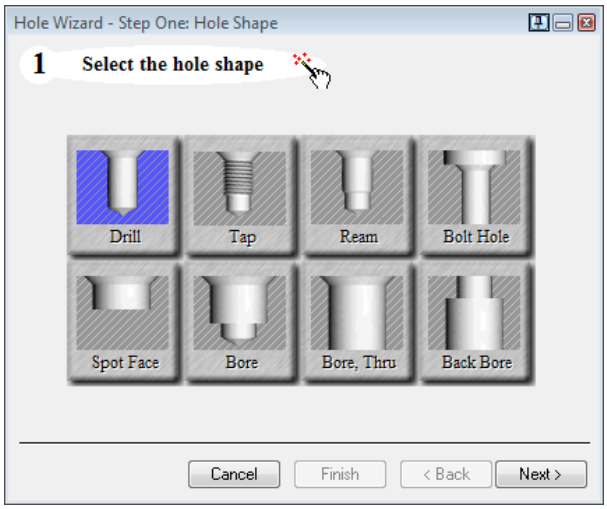
Comparing the Hole Wizard to the standard GibbsCAM system, we would say that the standard system is relatively blind. You lead the standard system along, step by step. To create a tapped hole, you select a specific spot drill in the first process you create, a specific center drill in the second process you create, and a tap in the third process you create. After setting up these three processes, you specify the point to machine and create operations. The system does not know it is making a tapped hole, just a series of drilling operations that happen to be at the same place. To automate making the same exact hole again, you can save the processes.

The Hole Wizard, unlike the standard process, leads you through the hole creation process. The Hole Wizard knows what a hole is and knows the difference between a bolt hole and a bored hole. With the Hole Wizard, you can simply state the shape of the hole, the tools that may be used, and where to make the hole. The Hole Wizard creates all of the processes and operations for you.

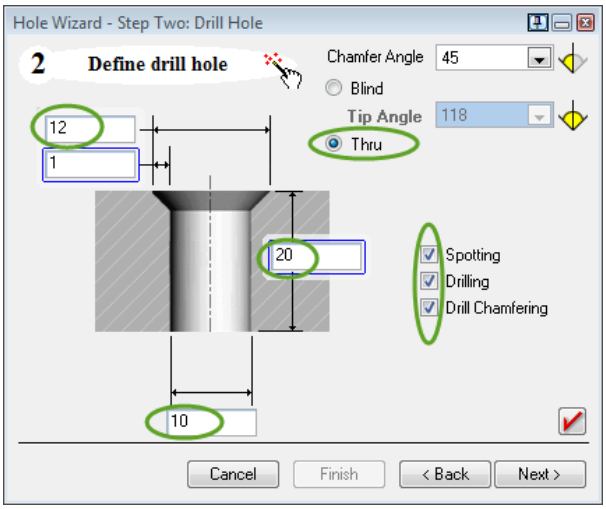
Why Use the Hole Wizard?

The Hole Wizard automates the process of making holes to reduce part programming time. The Hole Wizard follows your commands for hole creation and tool selection preferences. If you machine many similar parts, similar materials, and use similar methods, the Hole Wizard can significantly automate hole creation. If you work with widely varying parts, the Hole Wizard can speed up programming.

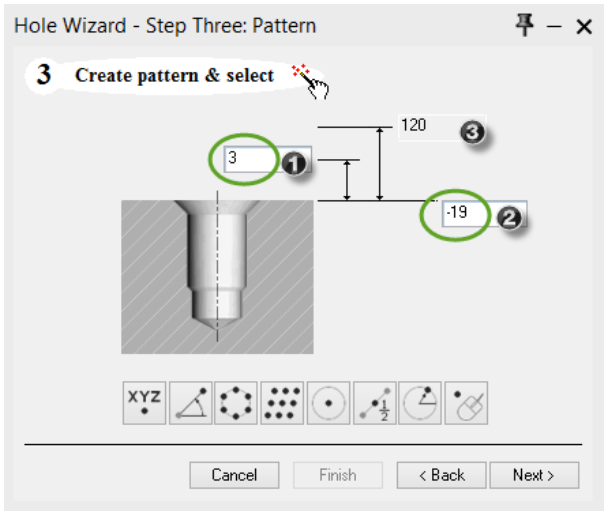
The Hole Wizard guides you through four simple steps to create holes, shown in the following images.



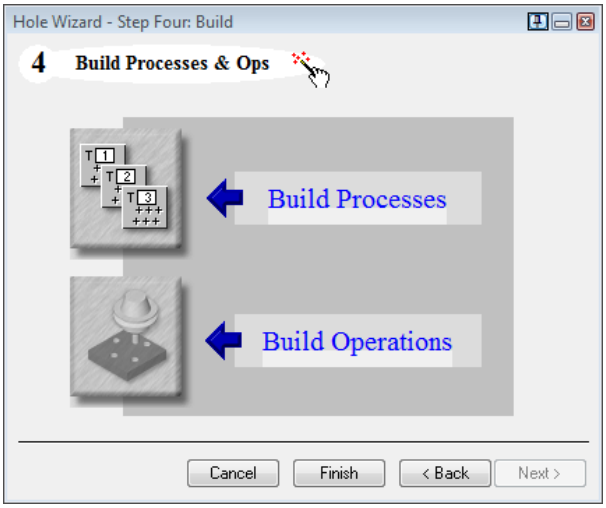
1. Select the type of hole.



2. Define the size of the hole.



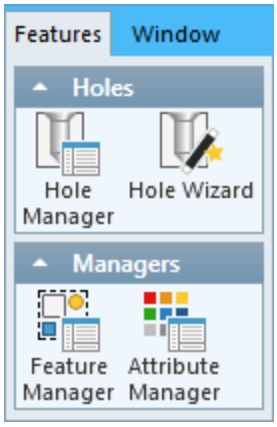
3. Select geometry and set the clearance values.



4. Build the processes and operations.

Open the Hole Wizard

You can find the Hole Wizard in the Top level menu Features> Holes.



Elements of the Hole Wizard

The following sections describe elements and features that are specific to the Hole Wizard or are a product of its capabilities, including Preferences, Document Control dialog, Hole Wizard dialog, and Point Creation palette.

Preferences

Hole Data, Tap Table and Bolt Table Data Preferences can be set in the Preferences > Machining Prefs tab. These items constitute the Hole Wizard's Knowledge Base. Default data is entered in these dialogs but you may change them as you see fit. See the Preferences section starting on Preferences for more information on the Knowledge Base data.

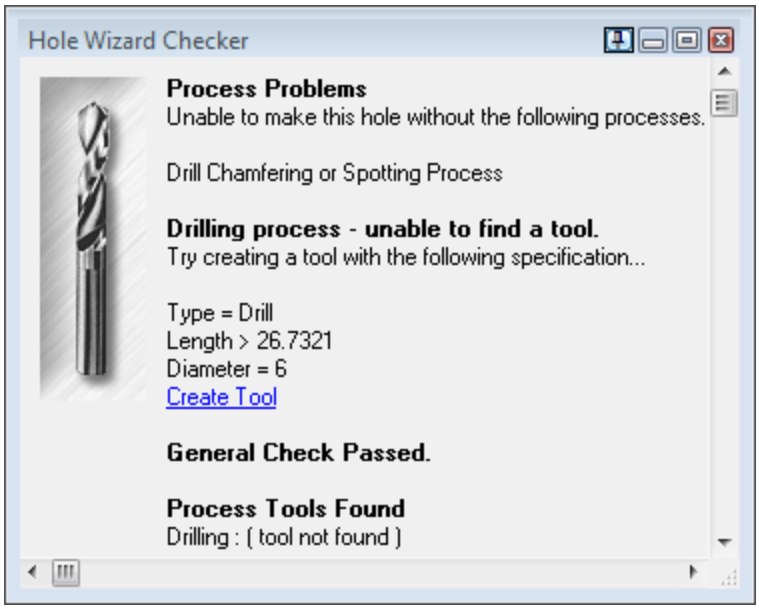
Hole Wizard Dialog

The Hole Wizard is a single activity that consists of four steps. You navigate through the Wizard using the action buttons at the bottom of the dialog.

Hole Wizard Checker

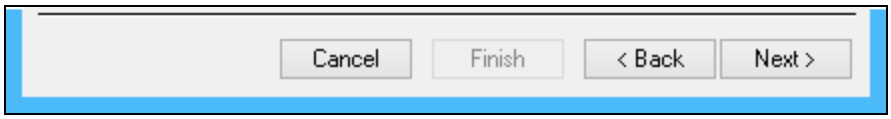
The checker is a dialog that is first accessible from Step Two. It is a utility that compares the tools needed to complete an operation with your tool list. It makes sure the specified tools are in your tool

list and alerts you when a tool is missing. It also checks the parameters of the hole being made for errors, such as negative values.



Navigation Buttons

Navigation buttons help you to move through the Hole Wizard.



Cancel
This button closes the Hole Wizard. Any changes you have made are not saved and Holes processes are not created.

Finish
This button becomes active when the dialog has sufficient data to define a Holes process. Clicking this button creates one or more processes.

Back
This button returns you to the previous step, so that you can make any needed modifications.

Next
The Next button becomes available once you supply the needed data for the current step. Clicking the Next button advances you to the next step.

Point Creation with the Hole Wizard

Wizard button:

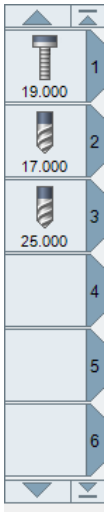
This button replaces the Return button when creating points in Step 3 of the Hole Wizard. This button appears in the point creation palette instead of the standard Return button. Pressing this button returns to the Hole Wizard.

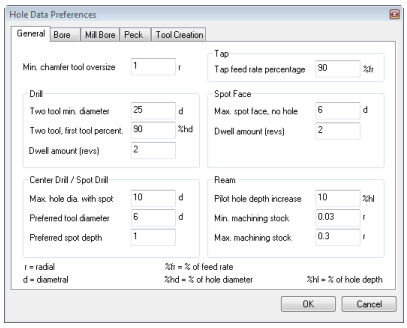
How Does the Hole Wizard Work?

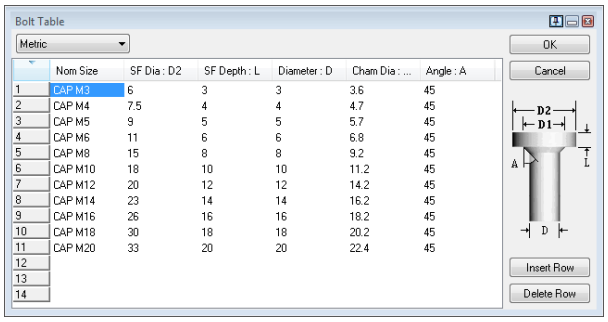
The Hole Wizard has a Knowledge Base that provides the Hole Wizard with its intelligence. The Knowledge Base is machining data that you help to define. When the Hole Wizard is presented with a hole to be machined, it looks to the Knowledge Base and figures out how to make the hole from that data.

The Knowledge Base consists of the tool list, the Preferences data, the Tap Table and the Bolt Table. The tool list is a part of the Knowledge Base because the tools in your tool list can indicate to the Hole Wizard what is to be done. For example, if you want to bore a hole the Hole Wizard will respond differently depending on what is in the tool list. If there is a bore tool, the Hole Wizard makes a bore operation, whereas if there is no bore tool but there is an end mill it can make a mill bore operation.

The Hole Wizard uses internal logic and the data in the Knowledge Base to perform hole making operations. You can modify the information in the Knowledge Base to meet your needs. You can override the Hole Wizard's suggestions at every step. For more detailed information about the internal workings of the Hole Wizard, see Appendix B: Advanced Hole Wizard Use.







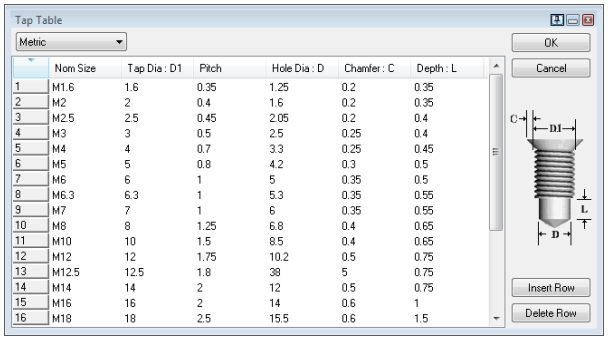


Figure 1: Elements of the Hole Wizard Knowledge Base.

Using the Hole Wizard

The Hole Wizard guides you through four steps to make a hole

- Step 1. Select the Hole Shape
- Step 2. Define the Hole Size
- Step 3. Select the Hole Position
- Step 4. Build the Processes and Operations

These steps are introduced below and detailed in the following sections.

Step 1. Select the Hole Shape

In Step 1, you select the type of hole or holes to machine from the following:

- Drill hole
- Tap hole
- Ream hole
- Bolt hole
- Spot face
- Bore hole, including through-bore and back bore

Step 2. Define the Hole Size

In Step 2, when you define the hole size, you can accept the default values or you can enter parameters as necessary. You typically use the Hole Wizard after you define the tools and geometry, but this is not a requirement. If the Hole Wizard does not find the tool or tools needed to make a hole, the Hole Wizard checker provides recommendations for the needed tools.

Step 3. Select the Hole Position

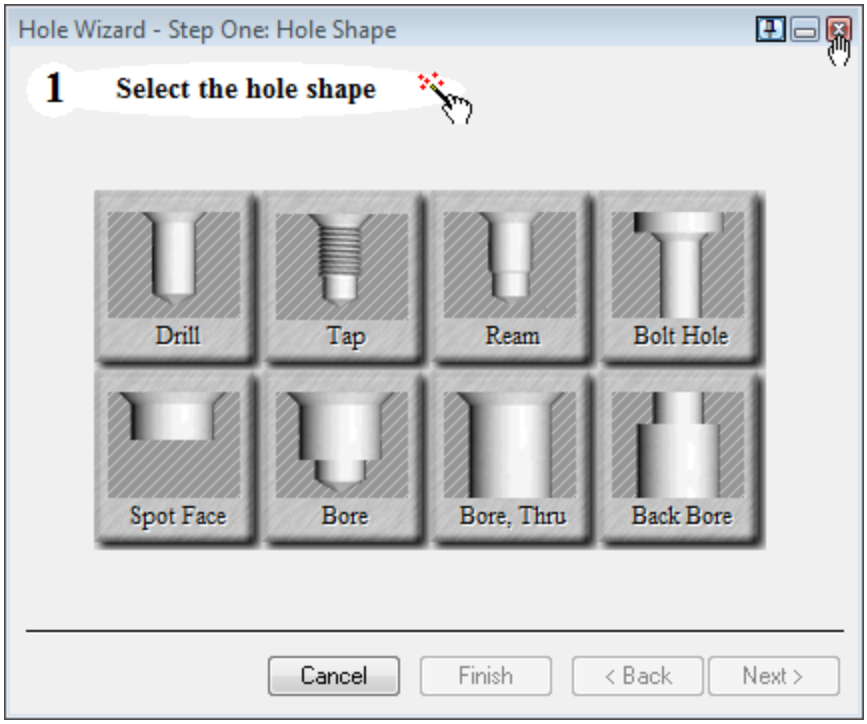
In Step 3, you can define a hole pattern or simply select existing geometry. If the geometric points for positioning the hole do not exist, you can create them while in the Hole Wizard dialog.

Step 4. Build the Processes and Operations

In Step 4, you build the processes and operations. Each process represents a machining function with a specific set of parameters and tool. Each operation applies a process to the part model to generate a toolpath for the part.

Step One - Choose the shape of the Hole

The first step to creating the hole is to define what type of hole is to be made. This dialog has a selection of eight general types of holes displayed. Each type is displayed on a selectable button that includes a graphic of the hole shape and the name of the hole type. Included are Drill, Tap, Ream, Bolt Hole, Spot Face, Bore, Bore Thru and Back Bore.



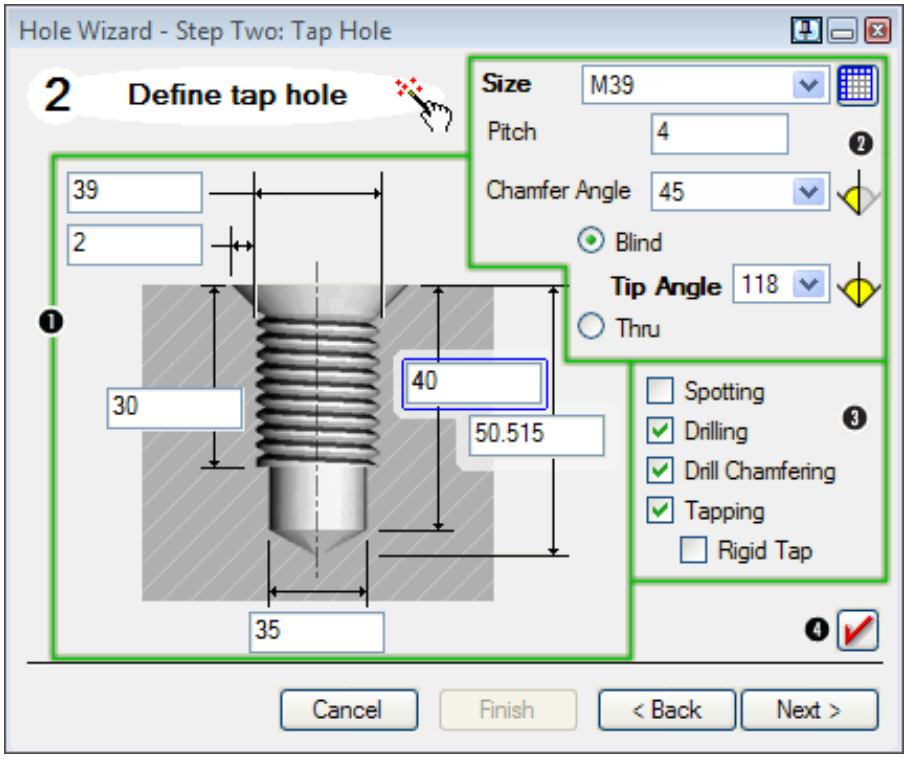
Hole Shape dialog

To select a hole shape just click the appropriate selection button. When a hole shape is selected you will automatically move on to Step Two. We will now select a Tap hole and move on to Step Two.

Step Two - Define the Hole Parameters

Once you have selected a hole shape you need to define the hole. Each of the hole shapes in Step Two will present you with a graphic of the hole. Also included in the Step Two dialog are text boxes to enter the sizes that define the hole, pull-down menus for defined data, check boxes to specify the types of machining to be performed on the hole and selection buttons to access tables and the Checker. For information specific to each hole shape, see Hole Shape Dialogs in Step Two.

Hole Dimensions:
This area of the dialog displays a section view of the hole you are machining and has text boxes to enter the parameters of the hole. Included among the parameters are the diameter, opening diameter, depth and chamfer width.



1. Hole Dimensions
2. Hole Data
3. Hole Processes
4. Checker

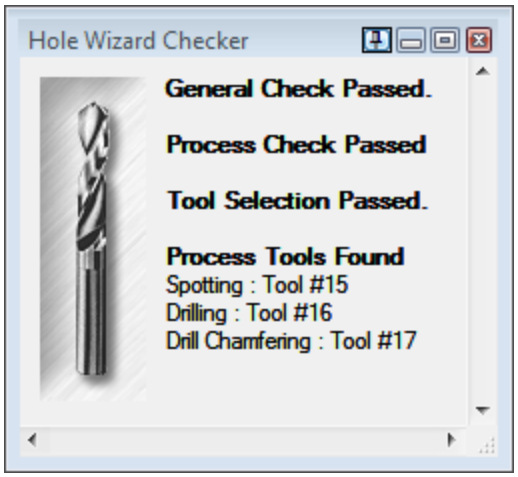
Many of the text boxes are directly related to other text boxes. As values are entered or changed in one, the related boxes will be automatically updated to reflect the changes you make. There is a white box that surrounds these text boxes. That white box tells you that these items directly affect the other. See Tap Table for details on tapped holes.

Some of the text boxes have a blue line surrounding them. This blue line indicates the last item that was changed and locks that value. The locked value will not change until you select and change it. Using the previous image again, the 39mm diameter has a blue ring around it and the 2mm chamfer width does not. If the diameter of the tap hole is modified, the chamfer diameter will not change but the chamfer width will. So if the 35mm hole diameter were changed to 33mm, the chamfer width would change to 3mm.

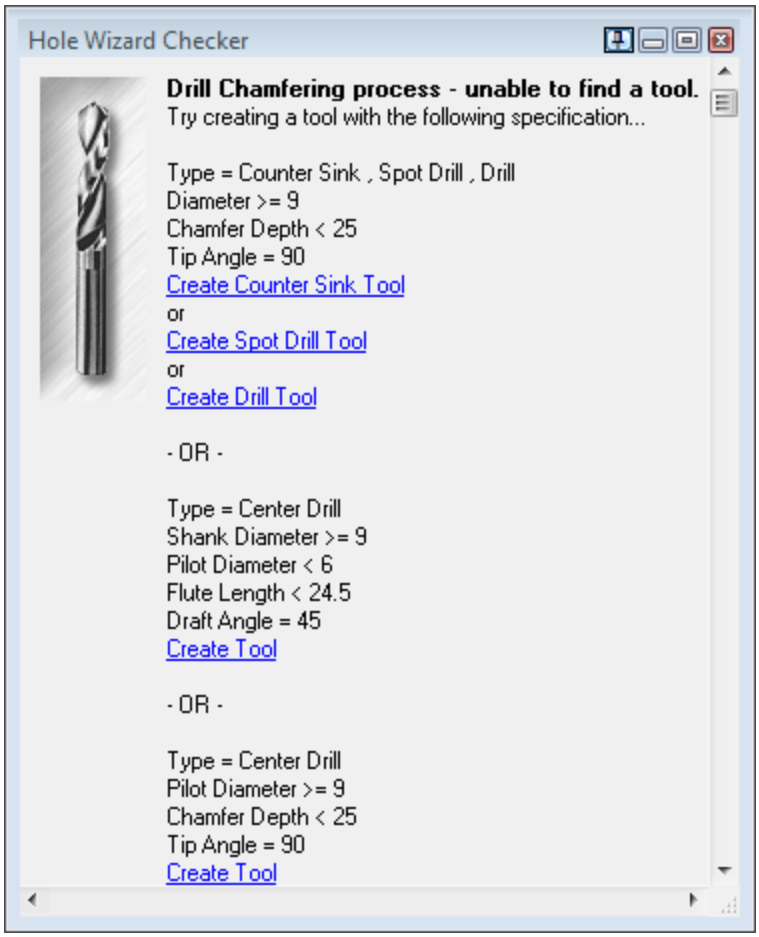
Hole Data:
This area of the dialog displays various pull-down menus, text boxes and buttons that will open the Tap or Bolt Table. Nominal sizes from your Knowledge Base and angle information is located in this section of the diagram. Here you may select predefined data for a tap or bolt hole. When the data is selected the system will fill in all appropriate text boxes automatically.

Hole Processes:
This area of the dialog displays a set of check boxes that you use to define the operations to create the hole. Each of these items is similar to a process. They each may create one or more processes to fulfill their function. The items will vary from hole shape to hole shape. Common items are Spotting, Drilling and Chamfering. The Hole Wizard will check and un-check items as the hole is defined or changed. The exact contents will change with each hole type as defined by your Knowledge Base. When you have all the correct data entered, you may check or uncheck any of these items you wish. This allows you to override the recommendations being made by the Hole Wizard.

Hole Wizard Checker:



You can click this selection button to open the Hole Wizard Checker after you set up the parameters for the hole. The Checker is a utility that analyzes the hole parameters and reviews the data entered in Step Two. The Checker also analyzes the tools needed to machine the specified hole and checks those needs against your tool list. The Checker ensures that you have the tools needed to make the hole with the parameters you define. If all the needed tools are not available, the Checker will make recommendations on the tools that are needed. You do not need to resolve the problems reported if you do not wish to. The Hole Wizard will do the best it can with what it has.



The Checker runs its tests automatically when the Next button is clicked to move to Step Three. If there are any needed tools the Checker will come up but you will be advanced to Step Three. This provides you with a reminder that tools need to be defined, and the option of automatically creating the tools, but does not keep you from defining geometry for a machining operation. The Checker will also update its results every time the button is clicked.

The tools that the Hole Wizard will use to machine this hole are highlighted in yellow in the Tool List. To continue with Step Three, click the Next button. You can also click the Back or Cancel buttons.

Step three - Select a Hole Pattern

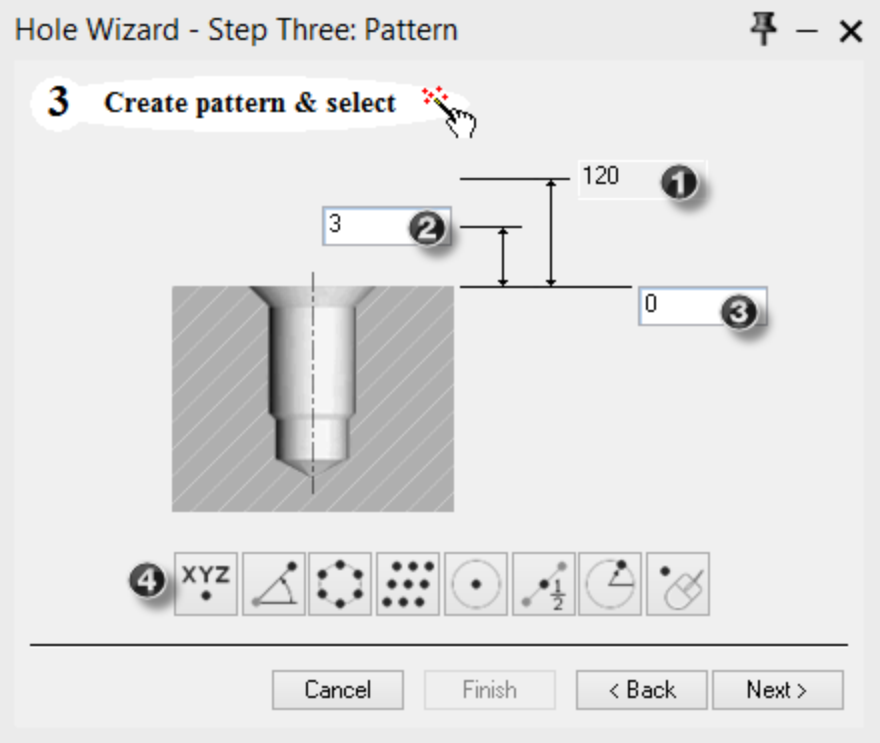
Once the hole shape has been defined you need to select the geometry to be machined and set the clearance values. If geometry does not yet exist in the part you can create it from the Hole Wizard. This is all done from Step Three.

The first thing to be done is to enter the Z value for the Entry/Exit Clearance plane and Surface Z value for the hole or holes you wish to create. The Top Clearance Z is automatically set from the Document Control dialog and cannot be modified from the Hole Wizard. All of the Z values are incremental from the Surface Z.

The second thing to be done is select the geometry to be machined. Any points or circles existing in the Workspace may be selected for machining. If the geometry already exists in the part just select it in the order you wish for it to be machined and move on to Step Four.

If the geometry does not exist it can be created from the Hole Wizard. Below the clearance plane settings is the Point sub-palette but it is integrated into the Hole Wizard. This provides access to point creation. When one of the point buttons is selected to create geometry, the Hole Wizard is

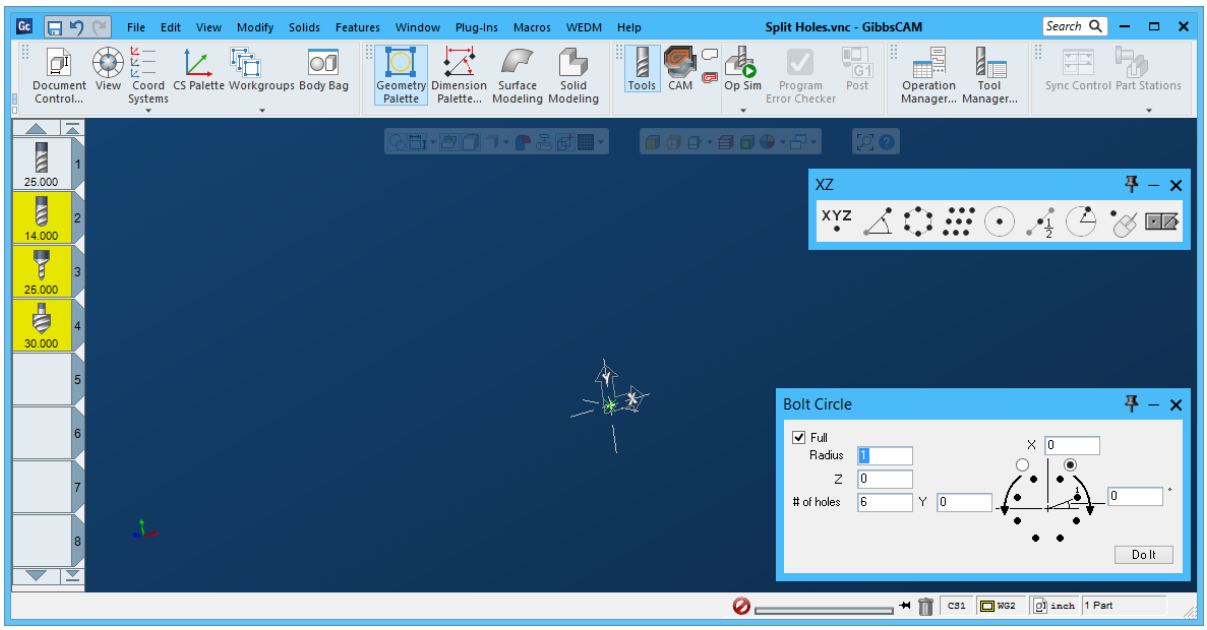
minimized to the Point palette and the selected dialog is opened. This allows you to more easily see the workspace and the points you are creating.



1. Top Clearance Z
2. Entry/Exit Clearance Z
3. Surface Z
4. Point palette

Pattern hole dialog

For example, if you were to click the Bolt Circle button while in the Hole Wizard, the Hole Wizard would be minimized to the Point palette and the Bolt Circle dialog would open. The following image shows the Hole Wizard in this minimized mode.

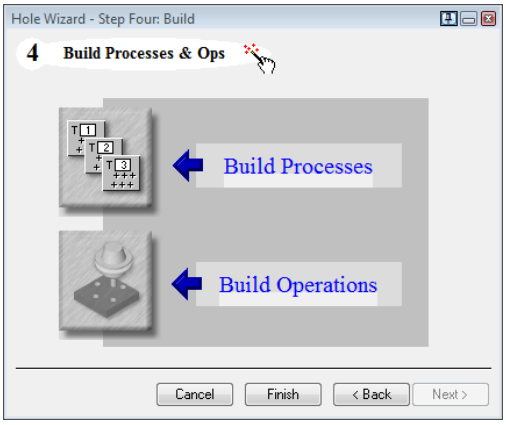


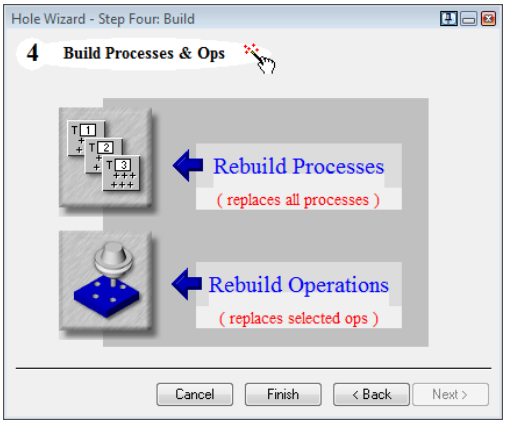
Create the points to be machined and select them in the order they are to be machined. To maximize the Hole Wizard, click the Wizard button in the Point sub-palette. This will close any open dialogs and allow you to progress to Step Four.

Clicking the Next button progresses you to Step Four. The Back or Cancel buttons are also available.

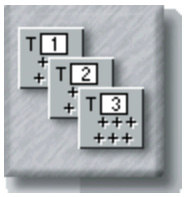
Step Four - Build the operation

The final step is to build the processes and the operations. This step creates process and operation tiles. These are the same process tiles that you would need to make if you did not use the Hole Wizard. Click each button to build the Processes and Operations. If there are existing tiles in the process or operations list, the Hole Wizard displays Rebuild instead of Build. A message appears in red stating the existing items will be replaced.





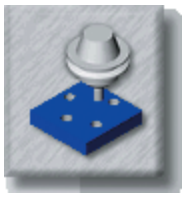
Build/Rebuild dialog



Clicking the Build Processes button creates the needed processes in the Process List. You can change the tools assigned to a process by dragging and dropping a new tool. You may also open a process dialog and change any data you wish.

If there are process tiles in the Process List the Hole Wizard will inform you that clicking this button will overwrite those processes. The message (replaces all processes) appears in red in the dialog. If you do not wish to overwrite all of the existing processes, be sure to deselect the tiles by clicking an empty space or insertion point in the Processes list.

The Rebuild Processes button returns a process to the Hole Wizard's recommendations. You can modify the information in processes in any way, including which tool is to be used. But if you click Rebuild Processes after making the modifications, the Hole Wizard loads its recommendations.

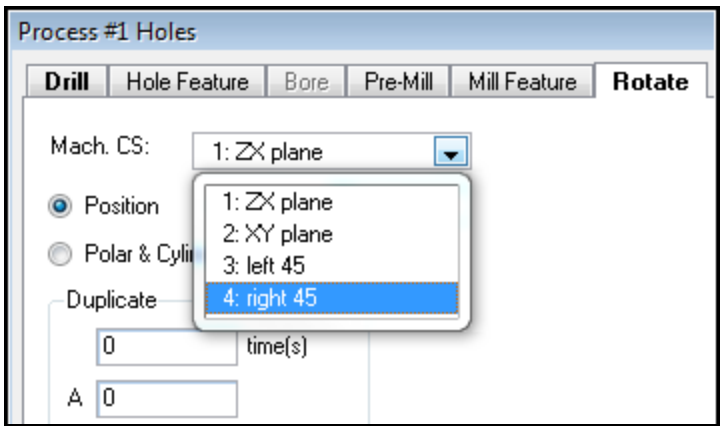


Clicking the Build Operations button creates toolpath and place operation tiles in the operations list. If there are selected operation tiles in the Operations List, the Hole Wizard displays the message (replaces selected ops) in red. If you do not want to overwrite the selected operations, be sure to click an empty space in the Operations list or on an insertion point. Note that this is different from the processes. All processes in the process list are overwritten when rebuilding processes, but only selected operations are overwritten when rebuilding operations.

Double-clicking the operation tiles allows you to modify an operation, even after the Hole Wizard is closed. If operations were created using the Hole Wizard, then the Hole Wizard appears, the processes will be loaded and geometry for the operations will be selected. The Hole Wizard will be in Step Four. Make any changes you wish to the Process data. If you wish to change the tool or hole shape data, you should return to Step Two and then forward to Step Four again. Remember that clicking Rebuild Processes will load the Hole Wizard's recommendations and overwrite any edits or changes you made earlier.

To modify a process or an operation double-click the operation tiles. This will open the Hole Wizard and load the process tiles if necessary. Open the process tile and modify the value you wish to change and click Rebuild Operations in the Hole Wizard. Rebuild Processes will reset the values and overwrite any changes you have made.

Coordinate Systems other than XY



You can use the Hole Wizard with geometry that is not in the standard XY plane. You must manually specify a machining coordinate system before creating Operations. Use the Hole Wizard to build processes then manually specify the CS to be machined. Once the CS is specified, have the Hole Wizard build the operations.

Hole Manager

The Hole Manager allows you to identify holes on a model or specify geometry to use as a hole. Once loaded, hole data can be grouped by type, size, or alignment; attributes can be sorted by specifications such as Type, CS, TPI or Pitch; machining and operations can be created using AutoWiz or Hole Wiz. Because Hole Manager retains data related to holes and allows for batch edits of hole specifications, it is especially powerful when dealing with parts containing a large number of holes.

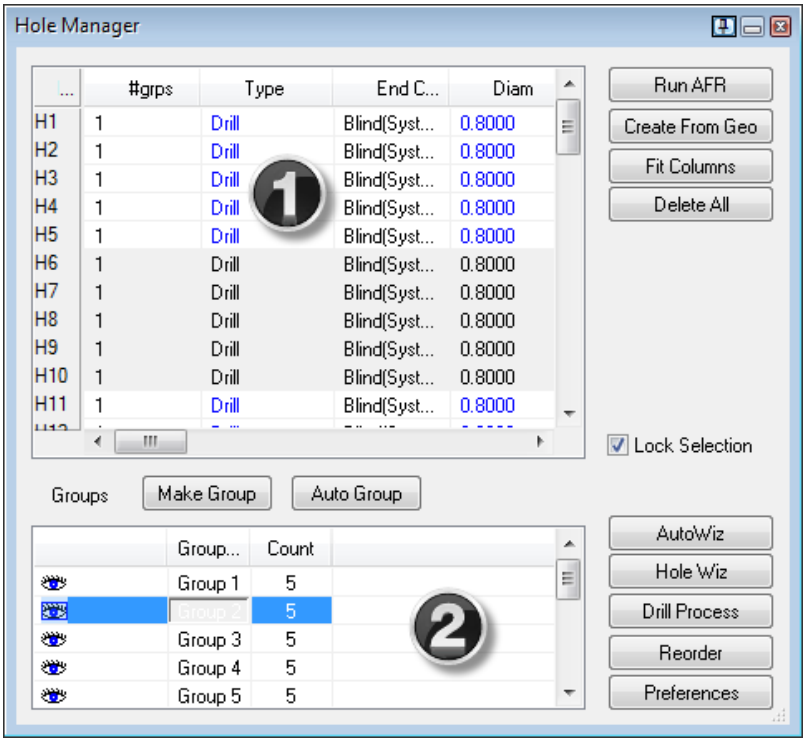
Hole Manager Dialog

The Hole Manager dialog contains two areas:

- Hole List
- Group List

The lists can be expanded by Left-mouse-button dragging the margins of the dialog. Drag the top margin and the Hole list expands. Drag the bottom margin and the Group list expands.

To the right of the lists are buttons for controlling the Hole Manager. You can access additional commands through the `right-click` context menus.



1. Hole List
2. Group List

Hole List

The Hole List is a complete list of all hole features in the current workgroup. Entries include the holes diameter, top and bottom depths, any chamfer data, tip angles, and so forth. You can view and edit single or multiple entries. You can also sort entries by any parameter, such as size and type of hole. For more information, see "Feature List" on page 24.

Group List

The Group List is a list of collections of holes that you have created. You can create groups automatically based on hole parameters or you can manually make a group based on your own criteria. Groups usually consist of holes that lie on the same plane or are of the same size and type. A group should contain all holes that are to be machined together in single or multiple drill processes. For more information, see "Group List" on page 29.

Using the Hole Manager

You use the Hole Manager to perform Automatic Feature Recognition, list features, group feature, start the Auto Wizard and start the Hole Wizard.
To use the Hole Manager:

1. Choose Features >  Hole Manager. The Hole Manager dialog appears.
2. To set Auto Wizard Preferences, click Preferences.
3. Select a the faces that make up one or more holes. You can select a complete solid, or multiple solids.
4. You can set Auto Wizard Preferences to further streamline Hole Wizard operation and adjust the amount of Hole Wizard automation.
5. Click the Run AFR button and set the options.

   The Hole Manager will create hole features based on the selected faces and AFR options.
6. Choose how to define coordinate systems associated with the holes and any default values you want to apply, such as clearance and bottom level adjustments.
7. Group the holes.

   You can group holes manually or automatically. To group holes manually, choose one or more holes in the Feature list and click Make Group. You can select multiple holes by holding down the Control key when selecting holes or select a range of rows by holding down the Shift key. To group holes automatically, click Auto Group. The Grouping Parameters dialog opens, you can set how you want to group the holes.
8. Select a group and click either the Hole Wiz, Drill Process or AutoWiz button. Choosing Hole Wiz opens the Hole Wizard where you set the parameters and complete the process. The Drill

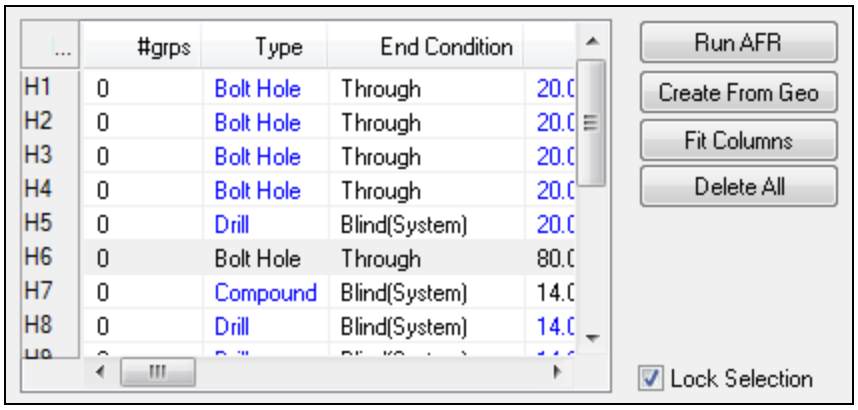
Process option creates a Holes process with the currently selected tool and point. You set the process parameters and click Do It. The AutoWiz option generates tools and hole data for each hole selected in a group.

Hole Manager Interface Items

This section describes the Hole Manager interface, including the “Feature List” on page 24, Hole List Context Menu, “Group List” on page 29, “Group List Context Menu ” on page 35, and “Hole Manager Preferences” on page 36.

Feature List

The Hole list is a table of all hole features, in the current WG, loaded into the Hole Manager. Each item type, except for “Compound,” corresponds to a Hole Wizard hole type. The values are not “process” values (that is, data that defines how the hole is machined) but are sets of information that define the hole. Processes can use some of this information, but may not. You can edit blue text entries. Black entries are not editable, but may update as you edit other data. For example, the chamfer diameter changes as the hole diameter and chamfer width change. Gray entries are not editable and do not change. Often, the gray items do not apply to the hole shape.



Click the column title to sort the items by column.

To highlight a hole in the Feature List on the corresponding drawing:

a. Ensure Lock Selection is selected (Lock Selection matches the dialog selection with its corresponding screen item.)

b. In the Hole List, click the hole entry. The faces that define the hole and the point created for the hole highlight.

Right-click anywhere in the Feature List to bring up the Hole List Context Menu.

Hole List Buttons

Run AFR
Click the Run AFR button to create Hole Manager data for the selected solid faces. A hole feature will not be created unless you select every face associated with a hole. See Using the Hole Manager. Clicking the Run AFR button opens AFR Options and AFR / Import Hole Data .

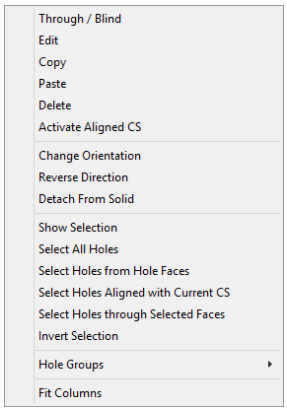
Create From Geo
Click this button to create Feature List entries from points and circles selected in the workspace.

Fit Columns
Adjusts the widths of all columns in the list so that all data is visible within a cell. The titles of a cell may or may not be cut off depending on the “Column Fit” preference setting.

Delete All
Deletes all hole feature definitions from the current WG and removes all entries from the Hole Manager List.

Lock Selection
Highlights the Feature List entry or entries for any currently selected geometry in the workspace.

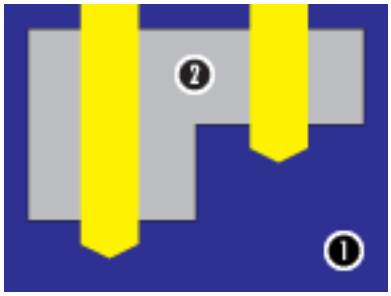
Hole List Context Menu



Right-click anywhere in the Hole list to display a context menu.

Through/Blind:
Through/Blind toggles whether the hole is a Through Hole or a Blind hole. The model may show a hole as a through hole. However, the stock condition may make the hole a blind hole. By toggling the type of hole, you modify the machining parameters.

Using the following image as an example, the Hole Manager recognizes the two holes as through holes, but the stock condition shows there is material above and below the holes. By toggling the holes to blind holes and adding top and bottom adjust values, you convert the hole as well as surface clearance and final Z depths.



1. Stock Condition
2. Model

Edit: (multiple holes)

If more than one hole is selected for edit, the Edit function displays the Edit Hole Parameters dialog allowing you to modify the features of multiple selected holes. Items that are not identical display with an asterisk (*). Anything you enter in these items overrides existing data. For more information, see "Edit Hole Parameters Dialog" on page 40.

Edit: (single hole)

This enables a single hole to be edited and visualized and/or a compound hole to be created. You can use the Show button to select and highlight the hole in the Hole Manager and 3D viewer.

The End Condition can be defined from the dropdown menu, either Blind, Through, Through Obstructed, or Spherical.

Segment Data

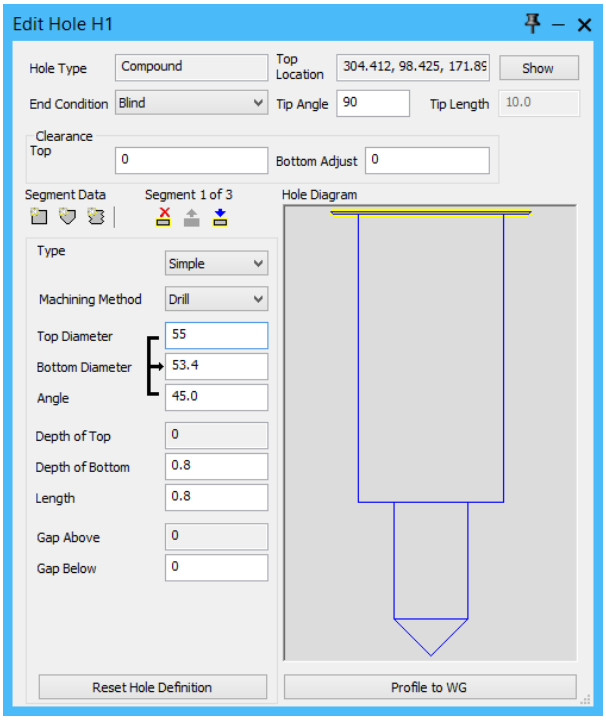
1. Straight (Square angle)
2. Chamfer (Taper angle)
3. Threaded

Use Segment Data to add any of the three types of segment to the hole. Click and drag a segment icon and drop it into position.

Click the Delete/Scroll up/down buttons  or mouse-click to highlight (in yellow) a segment to be edited.

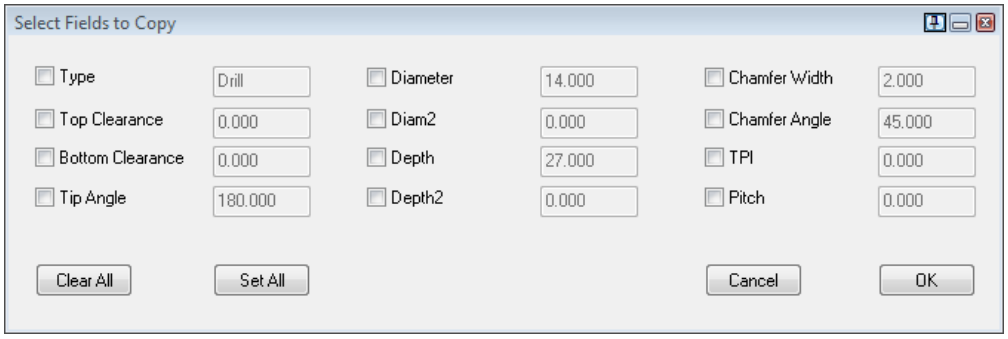
The segment Type can be set as either Simple or Threaded. Use the parameters to specify the dimensions. Gap parameters can be defined if there is a gap in the material.

Machining Approach can be set to either Drill, Ream or Bore/Mill.



Copy:

To copy fields from a specific feature to other features, select the feature you want to copy from and then click the Copy button. Select the items and click OK.



Paste:

Clicking this item modifies any currently selected holes. You can paste more than one entry simultaneously. Data copied from another hole is applied, including the type of hole and any attributes such as the depth, width or chamfer values.

Delete

This deletes all the hole data from selection.

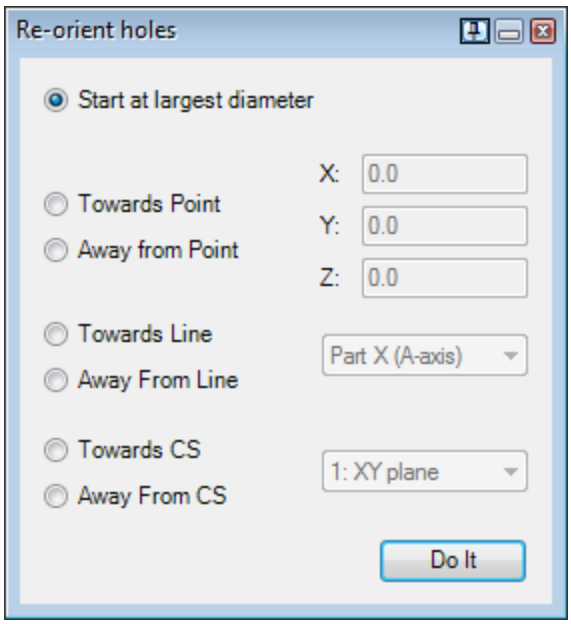
Activate Aligned CS

Activates the CS associated with the highlighted hole. If there is no CS, one is created.

Change Orientation

The End of the selected hole(s) will be oriented Towards or Away from the Point, Line or CS specified. Start at largest diameter, when used with a Combination hole (for instance a counter bored

hole) will orient the hole starting from the largest diameter to the smallest.



Reverse Direction:
This command only applies to through holes which may be machined from either end and toggles the direction.

Detach from Solid
Select a hole feature then click Detach from Solid. This disassociates the hole feature from the associated solid and attaches it to a newly created geometry point.

Show Selection:
Show Selection highlights in the model the features you select in the Hole Manager dialog.

Select All Holes
This highlights all holes in the current Work Group

Select Holes from Hole Faces
In the case where you have Lock Selection toggled off, this allows you to highlight a hole from a selected face.

Select Holes aligned with CS
This highlights all holes aligned with the current CS

Select Holes through Selected Faces
This highlights all holes that share an edge with the currently selected face

Invert Selection:
Invert Selection is useful for selecting multiple entries. Choosing this command deselects currently selected entries and selects entries that are not highlighted.

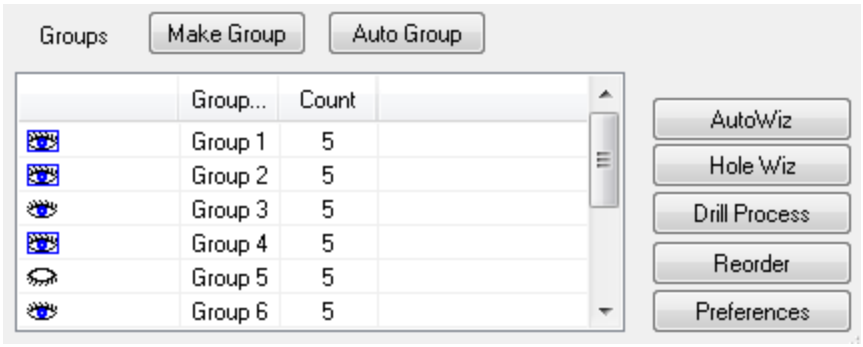
Hole Groups
Lists all the Hole Groups and indicates with a checkmark which group(s) the selected hole(s) appear in. You can use the checkmark to add or remove the selected hole(s) from groups.

Fit Columns:
This command adjusts the widths of all columns in the list so that all data is visible within a cell. The titles of a cell may be truncated depending on the “Column Fit” preference setting.

Group List

The Group List is a list of collections of holes. Entries are grouped by common attributes such as drill diameter. You can group holes manually by selecting holes in the Hole list and using the “Make Group” button, or automatically using the “Auto Group” button. The list shows the group name and the number of holes in the group. To change the name of the group, double-click the entry in the Group Name column.

Double-clicking on a group name highlights its contents in the Feature list and in the workspace. You can use the eye symbol to switch the workspace display of groups of holes on and off. An open eye means the holes are displayed with an orientation line and crosshatch. Close the eye and only the geometry is shown. You can also use the box selection (the square around the eye) to show/hide multiple groups at the same time.



Make Group Button

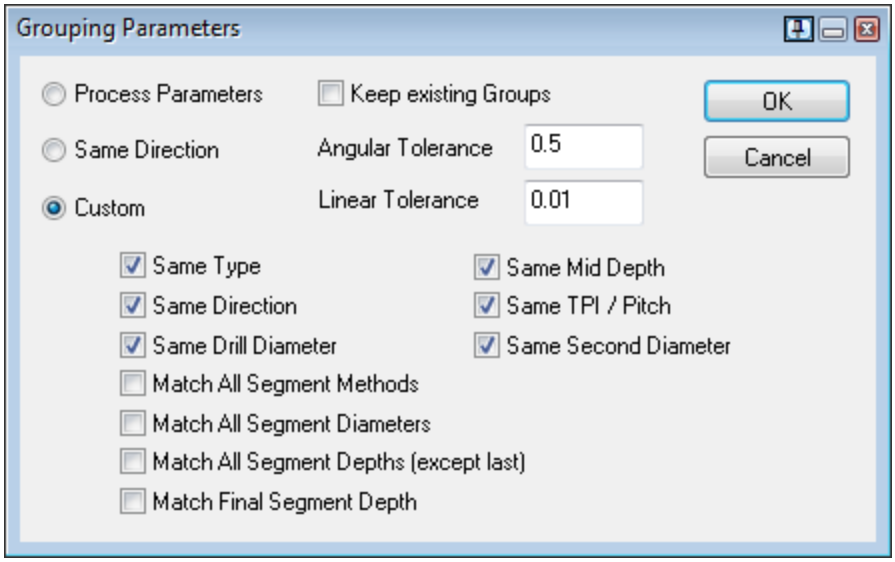
Click this button to group the currently selected Hole Manager List entries.

Auto Group Button

This option automatically groups the holes in the Hole Manager List. The holes are grouped according to parameters which you can specify. The Grouping Parameters dialog is available by clicking the Preferences button.

Grouping Parameters

The Grouping Parameters dialog provides the following options, if Use Dialog is selected in the Preferences menu.)



Process Parameters:
This option groups features by drilling process. Holes are grouped by hole direction, diameter, type, and bottom angle and depth.

Same Direction:
This option groups features that lie in the same hole direction.

Items that are within the Angular tolerance specified are placed in the same group. This tolerance refers to hole direction. Items beyond the tolerance are in separate groups.

Items that are within the Linear Tolerance of each other are placed in the same group. This tolerance refers to diameter, and depth. Items beyond the tolerance are in separate groups.

Custom:
This option groups features based on a custom combination of attributes. Selecting all items is effectively the same as selecting “Process Parameters”.

Same Type:
The grouped holes are of the same type of hole (Drill, Compound, Bolt, and so forth).

Same Direction:
The grouped holes lie in the same direction.

Same Drill Diameter:
The grouped holes are drilled using the same size drill.

Same Mid Depth:
The grouped holes all have a common Mid depth.

Same TPI/Pitch:
The grouped holes all have the same TPI/Pitch.

Same Second Diameter:
The grouped holes will have the same second diameter value.

Holes with the same number of segments as the Reference Hole can be grouped as follows:

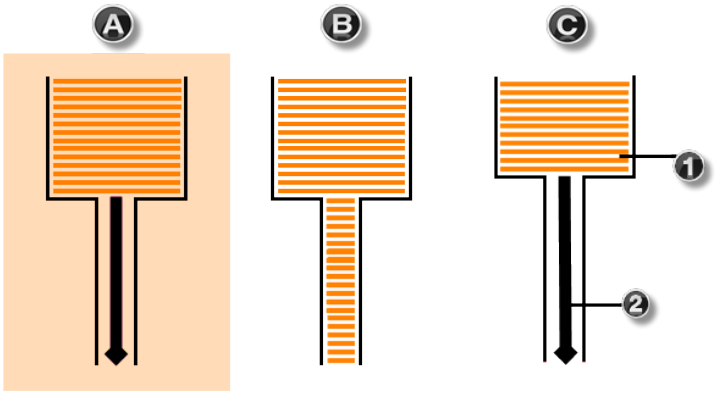
Match All Segment Methods
The grouped holes have corresponding segment numbers with identical machining methods.

Match All Segment Diameters
The grouped holes have the same diameters in corresponding segment numbers.

Match All Segment Depths (Except last)
The grouped holes have identical corresponding segment depths apart from the bottom segment.

Match Final Segment Depth
The grouped holes have the same depth of bottom segment.

The diagram below illustrates how segment matching will select holes. Hole A in this example is the Reference Hole.



1. - Milled segment
2. - Drilled segment

Matching Method	Result
Match by Machining Method	A C
Match by Diameter	A B C
Match By Length	A B
Match Final Segment Depth	A B
Combine Match Machining Method and Match Length	A

Choose a different Reference hole and the results may vary.

Keep Existing Groups:
Selecting this option will not modify or overwrite existing groups.

AutoWiz Button

Click this button to start the Auto Wizard. This will create operations for all selected groups using the data in the Hole Table.

Hole Wizard Button
Select a group and click this button to activate the Hole Wizard. You use the Hole Wizard to create processes and operations on the holes in the group. For specific details, see the Hole Wizard Basics section.

Drill Process Button

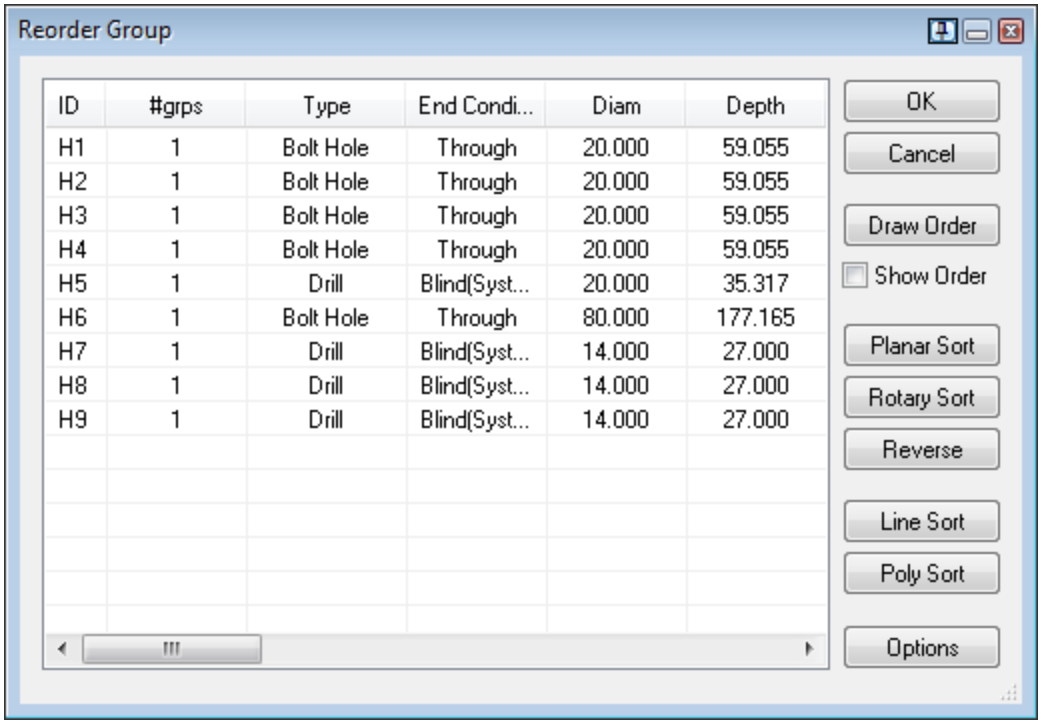
Select a group and a drill then click this button to create a Holes process. The clearances, surface Z and cut depth are automatically set, based on the Hole Manager data. The Z position is calculated to be the tip location for the selected tool on blind holes, and as the shoulder or “through depth” for through holes. Additionally, the geometry to be machined is selected. You must select the drill to use for the drill process. If you select multiple tools, the Hole Manager creates multiple processes in a group, 1 process for each tool selected.

Reorder Button

This item opens the Reorder Group dialog that allows you to reorder or sort the features in a selected group.

Reorder Group

The Reorder Group dialog provides the following options.

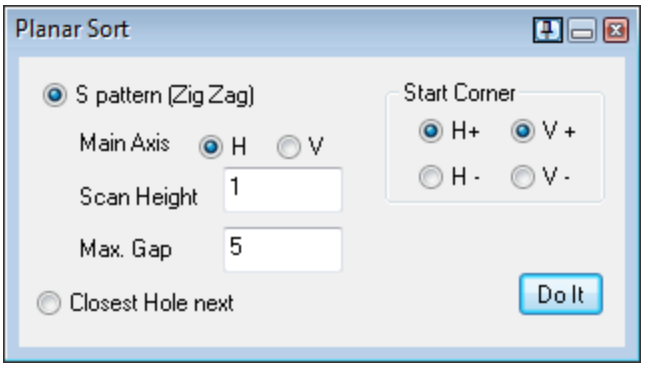


Reorder Group List:
A list of the features in the current group. These items may be resorted or you may click and drag an entry to a new position in the list, to reorder the group.

Draw Order:
This item displays the machining order of the features in the current group. A number that corresponds to the order displays above the feature.

Show Order
This checkbox will turn on/off the machining order display.

Planar Sort
Creates a new machining order of the current group by planar sorting of the holes.



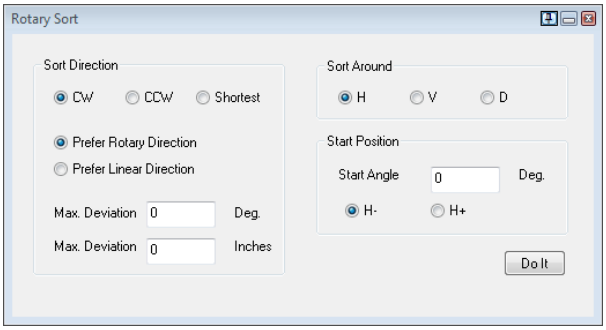
S Pattern (Zig Zag)
This option sorts the holes in an S-shaped or back-and-forth pattern. Main Axis specifies that the holes are sorted so that the primary motion of the tool is back and forth along the H (horizontal) or V (vertical) axis. Scan Height is the height of the area for each pass in which the system sorts the holes. Max Gap sorts the holes into smaller groups. Spaces between holes greater than the Max Gap creates a grouping of holes. After the first group is done, the holes beyond the Max Gap value are sorted. The image above shows holes sorted using S Pattern. The Main Axis is the H axis and the holes are divided into two groups because The distance between the two sets is greater than the Max Gap.

Closest Hole Next
The holes are sorted by proximity, starting with the point closest to the corner you designate as the Start corner. The second hole will be the closest hole to the starting hole.

Start Corner
H+ (right side of the coordinate system) H- (left side of the coordinate system)

V+ (top of the coordinate system) V- (bottom of the coordinate system)

Rotary Sort
Creates a new machining order of the current group by rotary sorting of the holes.



Sort Direction
Choose either Clockwise, Counter-clockwise or Shortest, which looks for the closest hole in either direction. You can choose to limit travel around the rotary or linear axis by using the Prefer Rotary/Linear Direction radio buttons. Two holes separated by the Max Deviation angle or distance specified will be treated as the same angular/linear position for sorting purposes.

Sort Around
Specify if you wish to sort around the H (Horizontal) V (Vertical) or D (Depth) axis of the current CS.

Start Position
The hole closest to the angle specified will be the start hole, beginning at either the H+ (largest) or H- (smallest) position along the sorting axis (HVD).

Reverse:
This reverses the order of the group.

Line Sort Button:
This performs a sort between two points. The points in the current group will be sorted according to their position between two selected points. The selection will travel along an axis between the points, selecting the points to either side of the axis line.

Poly Sort Button:
This performs a sort following connected features. The points in the current group will be sorted according to their position along geometry. You can create mouse lines to rough out the sort order, then select the line and click this button.

Options:
Set the AFR display preferences for the Hole Manager data.

Symbol
Select the size at which you would like the hole markers displayed. Selecting “None” turns off the hole markers.

Label
Select the size at which you would like the hole order numbers displayed. Selecting “none” hides the machining order numbers.

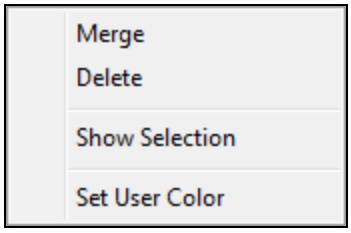
Connect Lines
Select the size at which you would like connecting lines between the hole markers displayed. This follows the machining order. Selecting “No” turns off the display of connecting lines.

Delay (ms)
This option is used with the “Show Order” button. The speed of the displayed cutting order is controlled using this option. The smaller the number, the faster the redraw. With a fairly large number (250-500) you can clearly see the order of the drilled holes.

Preferences Button

Click this button to display the “Hole Manager Preferences” on page 36. You can set the behavior of the Hole Manager.

Group List Context Menu



Right-click anywhere in the Group List to open a context menu that contains the following items.

Merge:
Selecting this item unites two or more selected groups into a single group.

Delete:
Selecting this item deletes any selected groups.

Show Sel:
Selecting this item highlights the faces and points of all holes in the currently selected group or groups.

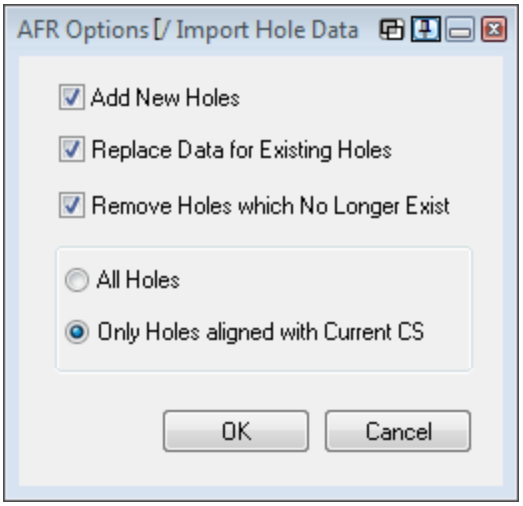
Set User Color:
Selecting this item displays the Color dialog where you can assign a color to the selected group or groups.

AFR Options and AFR / Import Hole Data

When you run AFR to automatically recognize hole features and import them, you might want to be prompted to review or change the following settings.

- If you want to be prompted to review these settings each time you run AFR, you can set your preference in the **Hole Manager Preferences** dialog, AFR / Import Dialog section: Select Use Dialog.
- If you want to re-use the same settings each time you run AFR, you can set your preference in the **Hole Manager Preferences** dialog, AFR / Import Dialog section: Select Do Not Use Dialog and then click the Set Dialog Data button to register and save your settings.

When importing, hole data from the imported model will be added as new, regardless of the selected import options. None of the existing hole data will be modified.



Add New Holes:

If this checkbox is selected, then new hole data will be added to any pre-existing data when you run AFR.

If it is not selected, then data for new holes will be ignored when you run AFR.

Replace Data for Existing Holes:

If this checkbox is selected, then hole data for existing holes will be replaced when you run AFR.

If it is not selected, then hole data for existing holes will be preserved unchanged, despite running AFR.

Remove Holes That No Longer Exist:

Please note - This option works only on geometry in the current Workgroup. Holes in the body bag are also unaffected. If this checkbox is selected, then existing hole data is deleted when you run AFR and no matching hole data is found in the selected solid. When All Holes is chosen (see below), then hole data will be discarded if it fails to match *any* of the holes in the selected solid. But if, instead, Only Holes aligned with Current CS is chosen, then the only hole data that will be discarded will be hole data that mismatches all holes normal to the current CS.

If it is not selected, then hole data will be preserved, even if it references holes that have been deleted, when you run AFR.

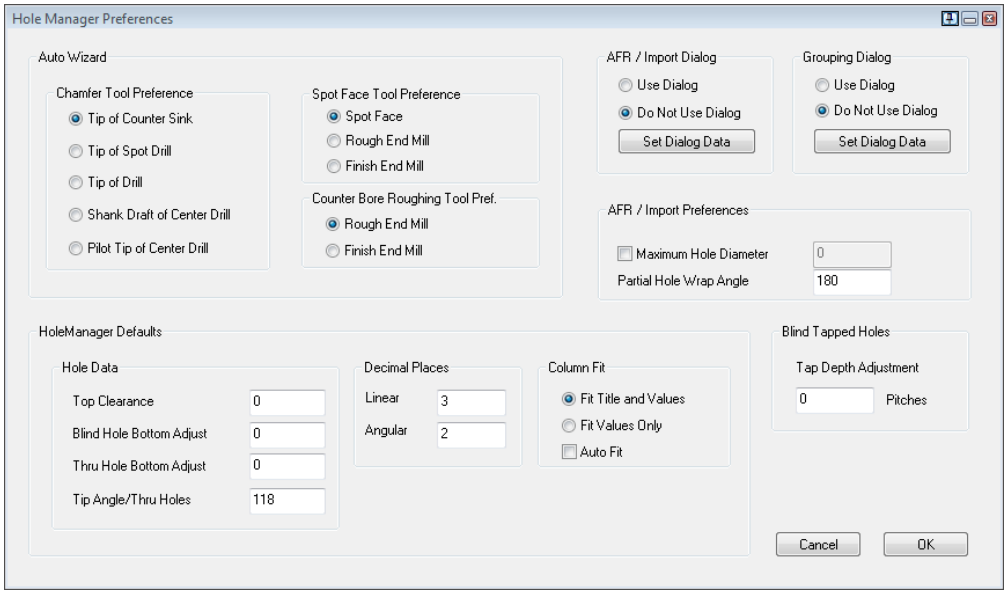
You also specify the scope of the selected operations (Add, Replace, and/or Remove):

- Choose All Holes to specify that all holes in the selected solid will be modified.
- Choose Only Holes aligned with Current CS to specify that the only holes that will be modified are those holes whose axis is perpendicular to the current CS.

Hole Manager Preferences

To display Hole Manager Preferences:

In the Hole Manager, click the Preferences button. The Hole Manager Preferences dialog appears. You can also access this dialog by selecting the File > Preferences > Machining Prefs >Hole Manager Preferences button.



The Hole Manager Preferences dialog displays the following options.

Auto Wizard

Chamfer Tool Preference

Tip of Counter Sink:
Use the tip of the counter sink to make the chamfer.

Tip of Spot Drill:
Use the tip of the spot drill to make the chamfer.

Tip of Drill:
Use the tip of the drill to make the chamfer.

Shank Draft of Center Drill:
Use the shank draft of the center drill to make the chamfer.

Pilot Tip of Center Drill:
Use the pilot tip of the center drill to make the chamfer.

Spot Face Tool Preference

Spot Face:
Use a spot face counterbore to make the spot face.

Rough End Mill:
Use a rough end mill to make the spot face.

Finish End Mill:
Use a finish end mill to make the spot face.

Counter Bore Roughing Tool Pref.

Rough End Mill:
Use a rough end mill to make the counter bore.

Finish End Mill:
Use a finish end mill to make the counter bore.

AFR/Import Dialog & Grouping Dialog

These both control the defaults for AFR/Grouping

Use Dialog
If this is selected, the dialog will prompt the user to enter the data each time.

Do Not Use Dialog
Program will use defaults

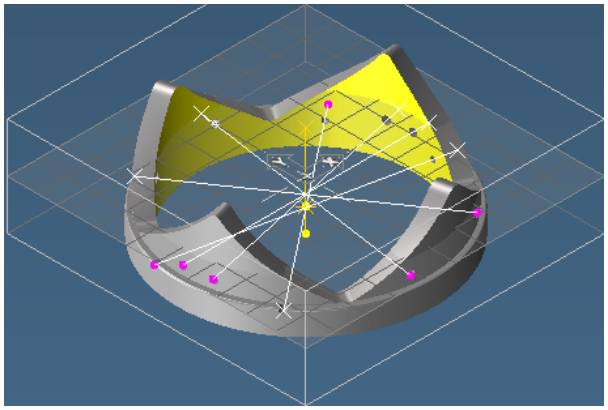
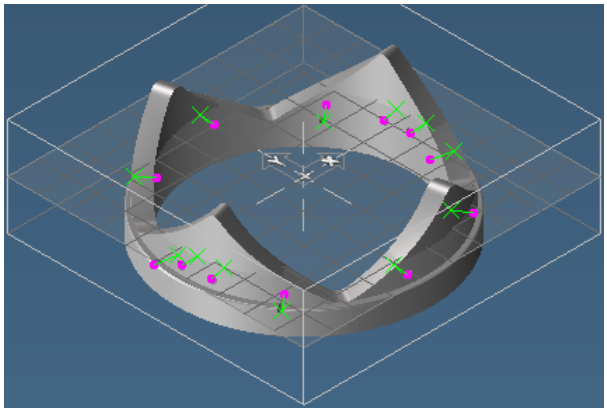
Set Dialog Data
This sets the defaults utilized when Do Not Use Dialog is selected.

For more information see AFR Options and AFR / Import Hole Data and Grouping Parameters

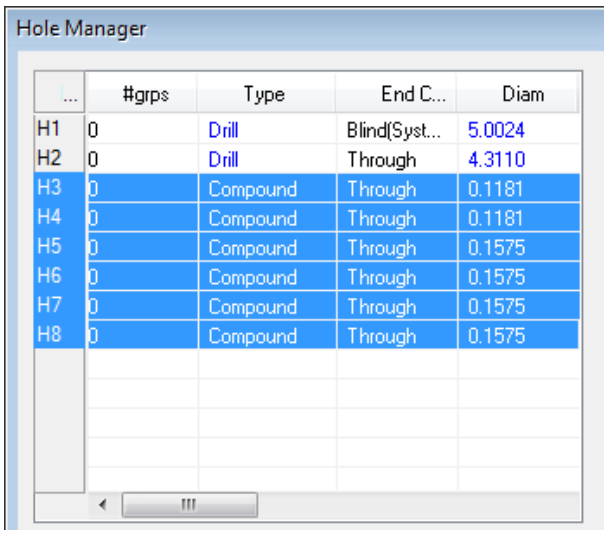
AFR/Import Preferences

Maximum Hole Diameter
You may set a maximum diameter for imported holes here.

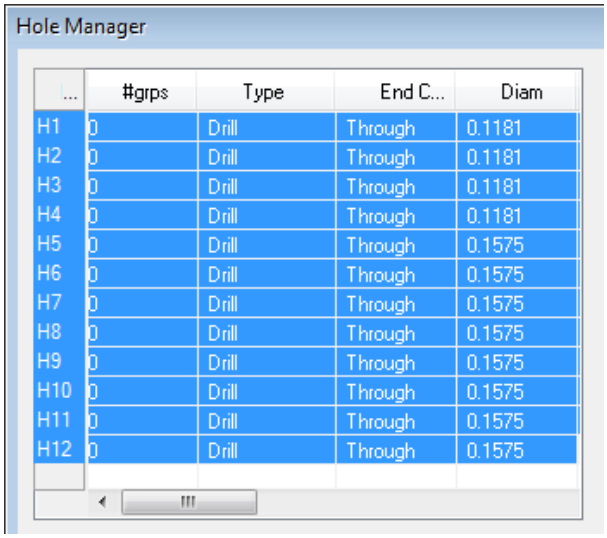
Please note that when importing a part with a large central hole with interrupted through holes as in the following illustration, Maximum hole diameter refers to the diameter of the largest hole, which in this case is the large central hole.

No Max Diameter set	Max Diameter
	

No Max Diameter set



Max Diameter



Partial Hole Wrap Angle
Set the partial hole minimum wrap angle to be created by AFR/Import.

Hole Manager Defaults

Hole Data

You can set default values for hole clearance and bottom level adjustment within Hole Manager for any holes, whether created by AFR, CAD import, or by Geometry creation.

Top Clearance:
Default clearance value applied (added) to the Part Top. Together these items calculate CP2.

Blind Hole Bottom Adjust:
Default adjustment for material that has not been removed yet. Essentially, this is a clearance value applied below the bottom of a modeled hole. This allows the drill to go deeper.

Thru Hole Bottom Adjust:
Default adjustment for material that has not been removed, yet. Essentially, this is a clearance value applied below the bottom of a hole to ensure a smooth hole all the way through.

Default Values - Tip Angle/Thru Holes:
Preferred value for the tip angle on tools that will be cutting thru holes. For blind holes, the system will attempt to use a tool with a tip angle that matches the hole. This will be used in situations where the tip angle is not vital.

Decimal Places

Decimal Places:
The Linear and Angular values in the Hole Manager will extend to this many decimal places.

Column Fit:

You can select the following options to determine how column widths are adjusted when you select Fit Columns.

Fit Title and Values:
When “Fit Columns” is selected, the columns will be adjusted to fit all data, including the column names.

Fit Values Only:
When “Fit Columns” is selected, the columns will be adjusted to fit only the hole data. Column names may or may not be fully displayed.

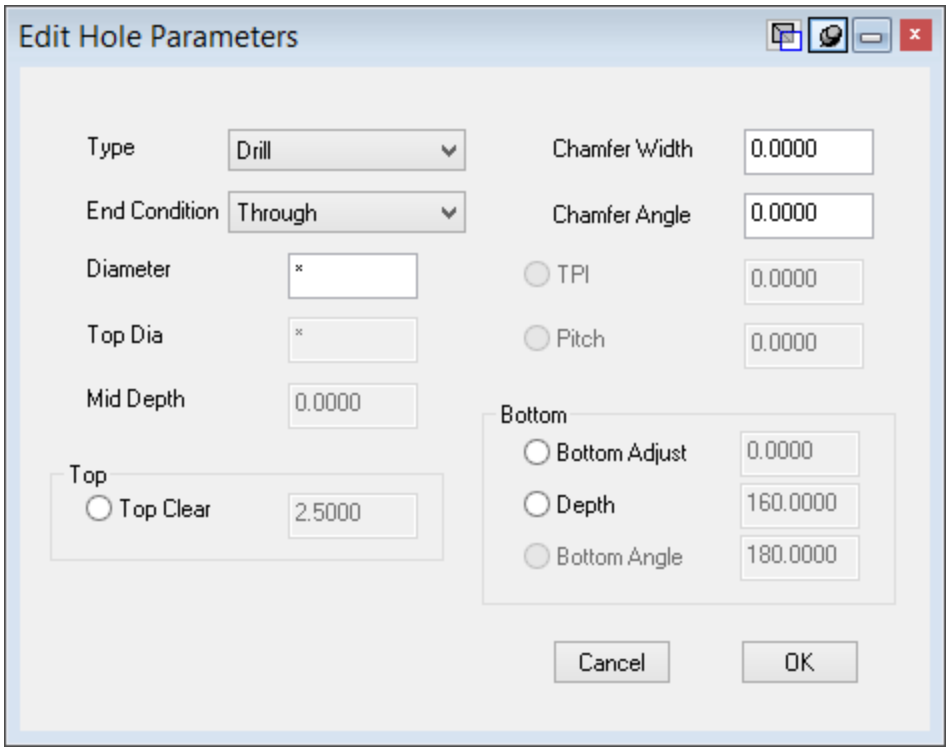
Auto Fit:
As data is loaded into the Hole Manager, the columns will be adjusted to fit the data.

Blind Tapped Holes

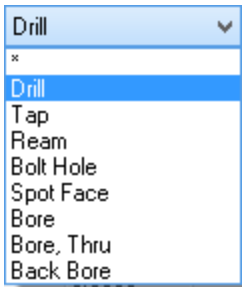
Tap Depth Adjustment
You may adjust the number of pitches for the (mid)depth tap for blind tapped holes. A positive value will tap deeper than the mid-depth, a negative value will tap shallower.

Edit Hole Parameters Dialog

The Edit Hole Parameters dialog displays options to define the hole shape, the top of the hole, and the bottom of the hole.



Type:
Select the type of hole for the feature.



Diameter:
Specify a primary Diameter for the selected entries.

Top Dia:
Specify a new Top Diameter for the selected entries.

Mid Depth:
Specify a new middle depth for the selected entries. The Mid Depth refers to a bore hole depth.

Chamfer Width:
Specify a new Chamfer Width value for the selected entries.

Chamfer Angle:
Specify a new chamfer angle for the selected entries. This angle is per side and is not the included angle.

Additionally for Tap holes you can specify TPI and Pitch.

Top

Top Clear:
Specify a new Top Clearance value for the selected entries.

Bottom

Bottom Adjust:
Specify a new Bottom Adjust value for the selected entries.

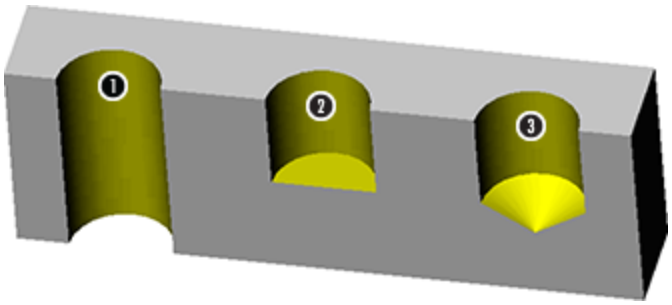
Depth:
Specify a new Depth value for the selected entries.

Bottom Angle:
Specify the tip angle of the selected blind hole entries.

Hole Manager Data

No matter how complex, hole shapes that the Hole Manager encounters can be broken down into three basic types, Through holes, Blind holes with flat bottoms and Blind holes with a tip. The tool to be used with each as well as the process parameters will vary based on the type of hole.

Through Hole
The Bottom Adjust value will be added to the depth. Through holes use the Z value of the shoulder (or the full diameter) for the output point position.



1. Through Hole
2. Blind Hole with Flat Bottom
3. Blind Hole with Correct Tip

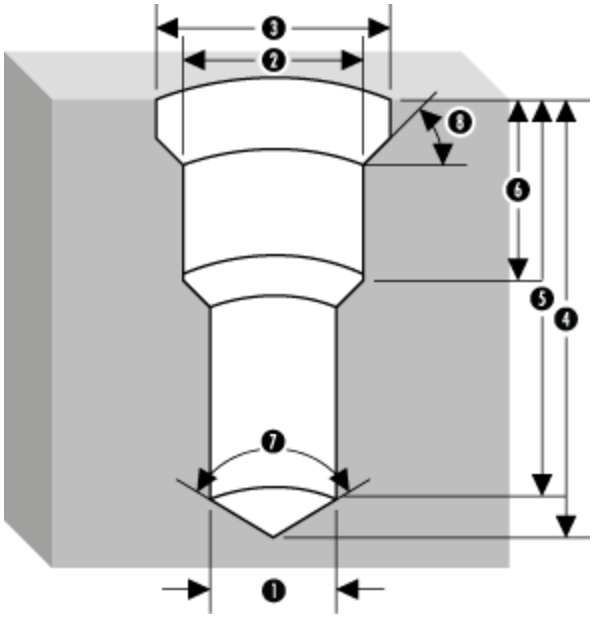
Blind Hole with Flat Bottom
The tool tip height will be added to the depth. The bottom angle will initially be set to 180°.

Blind Hole with Correct Tip
The correct tool bottom angle will be set. The tool tip height will be added to the depth. Blind holes use the Z value of the tip of the hole for the output point position.

Note that for a hole to be recognized by the Hole Manager's AFR, a hole should start and stop on the same level or, in other words, have its top and bottom edges exist on a single face. A hole may not be recognized by AFR if its top and bottom exists on different depth levels.

Hole Dimensions

The standard elements of a hole shape are defined in the accompanying image. The Hole Manager can fully describe compound holes using the Edit feature within the Feature List.



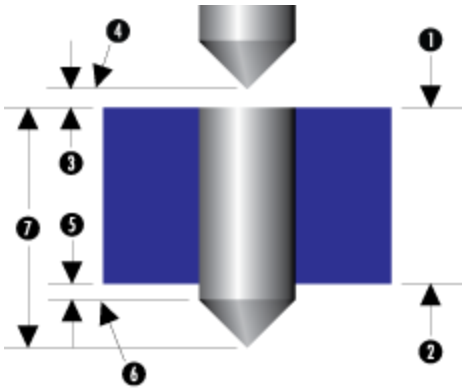
1. Diameter
2. Diameter 2
3. Chamfer Width
4. Depth
5. Full Depth
6. Depth 2
7. Tip Angle
8. Chamfer Angle

Hole Bottom

Depth of the hole from the model.

Mid Depth

This value may be the bottom of a tap or spot face or bored section of a hole. The clearance planes and cut depths entered in the Process dialogs are determined by the hole data. The variables are described below.



1. Part Top
2. Hole Bottom
3. Top Clear
4. CP2
5. Bottom Adjust
6. Bottom Z
7. Depth

CP2 = Part Top + Top Clear

Bottom Z = Hole Bottom + Bottom Clear

Depth = Part Top - Bottom Z - Tip Height

Tip Height = 0.5 * Diam / tan(0.5 * Tip Angle)

Tool Definition

Flute length = hole depth + tool diameter, rounded up to the next larger 1/4” or 1mm.

Total length = 2.0 * flute length, rounded up to the next larger 1/4” or 1mm.

Pre-Drill Depth

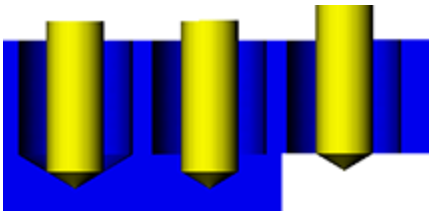
The drill depth for pre-drill operations depends upon the hole type.

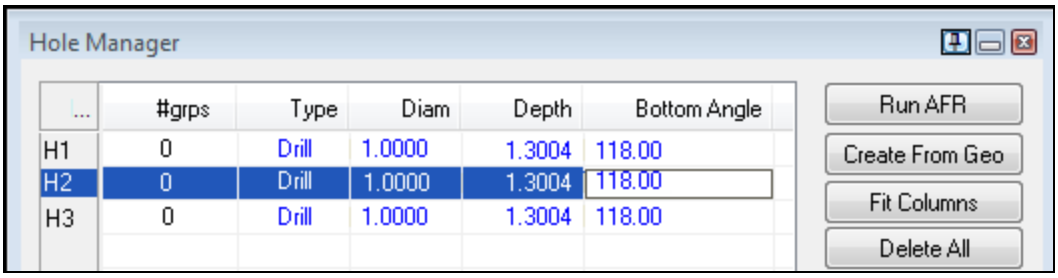
Through Holes

For Through holes, the shoulder of the pre-drill tool goes to the bottom of the through hole.

Blind Holes

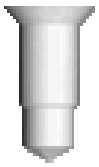
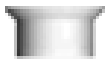
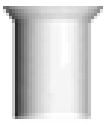
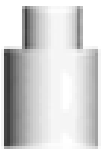
For Blind holes, the pre-drill tool tip will go to the tip of the bottom cone. In the image to the right, the drill goes past the bottom of a flat hole. For this hole the tip angle was manually set to 118° and the hole depth is recalculated. Any value that is gray is not editable, blue entries are editable and black entries may be modified as editable data changes.



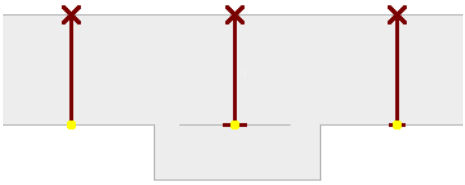


Hole Manager Colors & Symbols

The Hole Manager draws drill points (the final depth), an “X” at the top of the hole and a line leading to the point for each hole. For quick identification, the color of the line and “X” will vary depending on the hole type.

Drill	Tap	Ream	Bolt Hole	Spot Face	Bore	Bore, Thru	Back Bore	Com pound
								
Green	Blue	Yellow	Magenta	Cyan	Brown	Dark Green	Dark Blue	White
								

Additionally, the Hole manager will identify the hole shape by drawing a line at the bottom of Blind holes. Through holes only have the bottom depth marker point. If you convert a through hole to a blind hole then the Hole Manager will draw a short line at the bottom of the hole.



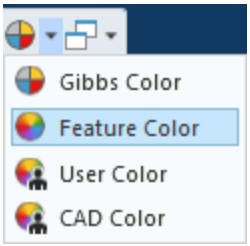
User-Defined Attributes, Features, and Color

You can define attributes, features, and colors for parts.

For more information, see the following topics:

- Color Display Modes
- About Attributes
- Attribute Manager
- About Features
- Feature Manager
- About User Colors and Color Palettes

Color Display Modes



On the Floating Task Bar, the Select Color Display Mode button indicates which color display mode is currently active and lets you select or toggle through other modes, if available.

Color modes other than the default (Gibbs Color) only affect elements that have had color applied to them. Elements in a selection set are displayed in yellow; faces affected by pre-selection highlighting are displayed in a blend between the element color and the pre-selection color.

Gibbs Color mode. *Default.*
Always available. In this mode, the color of workspace elements is governed by the values set under Preferences. When you select non-geometry elements, the selection color displays according to Gibbs Color mode.

To view or change default values for Gibbs Color mode: On the File menu, click Preferences; then, in the **Preferences** dialog, Display tab, click the Edit Appearance Settings button. Finally, in the **Appearance Settings** dialog: Select a configuration from the list, click Duplicate (or, if you can tolerate being unable to recover old settings, simply clear the Protect checkbox), and click the Colors tab.

Feature Color mode.
Available only when at least one element has an attribute of type Feature.

In this mode, the color of workspace elements is governed by the features they belong to. Workspace elements associated with two or more features display the color of the most recently created feature; elements not associated with any feature display according to Gibbs Color mode.

 **User Color** and other user color modes, (eg. CAD color as shown).
Available only when at least one element has an attribute of type `Color`. A part can include many such attributes, including both the reserved system attribute named **User Color** and also user-created or imported color attributes.

In User Color mode (or MyColor mode), the color of workspace elements is governed by the palette of the User Color (or MyColor) attribute attached to it. Any workspace element without the corresponding attribute is displayed according to Gibbs Color mode.

Note: Icons in the Body Bag are unaffected by color display mode.

About Attributes

Attributes are data that can be attached to elements. Any element (such as an edge, a face, or an entire solid body) can have zero, one, or hundreds of attributes attached to it. Any attribute can be attached to zero, one, or hundreds of different elements.

Every attribute has a name, a data type, and a default value. Data types are:

- **Integer** Examples: `6` or `0` or `-2`
- **Real** Examples: `6.283185` or `0.000000` or `-2.000000`
- **Text** Examples; "`Typical`" or "" (null) or even "`↑ ¿√►◄‼`"
- **Color** Examples: (`255,0,0;0%`) for opaque red or (`0,255,0;50%`) for translucent blue or (`0,0,255;100%`) for transparent green
- **Feature** associated with a particular RGB triplet and transparency percentage.

The reserved attribute named `User Color` cannot be renamed or deleted, unlike all other attributes.

Once an attribute is defined, its data type cannot be modified. Modifying an attribute's default value has no effect on elements already tagged with the attribute. (However, when the color/transparency of a Feature is changed, the change is immediately visible in Feature Color mode.)

Each attribute of type `Feature` has exactly one color assignment. If the default value of this color is changed, all elements tagged with the attribute are changed. Feature colors are visible in the Workspace only when `Feature Color` mode is active.

Each attribute of type `Color`, including the `User Color` attribute, allows you to define and use a palette of many colors. Each new color attribute is associated with a new user color mode, which is stored with the part. When a color attribute is attached to an element, its value (RGB and transparency) is initially set to the default but later can be changed independently for each element. User colors are visible in the Workspace only when the corresponding user color mode is active.

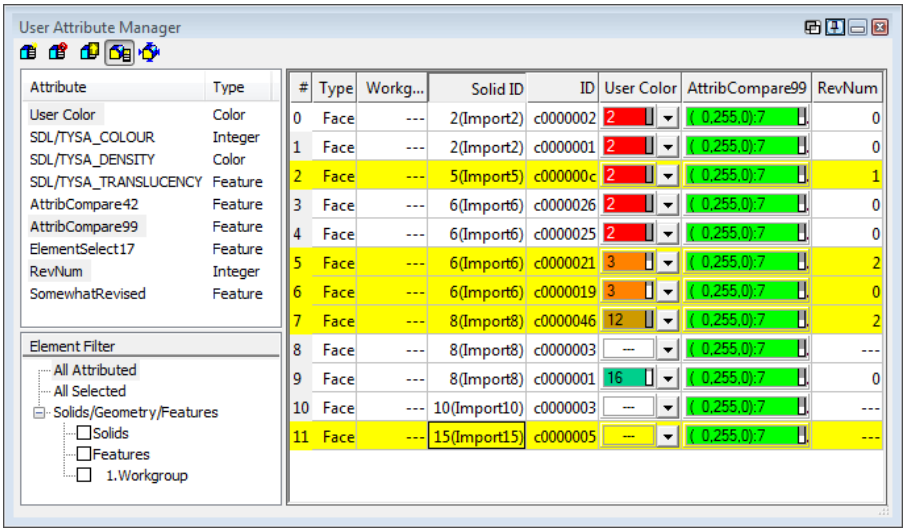
Please note: It is not advisable to mix processes where some contain "from attribute" or "from feature" and some do not. When GibbsCAM encounters this, processes set to "from attribute" or "from feature" are always machined first.

For information about creating and manipulating user attributes, see " Attribute Manager" on page 48.

Attribute Manager

To view, create, or modify user attributes in the current part, click **Features** > **Attribute Manager**. Attribute Manager presents attribute information in three panes:

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



As with most lists, you can `Shift-click` to select a range or `Ctrl-click` to select/deselect individual items, and you can sort by selecting a column heading.

Toolbar

The toolbar for Attribute Manager offers the following buttons:

Create New Attribute: Opens a dialog box that allows you to create an attribute and specify its name, data type, and default value.

Also available as a choice on the context menu (right-click menu).

Delete Selected Attributes: Deletes all attributes whose names are currently selected in the attribute list.

Also available as a choice on the context menu (right-click menu).

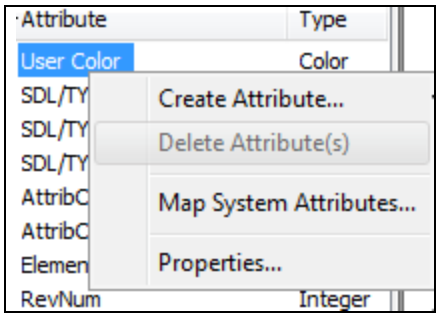
Add Selected Attributes to Elements: Attaches all currently selected attributes to all currently selected elements.

or Select Corresponding Elements mode: When this button is in the On mode ( ) and you select an attribute, the corresponding elements in the Workspace (or bodies in the Body Bag) are selected. When you deselect an attribute name, the corresponding elements in the Workspace or Body Bag are deselected.

or Zoom on Selection mode: When this button is in the On mode ( ), the Workspace display shrinks or enlarges to contain all elements selected in the element list.

Context Menu: Attribute List

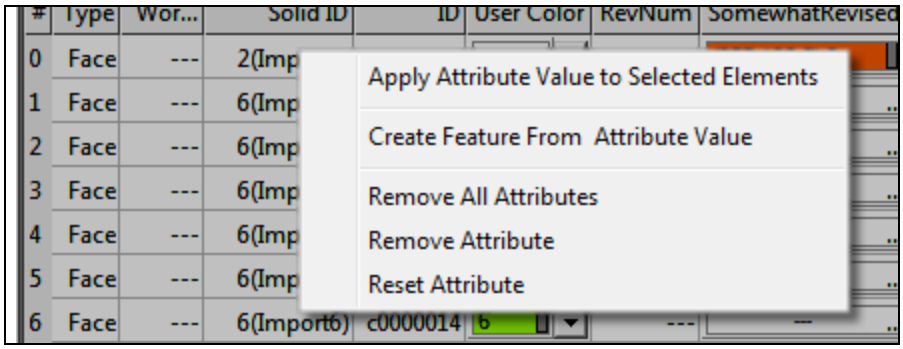
To display the context menu for a row in the attribute list, `right-click` the row.



- Create New Attribute and Delete Selected Attributes: Same as and in the toolbar, described above.
- Map System Attributes: See “Mapping System Attributes” on page 52.
- Properties: Has the same effect as double-clicking the attribute name. Opens the **Change Attribute** dialog, which allows you to modify the description or default value of the selected attribute.

Context Menu: Element List

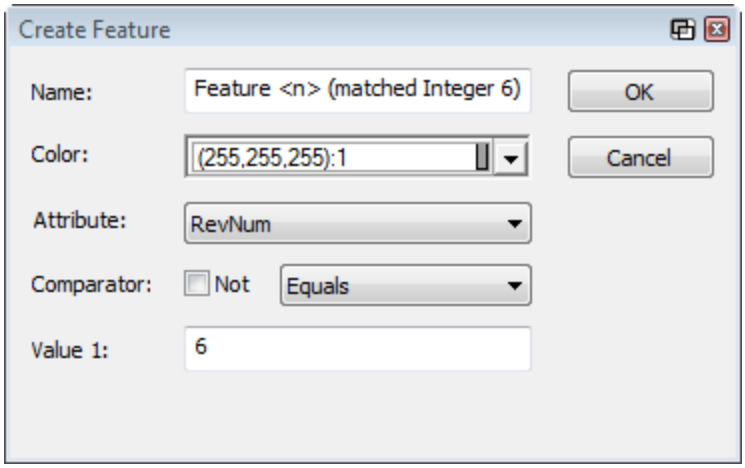
To display the context menu for an item in the element list, `right-click` one of its cells. The contents of the context menu depend on the type of cell you right-click.



- Apply Attribute Value to Selected Elements: Available only if one or more Workspace or Body Bag elements are selected and you right-click an attribute that has been set to a value. Similar effect as  in the Attribute Manager toolbar (described above), but attaches only the right-clicked attribute value to the selected elements.
- Create New Feature From Attribute Value: Operates only on selected rows; available only when you right-click a cell where an attribute has been set to a value. Has a similar effect as clicking  in the Feature Manager toolbar, but creates new features automatically, using the "Equals" comparator and the right-clicked attribute value or values.

Example:
Imagine that an Integer attribute named RevNum has been applied to one or more elements. If you highlight ten rows, right-click a cell in the RevNum column that has a value (let's imagine the column has exactly four values defined: 5, 3, 0, and 6), and choose Create Feature from Attribute Value, then four new Features will be created, named "**Feature <*n*> (matched Integer 5)**" and so forth.

The result will be the same as if you had used  Create Feature From Attribute, chosen the RevNum attribute, chosen the "Equals" comparator, and entered the value 5, and then repeated the process three more times with 3, 0, and 6),



After a feature is created, you can use Feature Manager to change its name, color, and Recreate flag.

- Remove All Attributes: Operates only on selected rows. Removes all attributes from the elements corresponding to the selected rows. For example, if you range-select rows 3 through 6 and then

right-click row 2 and choose Remove All Attributes, then all attributes are removed from elements corresponding to rows 3, 4, 5, and 6 (but not 2).

- Remove Attribute: Operates only on selected rows; available only when you right-click a cell where an attribute has been set to a value. Removes that attribute from the elements corresponding to the selected rows. For example, if you select rows 3, 4, and 8, and then right-click the User Color cell for row 9 and choose Remove Attribute, then the User Color attribute is removed from elements corresponding to rows 3, 4, and 8 (but not 9).
- Reset Attribute: Operates only on selected rows; available only when you right-click a cell where an attribute has been set to a value. Resets the attribute value (to the originally defined default value) on the elements corresponding to the selected rows.

  Example: Imagine you have defined a Text attribute named Vendor whose default value is "Internal". If you select rows 2 and 5, and then right-click the Vendor cell for row 1 and choose Reset Attribute, then theVendor attribute is reset to "Internal" for elements corresponding to rows 2 and 5 (but not 1).

About System Attribute Mapping

When you import CAD attributes, you normally retain the system attribute name and attribute type without changes. After a system attribute is mapped to a particular name and type, re-mapping affects the attribute's values only, not its name or type.

Note: To re-map to a different attribute name or attribute type, you must first delete the existing user attribute.

System Attribute Type	New Type	Description
Integer	Real or Text	Each integer value is retained and translated. For example, the integer 1 becomes real 1.0000 or text string "1.0000".
Integer	Color	The first 64 unique integer values are mapped to 64 different colors; the next 64 unique integer values are mapped to the same set of 64 colors; and so forth.
Integer	Feature	Each integer value is mapped to a new feature ID, but feature colors are reused in cycles of 64.
Real or Text	Integer	The real or text values become undefined.
Color	Integer	White is mapped to 1 and other colors are mapped to 0.
Color	Real (or Text)	White is mapped to 1.0000 (or "1.0000") and other colors are mapped to positive real values less than 1.0000.

Importing CAD Attributes

GibbsCAM preserves system attributes on CAD models that are imported from other systems. You can map CAD attributes to GibbsCAM user attributes:

- When you initially import a CAD model
- At any time using Attribute Manager

The attributes are automatically mapped by the system. To view them choose the Cad color option from the Color attributes dropdown on the floating toolbar.

To map or re-map system attributes of a previously imported CAD model

1. Click Features > Attribute Manager.

   The **User Attribute Manager** dialog appears.

2. Right-click in the attributes list and select Map System Attributes..

   The Define CAD Attribute Map dialog box appears.

3. Select one or more checkboxes for system attributes you want to map or re-map.

   Warning: For any system attribute that is already mapped, re-mapping it restores the previous system values, undoing any changed values. However, the attribute name and attribute type of the previous mapping are retained.

Mapping System Attributes

You can use attributes that were defined in models imported from other CAD/CAM systems, such as SOLIDWORKS:

When you import a model into GibbsCAM (using File > Import), options display so you can define the mapping from the existing attributes to GibbsCAM user-defined attributes. After the model is imported, you can view or reimport the mappings.

To view or reimport the mappings

1. Click Features > Attribute Manager) to open Attribute Manager
2. Right-click in the attribute list and select Map System Attributes.

   The **Define CAD Attribute Map** dialog lists all attributes found in the imported model.

3. To make a change, modify one or more of the following items for each attribute and then click OK.

   - For Import: Select the checkbox for each attribute you want to reimport.
   - For CAD Attribute Name: Shows the name of the attribute as set in the originating system.
   - For User Attribute Name: Specify the name of the attribute to be created in GibbsCAM.
   - For Type: Specify the data type of the attribute to be created in GibbsCAM.

About User Features

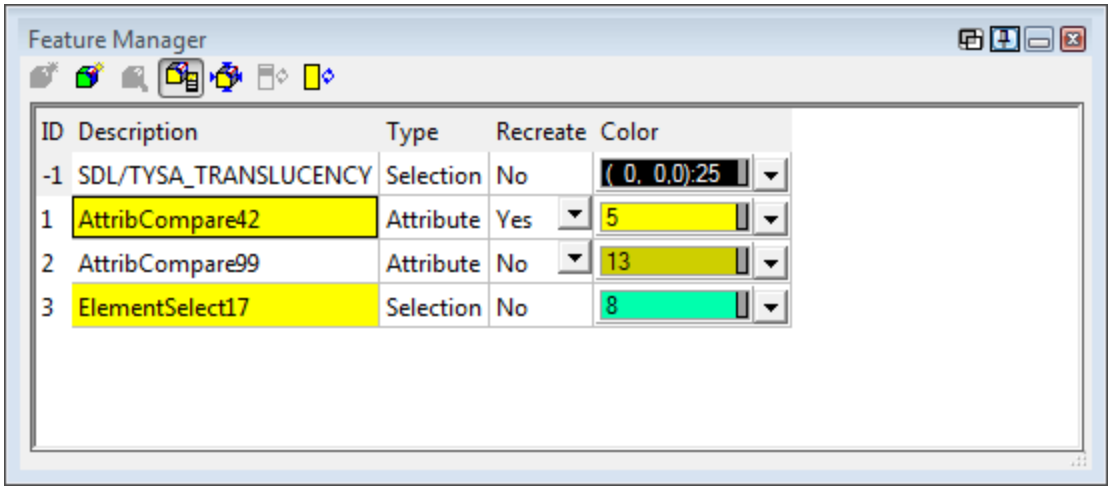
A user feature is collection of elements that are tagged with the same feature attribute. For example, you can select the faces of the slot in a part and tag them with a feature attribute named "MySlot1".

User features differ from system features such as Holes. For example, user features are not automatically associated with a machining process, and they are not detected by AFR (automatic feature recognition).

For step-by-step procedures for creating and manipulating user features, see "Working With Feature Manager" on page 55.

Feature Manager

To view, create, or modify user features in the current part, click Features > Feature Manager. All user features are listed in a table that displays their ID, Description, Type (element-selection-based or attribute-comparison-based), Recreate flag, and Color:



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

Toolbar

The toolbar for Feature Manager offers the following buttons:

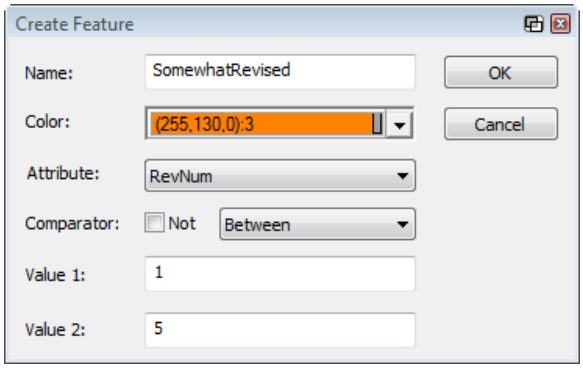
Create Feature From Selection: Available only when one or more elements are selected in the Workspace or Body Bag. Opens a dialog box that enables you to tag all elements in the current selection set as a new user feature.

Also available as a choice on the context menu (right-click menu).

Create Feature From Attribute: Opens a dialog box that enables you to create a new feature derived from a subset of values of a specified attribute.

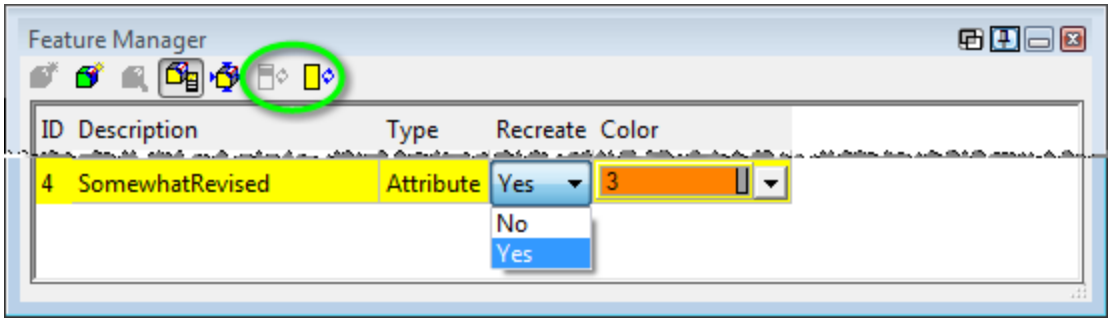
If a feature is initially created by means of an attribute comparison, you can choose whether it will be updated dynamically (Recreate=Yes, the default) or will remain unchanged (Recreate=No) when a Recreate Contents button is clicked.

Example:
Imagine a feature that was initially defined as the set of all elements whose RevNum attribute was between 1 and 5:



Now further imagine that the RevNum attribute has changed for some elements. For example, some elements originally at RevNum=4 are now at RevNum=5 or above (or have had their RevNum attribute removed), and some elements originally at RevNum=1 (or lacking a RevNum value) are now at RevNum=2 or 3 or 4:

- If you want the feature to be unaffected by Recreate Contents, then set its Recreate flag to No.
- If you want the feature to be reconstituted each time Recreate Contents is clicked, then keep its Recreate flag set to Yes.



Modify Feature With Selection: Redefines the feature to consist only of elements in the current selection set. If the feature was originally defined using attribute comparison, its Recreate flag is reset to No.

Also available as a choice on the context menu (right-click menu).

or **Select Corresponding Element** mode: When this button is in the On mode ( ) and you select a feature, the corresponding elements in the Workspace (or body in the Body Bag) are selected. When you deselect a feature name, the corresponding elements in the Workspace or Body Bag are deselected.

or **Zoom on Selection** mode: When this button is in the On mode ( ), the Workspace display resizes to contain all elements associated with the selected feature.

Recreate Contents of Selected Features: Affects only features whose `Recreate` flag is set to `Yes`. Updates the selected feature or features based on the current attribute values of all elements in the Workspace and Body Bag.

Recreate Contents of All Features: Updates all features whose `Recreate` flag is set to `Yes`, based on the current attribute values of all elements in the Workspace and Body Bag.

Working With Feature Manager

To create a new user feature from selected elements

1. In the Workspace and/or Body Bag, select one or more elements.
2. On the **Feature Manager** toolbar, click **Create Feature From Selection**.
3. In the **Create Feature** dialog, enter a name and specify a color for the feature.

Result: A new row is added to the feature table. The new feature is of type `Selection`, and its `Recreate` flag is permanently `No` – in other words, the feature can only be modified by adding or removing elements.

To create a new user feature based on attribute criteria

1. On the **Feature Manager** toolbar, click **Create Feature From Attribute**
2. In the **Create Feature** dialog:
   - Enter a name and specify a color for the feature.
   - Choose which attribute to base the feature on.
   - Specify comparison criteria – such as all elements bearing the specified attribute at a value less than a certain amount, or all elements bearing the specified attribute at any value.

Result: A new row is added to the feature table. The new feature is of type `Attribute`, and its `Recreate` flag is initially set to `Yes` – in other words, if attribute values change, the feature can be updated by clicking one of the **Recreate** buttons on the toolbar.

To add selected elements to one or more user feature

1. In the Workspace and/or Body Bag, select one or more elements.
2. Do one of the following:
   - To add the elements to one particular feature, right-click the feature name and, on the context menu, choose **Modify Feature with Selected Solids/Geometry**.
   - To add the elements to several features, `click` and `Ctrl-click` the features you want to select and then click toolbar button **Modify Feature with Selected Solids/Geometry**.

Result: The feature or features are updated to include the elements you selected.

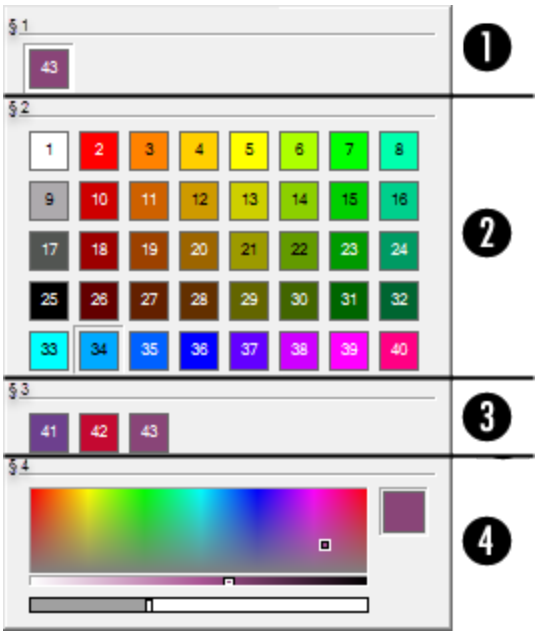
To recreate one or more user features based on current attribute values

1. In Feature Manager, examine the Recreate column:
   - If an attribute-comparison feature you want to update has its `Recreate` flag set to `No`, change it to `Yes`.
   - If you want to exclude feature from being recreated, ensure that its `Recreate` flag is set to `No`.
2. Do one of the following:
   - To update one feature, or only a subset of features, `click` and `Ctrl-click` the features you want to update and then toolbar button **Recreate Contents of Selected Features**.
   - To update *all* eligible attribute-comparison features, click toolbar button **Recreate Contents for All Features**.

Result: The feature or features are recreated; all elements that meet the defining criteria are added to the features, and all elements that fail the criteria are removed from the features.

About User Colors and Color Palettes

When Attribute Manager or Feature Manager prompts you to choose a color, you have several choices presented in a visual pull-down menu.

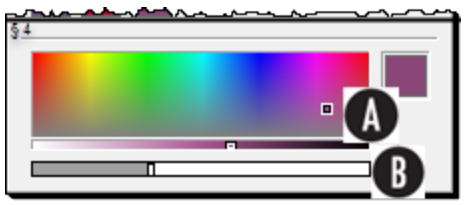


1. **Assigned Colors** section: If present, this provides a series of blocks showing you the subset of colors that have already been assigned.
2. **Preset Colors** section: This provides a series of blocks, numbered 1 to 40, showing you the colors that are defined within the current color palette.
3. **Custom Colors** section: If present, this provides a series of blocks, numbered 41 and above, showing you additional colors that have been defined outside the current color palette.
4. A **color picker** section with a spectrum and color controls. See Color Picker, below.

- Also, when you right-click any color block, a context menu offers you two additional choices. See Context Menus: Edit Color and Load Color Definition, below.

Color Picker

The bottom section of the visual pull-down menu provides a quick way to interrogate colors, and also a transparency control.



A When the pull-down menu first opens, its bottom section displays a spectrum of hue+saturation and a luminosity slider below the spectrum. Luminosity is the amount of white or black added to the hue. This area shows you how the current color compares to others. You can hover the cursor over the block on the section's right to interrogate the color's RGB triplet. You can also select elsewhere on the spectrum, or use the slider to audition variations in luminosity.

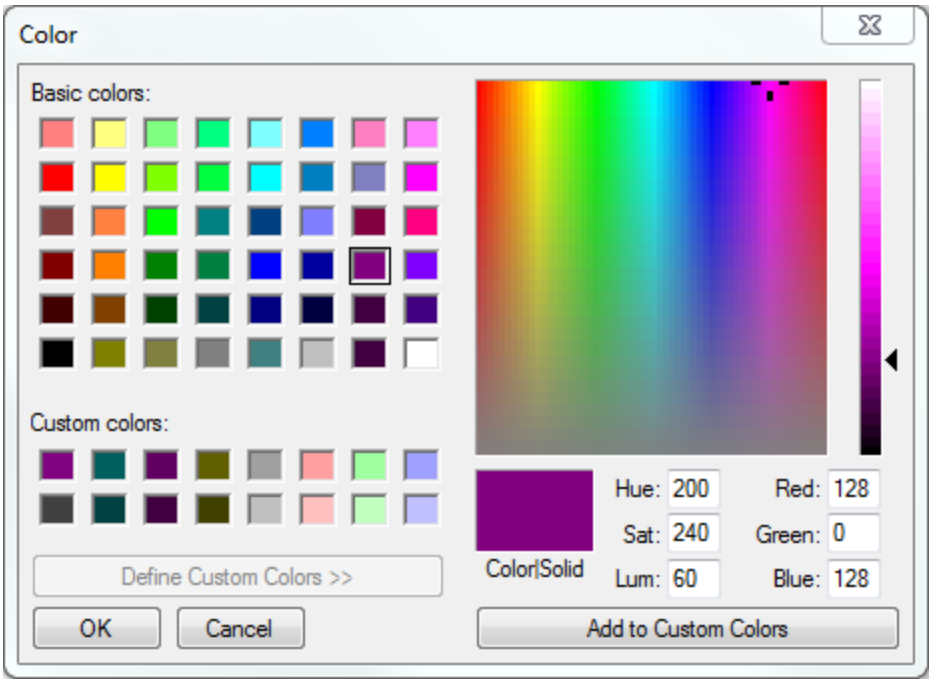
B The other slider lets you adjust the transparency of the current color. When the slider is far left, the color is most opaque; when the slider is farthest right, the color is most transparent.

Context Menu Items

In the visual pull-down color menu in Attribute Manager or Feature Manager, when you right-click a color block, a context menu presents two choices.

Edit Color

When you choose Edit Color, a more detailed dialog box opens that shows you the location of the selected color on the hue+saturation palette and allows you to specify colors by keying in values for either HSL (Hue/Saturation/Luminosity) or RGB (red/green/blue). This provides a way to add and modify custom colors that are not on the standard palette of 40.



Load Color Definition File

When you choose Load Color Definition File, a dialog opens that lets you navigate to and choose a color palette definition file of filetype `*.rgb`.

Examples of custom color palettes

WCMYKRGB hues horizontally, with luminosity varying vertically

X11 cool colors, with color names displayed in mouseover comments

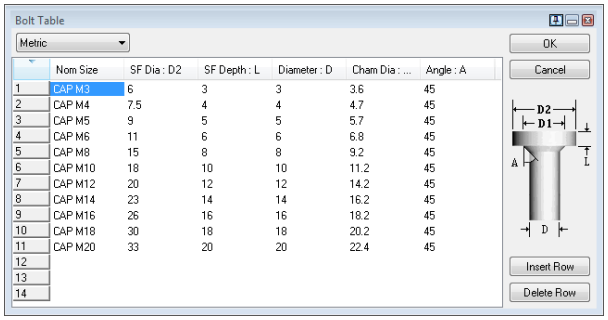
Appendix A: Hole Wizard Data

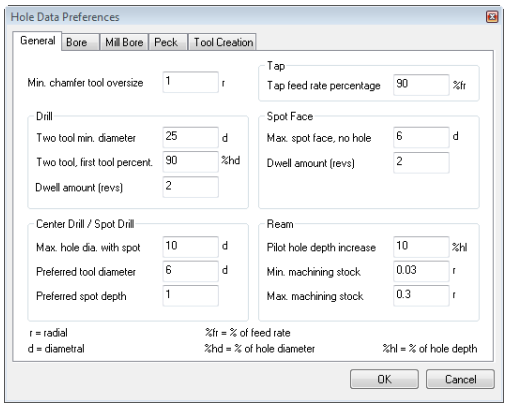
This chapter provides detailed information about Hole Wizard Preferences and explains the various Hole Shapes in Step Two of the Hole Wizard.

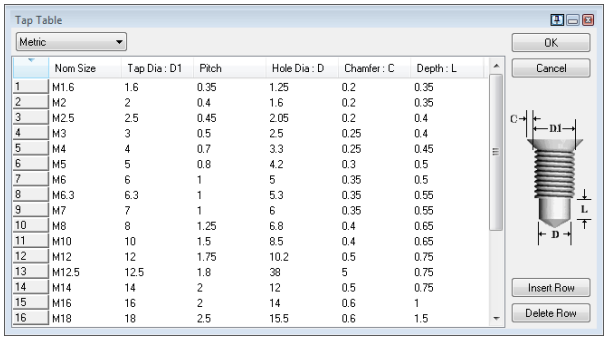
Preferences

The Preferences are the controlling factors of the Hole Wizard's Knowledge Base. This data controls the Hole Wizard behavior. Before using the Hole Wizard for the first time you need to go through the Preferences and enter your preferred values. It is important to know what all of the settings are and how they affect how the hole Wizard behaves. The Knowledge Base keeps track of metric and inch settings separately. Any change made to one measurement unit will not be reflected in the other.

The Knowledge Base consists of the Hole Data preferences, the Tap Table and the Bolt Table. The Hole Data preferences allows you to save preferences for how tools should be selected and which processes should be created. The Tap Table allows the definition of the tapped holes to be machined. This allows auto-selection of tap drills, etc. The Bolt Table allows for frequently created holes to be pre-defined.







Elements of the Hole Wizard preferences.

Hole Data Preferences

The Hole Data preferences consist of settings on several tabs in the dialog box including the General Tab, Bore Tab, Mill Bore Tab and Peck Tab.

General Tab

Minimum chamfer tool oversize:
This is the radial amount by which the chamfer or spotting tool must be larger than the desired chamfer. The tool must be larger than or equal to the chamfer by this amount per side.

Drill

Two tool minimum diameter:
Two drill passes will be used to create a hole if the hole's diameter is greater than or equal to this value.

Two tool, first tool percent:
If two drill passes are used, this is the maximum size hole percentage the first pass will remove.

Dwell amount (revs.):
The number of revolutions to dwell at the bottom of the hole during the drill chamfering process.

Center Drill / Spot Drill

Maximum hole diameter with spot:
Spotting will not be used in Step Two of the Hole Wizard if the hole diameter is greater than this value.

Preferred tool diameter:
This is the preferred tool diameter for spotting. When spotting, the Hole Wizard will first search for a tool of this diameter or a smaller will be suggested.

Preferred spot depth:
The Hole Wizard will use this value as the default depth when spotting.

Tap

Tap feed rate percentage:
This is the percentage of the full feed rate that will be used on the tapping cycle. This value is used in creating the tap process.

Spot Face:

Max spot face, no hole:
The maximum spot face diameter that will not require a pre-drilled hole. If the spot face is greater than this value then the hole must be pre-drilled. This value is used in determining the process

defaults.

Dwell amount (revs.):
The number of revolutions to dwell at the bottom of the hole during the spot face and the bore chamfer processes.

Ream:

Pilot hole depth increase:
The percentage of the ream hole depth to determine the depth of the drill process. The drill will descend the depth of the desired ream hole plus the percentage specified.

Minimum machining stock:
The minimum amount of stock to cut with the Ream Finish process as left by the Ream Medium process.

Maximum machining stock:
The maximum amount of stock to cut with the Ream Finish process as left by the Ream Medium process.

Bore Tab

Minimum diameter for Big Bore:
Used to determine whether the Hole Wizard is to use the Small Bore Stock Allowance or the Big Bore Stock Allowance when boring.

Minimum chamfer tool oversize:
This value is used in bore chamfering tool selection. This is the radial amount by which the chamfer tool must be larger than the desired chamfer, (the tool must be larger than or equal to the chamfer by this amount per side). The value indicated in this example states that the chamfer tool radius must be at least 1mm larger than the chamfer radius.

Maximum chamfer tool oversize:
This value is used in bore chamfering tool selection. The tool must be smaller than or equal to the chamfer by this amount per side. The value indicated in this example states that the chamfer tool radius may be at most 3mm larger than the chamfer radius.

Through bore over travel:
The amount of travel the bore may exceed past the end of the desired bore depth. This value is used in defaulting the over travel value in the bore thru definition dialog. (Step Two in the Hole Wizard).

Back bore clearance:
The amount of clearance the back bore tool may exceed past the bottom of the stock. This value is used in defaulting the clearance value in the back bore hole definition dialog. (Step Two in the Hole Wizard).

Pilot hole depth increase:
This is the percentage of the bore hole depth that is used to determine the depth of the drill process. The drill will descend the depth of the desired bore hole plus the percentage specified.

Small Bore Stock Allowance

Rough maximum:
The maximum amount of stock to remove during the rough pass. Used for process defaulting.

Medium minimum:
The minimum amount of stock to remove during the medium pass. Used for process defaulting and tool selection.

Medium maximum:
The maximum amount of stock to remove during the medium pass. Used for process defaulting and tool selection.

Finish medium:
The minimum amount of stock to remove during the finish pass. Used for process defaulting and tool selection.

Finish maximum:
The maximum amount of stock to remove during the finish pass. Used for process defaulting and tool selection.

Big Bore Stock Allowance

Rough maximum:
The maximum amount of stock to remove during the rough pass. Used for process defaulting.

Medium minimum:
The minimum amount of stock to remove during the medium pass. Used for process defaulting and tool selection.

Medium maximum:
The maximum amount of stock to remove during the medium pass. Used for process defaulting and tool selection.

Finish medium:
The minimum amount of stock to remove during the finish pass. Used for process defaulting and tool selection.

Finish maximum:
The maximum amount of stock to remove during the finish pass. Used for process defaulting and tool selection.

Mill Bore Tab

Rapid clearance amount:
This is the distance from the Clearance Diameter you wish the tool to start feeding. The tool will rapid from the bore center to this location.The Clearance Diameter value may be found under the Bore tab of a Drilling process.

Maximum end mill size:
Only end mills with a diameter equal to or less than this amount will be considered during tool selection.

Rough

Cut width:
The cut width for the mill boring process will be calculated as a percentage of the tool diameter. By default the system will perform a mill bore cut that is 50% of the tool's diameter.

Z step:
The Z Step for the mill boring rough process will be calculated as a percentage of the tool diameter. By default the system will Z step a mill bore cut by 25% of the tool's diameter.

Finish

Maximum finish cut, no spring:
If the finish cut is larger than this value, one spring pass will be taken.

Overlap:
The overlap amount for the mill boring process will be calculated as a percentage of the tool diameter. By default the system will overlap a mill bore cut by 10% of a tool's diameter.

Approach/Exit line:
The approach/exit line length for the mill boring finish process will be calculated as a percentage of the tool diameter. The default value is 5% of a tool's diameter.

Approach/Exit radius:
The approach/exit radius for the mill boring finish process will be calculated as a percentage of the tool diameter. The default value is 25% of a tool's diameter.

Separate finish end mill:
If you would like to use a different end mill for the finish pass of a mill bore process, check this box.

CRC:
Check this box to specify that CRC is active for the mill bore finish pass.

Peck Tab

Maximum depth for FI-RO:
If the [(hole depth ÷ hole diameter)*100] is less than or equal to this value then Feed In-Rapid Out is used for the entry/exit cycle, otherwise pecks will be used. By default the system will rapid out on a hole whose depth is less than twice its diameter.

Peck depth:
The Peck Depth is calculated as a percentage of the hole diameter. The value will be used with the peck full out or chip breaker entry/exit cycles during process creation. By default the system peck depth is equal to the hole's diameter.

Re-entry clearance:
The value represents how close to the prior peck we can rapid back in. This value will be used with the peck full out entry/exit cycle during process creation.

Entry/Exit Cycle

Full Out:
Select this option if you would like your pecking to be defaulted to full-out when pecking.

Chip Breaker:
Select this option if you would like your pecking to be defaulted to chip breaker when pecking.

Retract amount:
The Retract amount is calculated as a percentage of the hole diameter. This value will be used with the chip breaker entry/exit cycles during process creation.

Tool Creation

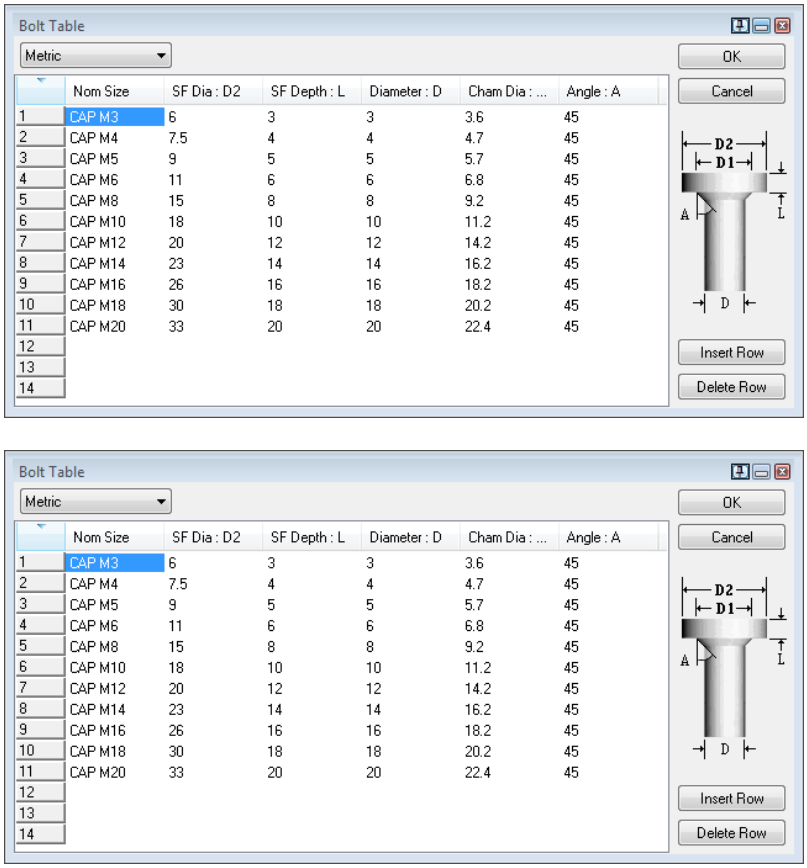
The Tool Creation tab has settings for the Default Angles used throughout the Hole Wizard. The system has a default Chamfer Angle of 45 degrees and the Tip Angle has two default values, 118 and 180 degrees.

Bolt Table

The Bolt Table stores pre-defined values for making bolt holes. It is a list of commonly used pre-set values that define a bolt hole. The Bolt Table comes with several Socket Head Cap industry standards loaded but you may also define your own holes.

The Bolt Table is accessed from the Machining Prefs tab in File >  Preferences, or by clicking the  Bolt Table button in the Hole Wizard. The table button becomes available when you are making a bolt hole. When defining a bolt hole you will have access to these pre-defined sizes from a pull-down menu in the Hole Wizard. The nominal size value (Nom Size) will be displayed in the pull-down menu.

Note: There are separate tables for metric and inch hole definitions. The table used is based on the part's unit system.



	Nom Size	SF Dia : D2	SF Depth : L	Diameter : D	Cham Dia : ...	Angle : A
1	CAP M3	6	3	3	3.6	45
2	CAP M4	7.5	4	4	4.7	45
3	CAP M5	9	5	5	5.7	45
4	CAP M6	11	6	6	6.8	45
5	CAP M8	15	8	8	9.2	45
6	CAP M10	18	10	10	11.2	45
7	CAP M12	20	12	12	14.2	45
8	CAP M14	23	14	14	16.2	45
9	CAP M16	26	16	16	18.2	45
10	CAP M18	30	18	18	20.2	45
11	CAP M20	33	20	20	22.4	45
12						
13						
14						

	Nom Size	SF Dia : D2	SF Depth : L	Diameter : D	Cham Dia : ...	Angle : A
1	CAP M3	6	3	3	3.6	45
2	CAP M4	7.5	4	4	4.7	45
3	CAP M5	9	5	5	5.7	45
4	CAP M6	11	6	6	6.8	45
5	CAP M8	15	8	8	9.2	45
6	CAP M10	18	10	10	11.2	45
7	CAP M12	20	12	12	14.2	45
8	CAP M14	23	14	14	16.2	45
9	CAP M16	26	16	16	18.2	45
10	CAP M18	30	18	18	20.2	45
11	CAP M20	33	20	20	22.4	45
12						
13						
14						

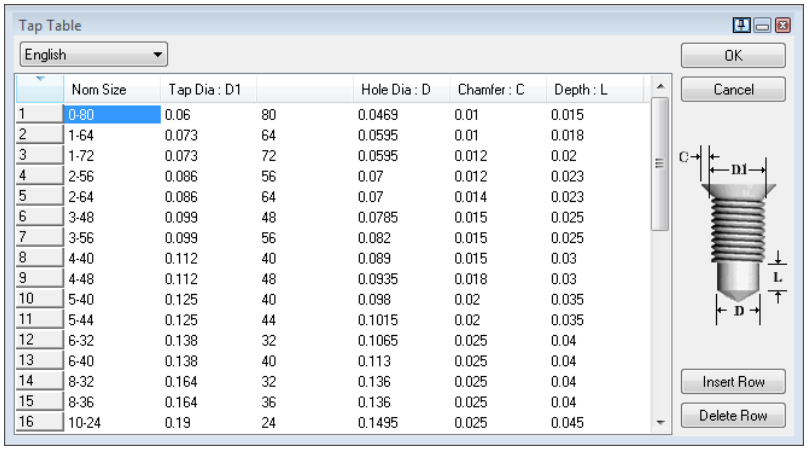
The Bolt Table shown with metric and inch data.

Tap Table

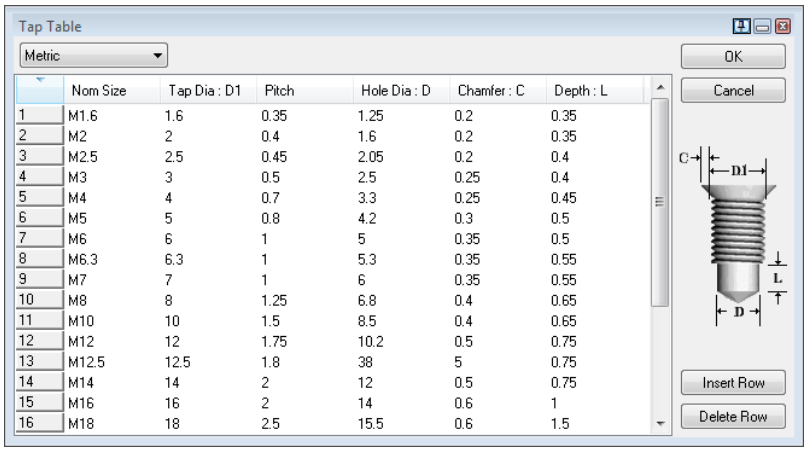
The Tap Table stores pre-defined values for tapping tools. It is a list of commonly used pre-set values that define a tap. The Tap Table comes with several industry standards values loaded but you may also define your own.

The Tap Table is accessed from the Machining Prefs tab or by clicking the Tap Table button in the Hole Wizard. The table button becomes available when you are making a tapping operation. When defining a tap you will have access to these pre-defined sizes from a pull-down menu in the Hole Wizard. The nominal size value (Nom Size) will be displayed in the pull-down menu.

Note: There are separate tables for metric and inch hole definitions. The table used is based on the part's unit system.



	Nom Size	Tap Dia : D1		Hole Dia : D	Chamfer : C	Depth : L
1	0-80	0.06	80	0.0469	0.01	0.015
2	1-64	0.073	64	0.0595	0.01	0.018
3	1-72	0.073	72	0.0595	0.012	0.02
4	2-56	0.086	56	0.07	0.012	0.023
5	2-64	0.086	64	0.07	0.014	0.023
6	3-48	0.099	48	0.0785	0.015	0.025
7	3-56	0.099	56	0.082	0.015	0.025
8	4-40	0.112	40	0.089	0.015	0.03
9	4-48	0.112	48	0.0935	0.018	0.03
10	5-40	0.125	40	0.098	0.02	0.035
11	5-44	0.125	44	0.1015	0.02	0.035
12	6-32	0.138	32	0.1065	0.025	0.04
13	6-40	0.138	40	0.113	0.025	0.04
14	8-32	0.164	32	0.136	0.025	0.04
15	8-36	0.164	36	0.136	0.025	0.04
16	10-24	0.19	24	0.1495	0.025	0.045



	Nom Size	Tap Dia : D1	Pitch	Hole Dia : D	Chamfer : C	Depth : L
1	M1.6	1.6	0.35	1.25	0.2	0.35
2	M2	2	0.4	1.6	0.2	0.35
3	M2.5	2.5	0.45	2.05	0.2	0.4
4	M3	3	0.5	2.5	0.25	0.4
5	M4	4	0.7	3.3	0.25	0.45
6	M5	5	0.8	4.2	0.3	0.5
7	M6	6	1	5	0.35	0.5
8	M6.3	6.3	1	5.3	0.35	0.55
9	M7	7	1	6	0.35	0.55
10	M8	8	1.25	6.8	0.4	0.65
11	M10	10	1.5	8.5	0.4	0.65
12	M12	12	1.75	10.2	0.5	0.75
13	M12.5	12.5	1.8	38	5	0.75
14	M14	14	2	12	0.5	0.75
15	M16	16	2	14	0.6	1
16	M18	18	2.5	15.5	0.6	1.5

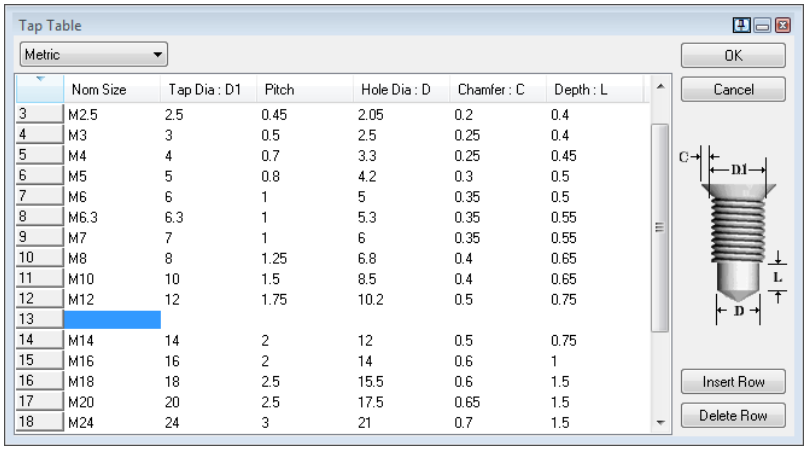
The Tap Table shown with metric and inch data.

Working with Tables

The Tap and Bolt Table may be modified. Any changes made to a table will be available immediately.

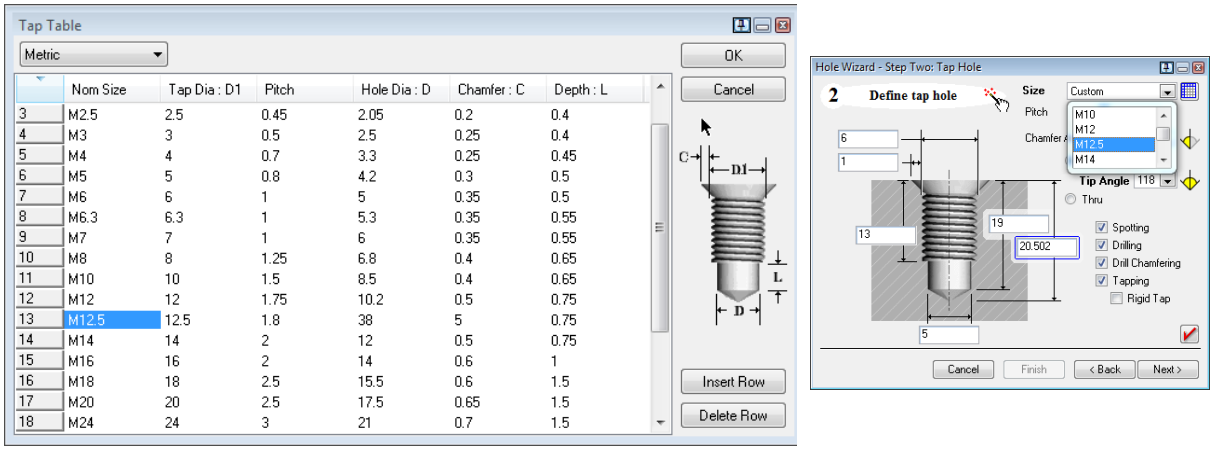
Adding Entries

New rows may be inserted between existing rows. To do this, select a text field in an existing row and click the Insert Row button. A new row will be inserted, moving the selected row down.



	Nom Size	Tap Dia : D1	Pitch	Hole Dia : D	Chamfer : C	Depth : L
3	M2.5	2.5	0.45	2.05	0.2	0.4
4	M3	3	0.5	2.5	0.25	0.4
5	M4	4	0.7	3.3	0.25	0.45
6	M5	5	0.8	4.2	0.3	0.5
7	M6	6	1	5	0.35	0.5
8	M6.3	6.3	1	5.3	0.35	0.55
9	M7	7	1	6	0.35	0.55
10	M8	8	1.25	6.8	0.4	0.65
11	M10	10	1.5	8.5	0.4	0.65
12	M12	12	1.75	10.2	0.5	0.75
13						
14	M14	14	2	12	0.5	0.75
15	M16	16	2	14	0.6	1
16	M18	18	2.5	15.5	0.6	1.5
17	M20	20	2.5	17.5	0.65	1.5
18	M24	24	3	21	0.7	1.5

Once the new row is in the table you may define the data. Six fields are required including the nominal size, (Nom Size) or name of the entry. The Nom Size is the identifier that will show up in the Size menu. Click the OK button to save and close the table. To undo any changes you have made, click the Cancel button.



	Nom Size	Tap Dia : D1	Pitch	Hole Dia : D	Chamfer : C	Depth : L
3	M2.5	2.5	0.45	2.05	0.2	0.4
4	M3	3	0.5	2.5	0.25	0.4
5	M4	4	0.7	3.3	0.25	0.45
6	M5	5	0.8	4.2	0.3	0.5
7	M6	6	1	5	0.35	0.5
8	M6.3	6.3	1	5.3	0.35	0.55
9	M7	7	1	6	0.35	0.55
10	M8	8	1.25	6.8	0.4	0.65
11	M10	10	1.5	8.5	0.4	0.65
12	M12	12	1.75	10.2	0.5	0.75
13	M12.5	12.5	1.8	38	5	0.75
14	M14	14	2	12	0.5	0.75
15	M16	16	2	14	0.6	1
16	M18	18	2.5	15.5	0.6	1.5
17	M20	20	2.5	17.5	0.65	1.5
18	M24	24	3	21	0.7	1.5

A new Tap Table entry and its use in the Hole Wizard.

Deleting Entries

Select the row you wish to delete and `click` the Delete Row button. To undo a delete click the Cancel button.

Hole Shape Dialogs in Step Two

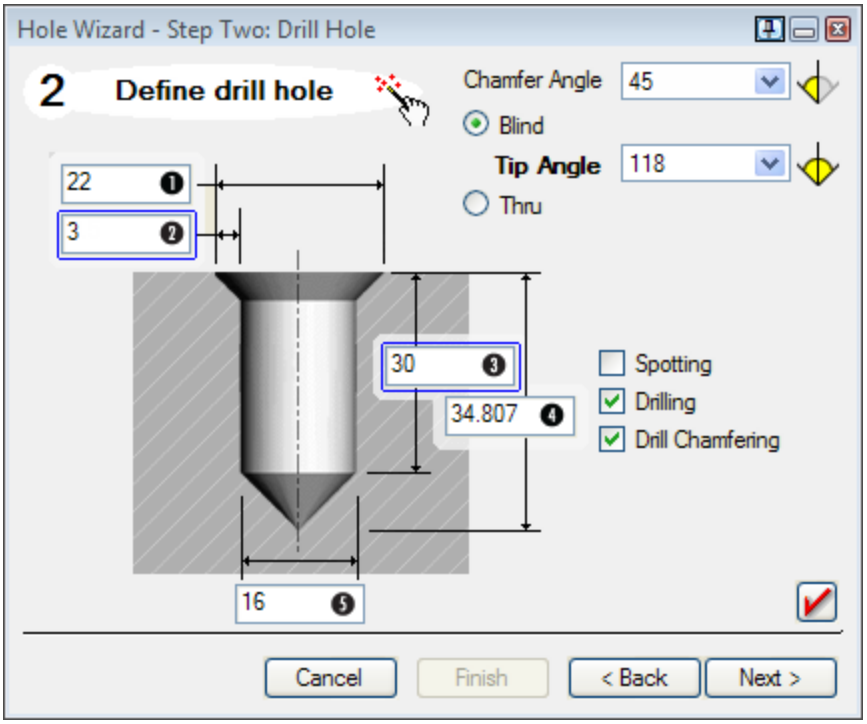
When creating a machining operation through the four step Hole Wizard process, you will encounter some unique and varied dialogs in Step Two. The details for each of the hole shapes may be found on the following pages.

Red arrows indicate dependencies. If the value is changed where the arrow originates from, it will recalculate the box the arrow points to. Changing the dependant value does not change the origin value.

Hole Shape Details

Drill

A drill hole must be defined as either a Blind hole or a Thru hole. Blind holes require a Tip Depth and a Tip Angle value.



1. Chamfer Diameter
2. Chamfer Width
3. Full Diameter Depth/Stock Depth
4. Tip Depth
5. Hole Diameter

Chamfer Diameter
This is the diameter of the hole's chamfer. This will calculate the Chamfer Width.

Chamfer Width
This is the width of the chamfer. This will calculate the Chamfer Diameter.

Full Diameter Depth/Stock Depth
For a Blind hole, this value specifies the lowest Z depth that the full diameter of the tool will drill. This value will recalculate the Tip Depth. For a Thru hole, this value is the thickness of the stock.

Tip Depth
The Tip Depth specifies the final Z depth of the tool tip. This value will dynamically recalculate the Full Diameter Depth.

Hole Diameter
The final width of the hole. This value will dynamically update the unlocked chamfer value.

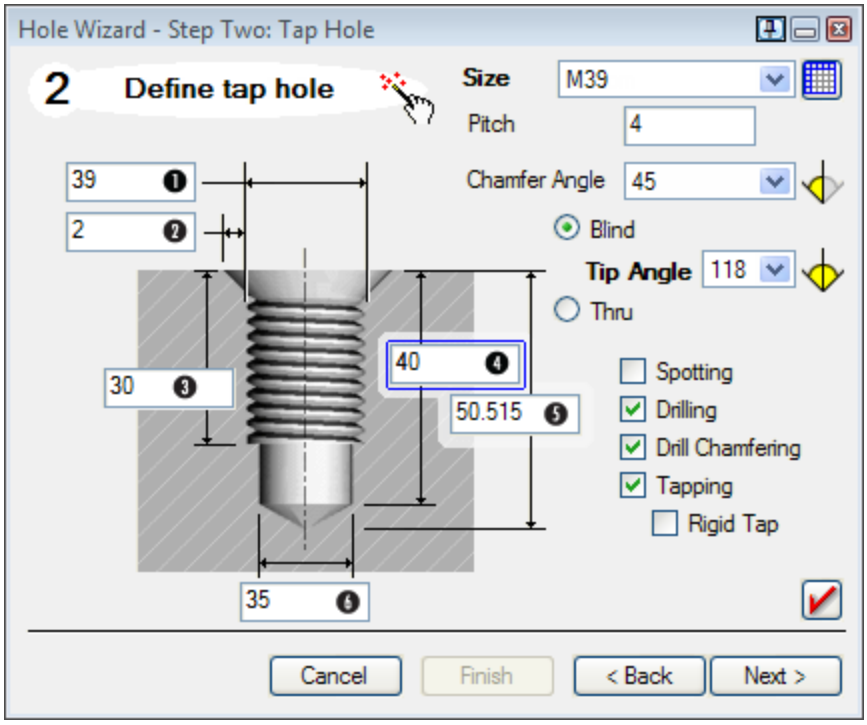
Chamfer Angle
This menu lists the chamfer angles that can be machined with the current tool list. The icon indicates that this value is a half angle, measured from vertical (for example, 45=90˚ tip).

Tip Angle
This menu lists the tip angles of the current tool list. The icon indicates that this value is the full included tip angle.

Processes
The process creation types will change with the hole dimensions. Each item may create multiple processes. Processes may be disabled, but at least one process must be selected. Large holes do not need a spot and may pre-drill first. Drill Chamfering is enabled if a chamfer amount is specified. For more information on the logic of each process, see "Appendix B: Advanced Hole Wizard Use" on page 81.

Tap Hole

A tap hole must be defined as either Blind or Thru. Blind tap holes require a Tip Depth and a Tip Angle value.



1. Tap Diameter
2. Chamfer Width
3. Depth of Thread
4. Hole Diameter
5. Tip Depth
6. Full Diameter Depth/Stock Depth

Tap Diameter
The diameter of the tap of the hole.

Chamfer Width
This is the width of the hole's chamfer.

Depth of Thread
This is the final Z depth of the thread.

Hole Diameter
The final width of the hole. This value will dynamically update the unlocked chamfer value.

Tip Depth
The Tip Depth specifies the final Z depth of the tool tip. This value will dynamically recalculate the Full Diameter Depth.

Full Diameter Depth/Stock Depth
For a Blind hole, this value specifies the lowest Z depth that the full diameter of the tool will drill. This value will recalculate the Tip Depth. For a Thru hole, this value is the thickness of the stock.

Size
This menu lists all the tap sizes in the Tap Table. Selecting a tap size will load the values from the Tap Table into the text boxes. The button next to the menu will open the Tap Table.

Pitch/TPI
This text box allows you to enter the tapped hole's Pitch or Threads Per Inch depending on whether the part is metric or inch.

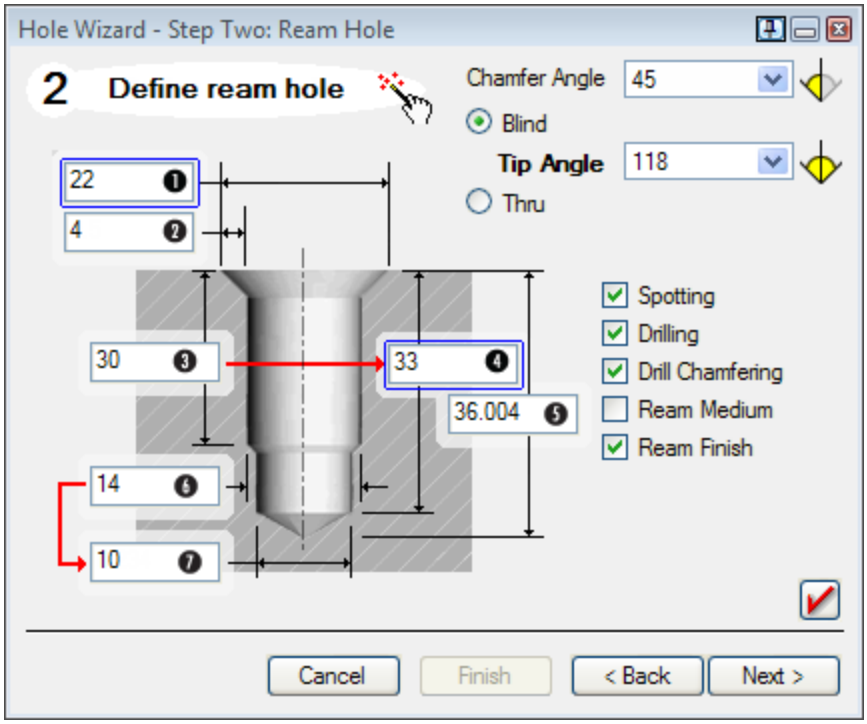
Chamfer Angle
This menu lists the chamfer angles that can be machined with the current tool list. The icon indicates that this value is a half angle, measured from vertical (e.g. 45=90˚ tip).

Tip Angle
This menu lists the tip angles of the current tool list. The icon indicates that this value is the full included tip angle.

Processes
The process creation types will change with the hole dimensions. Each item may create multiple processes. Processes may be disabled, but at least one process must be selected. Tapping will always be checked if a tap was found. Select Rigid Tap if desired. If Tapping is deselected, a hole will be made, but it will only be a drill hole with or without a chamfer. For more information on the logic of each process, see "Appendix B: Advanced Hole Wizard Use" on page 81.

Ream Hole

The smaller diameter shown at the tip is the drilled pilot hole, while the larger diameter is the reamed hole. A ream hole must be defined as either Blind or Thru. Blind ream holes require a Tip Depth and a Tip Angle value.



1. Chamfer Diameter
2. Chamfer Width
3. Full Ream Depth
4. Full Diameter Depth/Stock Depth
5. Tip Depth
6. Ream Finish Diameter
7. Ream Drill Diameter

Chamfer Diameter
This is the diameter of the hole's chamfer. This will calculate the Chamfer Width.

Chamfer Width
This is the width of the chamfer. This will calculate the Chamfer Diameter.

Full Ream Depth:
This is the full depth of the reamed hole. This value is used to calculate the depth of the pilot hole. This is indicated by the red arrow. The calculation is based on your Hole Data preferences.

Full Diameter Depth/Stock Depth
For a Blind hole, this value specifies the lowest Z depth the full diameter of the tool will drill. This value will recalculate the Tip Depth. For a Thru hole, this value is the thickness of the stock.

Tip Depth
The Tip Depth specifies the final Z depth of the tool tip. This value will dynamically recalculate the Full Diameter Depth.

Ream Finish Diameter:
This is the diameter of the reamed hole. The finish reamer must match this value. This value is also used to calculate the pilot hole diameter. The red arrow indicates this. The calculation is based on your Hole Data preferences.

Ream Drill Diameter:
This is the diameter of the ream drill hole.

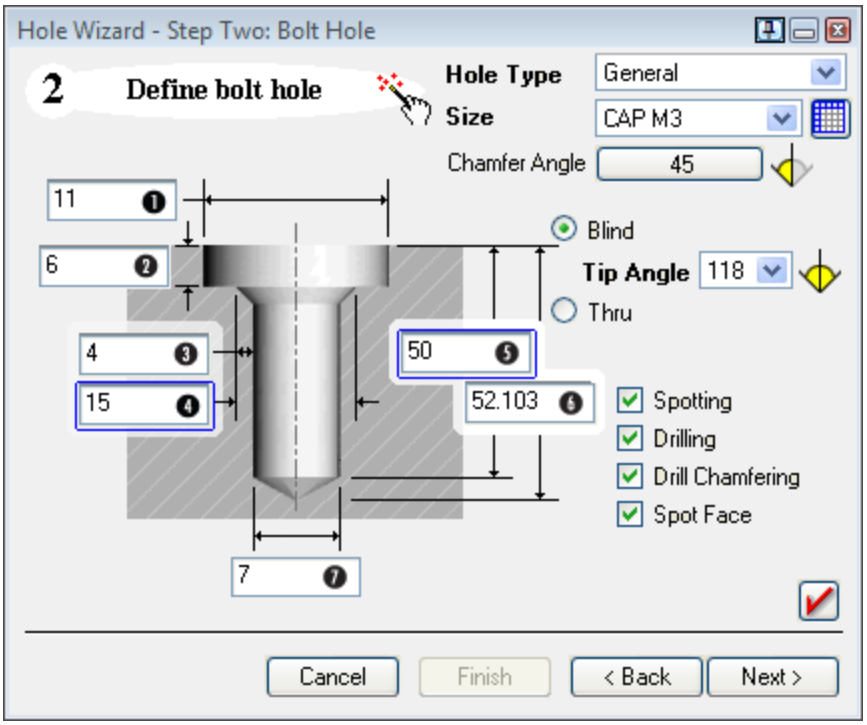
Chamfer Angle
This menu lists the chamfer angles that can be machined with the current tool list. The icon indicates that this value is a half angle, measured from vertical (e.g. 45=90˚ tip).

Tip Angle
This menu lists the tip angles of the current tool list. The icon indicates that this value is the full included tip angle.

Processes
The process creation types will change with the hole dimensions. Each item may create multiple processes. Processes may be disabled, but at least one process must be selected. You should always have Ream Finish checked if an appropriate tool was found. Check Ream Medium if there is too much material left to remove with a single Ream Finish. This is not automatic. For more information on the logic of each process, see "Appendix B: Advanced Hole Wizard Use" on page 81.

Bolt Hole

A bolt hole must be defined as either Blind or Thru. Blind bolt holes require a Tip Depth and a Tip Angle value.



1. Spot Face Diameter
2. Spot Face Depth
3. Chamfer Width
4. Chamfer Diameter
5. Full Diameter Depth/Stock Depth
6. Tip Depth
7. Hole Diameter

Spot Face Diameter:
This specifies the diameter of the spot face. If there is no spot face defined for the hole then set this value equal to the Hole Diameter.

Spot Face Depth
This is the depth of the Spot Face.

Chamfer Width
This is the width of the chamfer. This will calculate the chamfer diameter.

Chamfer Diameter
This is the diameter of the hole's chamfer. This will calculate the chamfer width.

Hole Diameter:
The final width of the hole. This value will dynamically update the unlocked chamfer value.

-Full Diameter Depth/Stock Depth
For a blind hole this value specifies the lowest Z depth the full diameter of the tool will drill. This value will recalculate the Tip Depth. For a thru-hole this value is the thickness of the stock.

Stock Depth:
This is the thickness of the stock.

Tip Depth
The Tip Depth specifies the final Z depth of the tool tip. This value will dynamically recalculate the Full Diameter Depth.

Hole Type:
This menu lists the variety of screw/bolt types. Different holes have less attributes and will disable some fields. General allows all hole shapes to be defined. A Drill Hole specifies a thru-hole with no chamfer. Since Drill Hole forces a 0 chamfer default, a different default will not be loaded from the table. The last five items are the types of fasteners to be used in the machined hole. Hex Head allows a spot face, but forces the defaults for the spot face to 0. Socket Head Cap Screw allows all fields and uses all defaults from the table, except chamfer which it sets to 0. Cheese Head and Pan Head allow a spot face but force 0 defaults. They do not allow a chamfer. Counter Sink will not allow a spot face, but will load the chamfer amount from the table. All hole types will load the thru-hole diameter from the table.

Size:
This menu lists all the bolt sizes in the Bolt Table. The button next to the menu will open the Bolt Table.

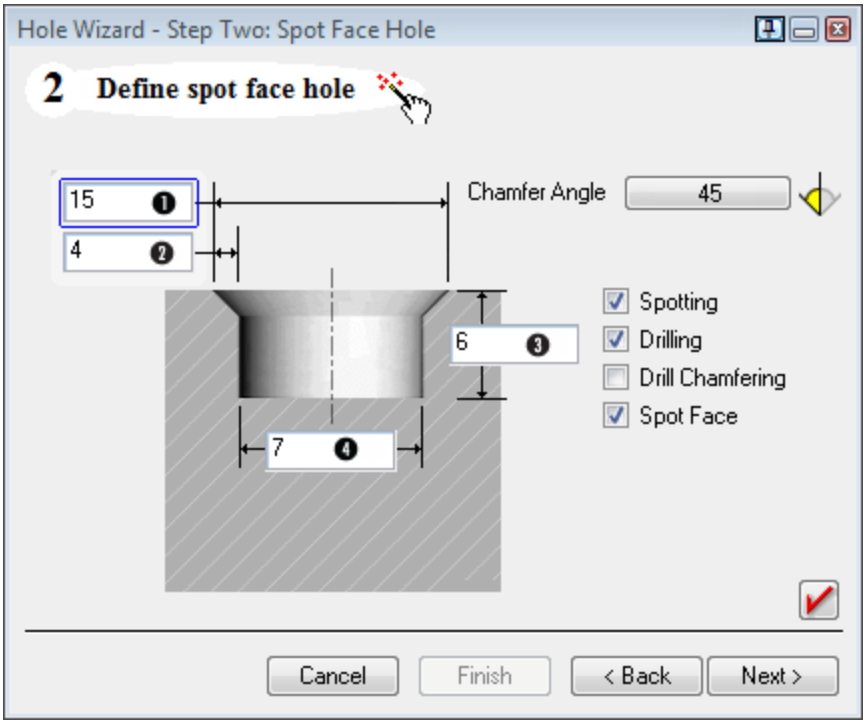
Chamfer Angle
This menu lists the chamfer angles that can be machined with the current tool list. The icon indicates that this value is a half angle, measured from vertical (e.g. 45=90˚ tip).

Tip Angle
This menu lists the tip angles of the current tool list. The icon indicates that this value is the full included tip angle.

Processes:
The process creation types will change with the hole dimensions. Each item may create multiple processes. Processes may be disabled, but at least one process must be selected. Spot Facing will first look for a spot face tool; failing that it will look for a finish end mill, then a rough end mill. To be an acceptable end mill, it must have no taper, a zero non-cutting diameter, and a corner radius less than the max allowable (max CR = larger of 0.01 inches or 5% of tool diameter). For more information on the logic of each process, see “Appendix B: Advanced Hole Wizard Use” on page 81.

Spot Face Hole

Chamfer Diameter
This is the diameter of the hole’s chamfer. This will calculate the chamfer width.



1. Chamfer Diameter
2. Chamfer Width
3. Spot Face Depth
4. Spot Face Diameter

Chamfer Width
This is the width of the chamfer. This will calculate the chamfer diameter.

Spot Face Depth:
This specifies the final depth of the spot face operation. 0 is a valid entry if no spot face is desired.

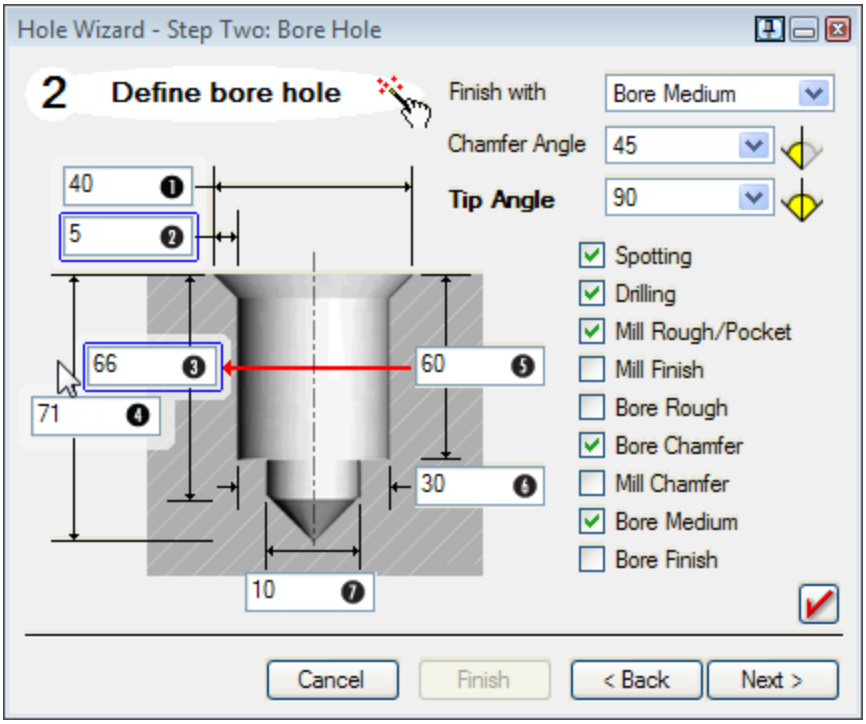
Spot Face Diameter
This is the diameter of the desired spot face operation.

Chamfer Angle
This menu lists the chamfer angles that can be machined with the current tool list. The icon indicates that this value is a half angle, measured from vertical (e.g. 45=90˚ tip).

Processes
The process creation types will change with the hole dimensions. Each item may create multiple processes. Processes may be disabled, but at least one process must be selected. Spot Facing will first look for a spot face tool; failing that it will look for a finish end mill, then a rough end mill. To be an acceptable end mill, it must have no taper, a zero non-cutting diameter, and a corner radius less than the max allowable (max CR = larger of 0.01 inches or 5% of tool diameter). For more information on the logic of each process, see Appendix B: Advanced Hole Wizard Use.

Bore Hole

The smaller diameter shown at the tip is the drilled pilot hole, while the larger diameter is the bored hole.



1. Chamfer Diameter
2. Chamfer Width
3. Full Diameter Depth
4. Tip Depth
5. Bore Full Depth
6. Bore Hole Diameter
7. Bore Drill Hole Diameter

Chamfer Diameter:
This is the diameter of the hole's chamfer. This will calculate the chamfer width.

Chamfer Width
This is the width of the chamfer. This will calculate the chamfer diameter.

Full Diameter Depth/Stock Depth
For a blind hole this value specifies the lowest Z depth the full diameter of the tool will drill. This value will recalculate the Tip Depth. For a thru-hole this value is the thickness of the stock.

Tip Depth
The Tip Depth specifies the final Z depth of the tool tip from the top surface. This value will dynamically recalculate the Full Diameter Depth. If the Full Diameter Depth is entered the Tip Depth will be calculated from the Full Diameter Depth and the Tip Angle.

Bore Full Depth
This specifies the full depth of the bored hole. This depth is used to calculate the depth of the pilot hole based on your Hole Data preferences. The red arrow indicates this.

Bore Hole Diameter
This specifies the finish diameter of the bored hole.

Bore Drill Hole Diameter
This specifies the diameter of the bore drill hole.

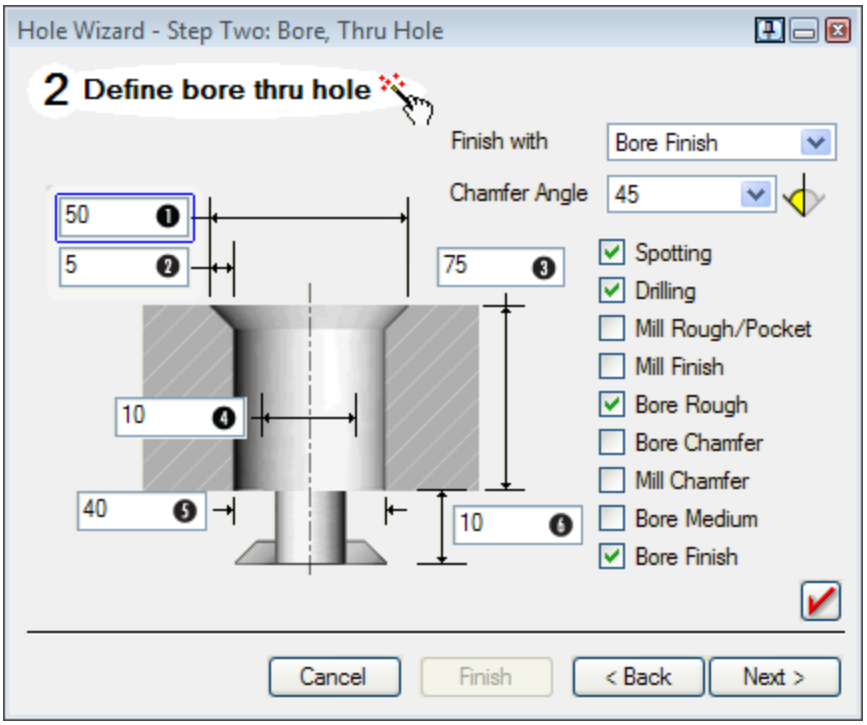
Finish With
The options in this menu allow you to indicate with which type of cycle to finish the bore.

Chamfer Angle
This menu lists the chamfer angles that can be machined with the current tool list. The icon indicates that this value is a half angle, measured from vertical (e.g. 45=90˚ tip).

Tip Angle
This menu lists the tip angles of the current tool list. The icon indicates that this value is the full included tip angle.

Processes
The process creation types will change with the hole dimensions. Each item may create multiple processes. Processes may be disabled, but at least one process must be selected. The process creation types will change with the hole dimensions. Each item may create multiple processes. Processes may be disabled, but at least one process must be selected. If you are boring a hole that has already been drilled, simply uncheck the Drilling and Spotting check boxes as needed before proceeding. A drilling function may or may not be needed based on the drilled hole. Bore Rough, Bore Medium, and Bore Finish all take one pass with a different boring tool, sized to take the cut amount specified in the Hole Data preferences. If appropriate tools aren't found, mill tools that can produce the same results will be used. For more information on the logic of each process, see Appendix B: Advanced Hole Wizard Use.

Bore, Thru Hole



1. Chamfer Diameter
2. Chamfer Width
3. Stock Depth
4. Bore Drill Hole Diameter
5. Bore Hole Diameter
6. Over Travel Allowance

Chamfer Diameter
This is the diameter of the hole's chamfer. This will calculate the chamfer width.

Chamfer Width
This is the width of the chamfer. This will calculate the chamfer diameter.

Bore Drill Hole Diameter:
This specifies the diameter of the bore drill hole.

Bore Hole Diameter
This specifies the finish diameter of the bored hole.

Stock Depth
This is the thickness of the stock.

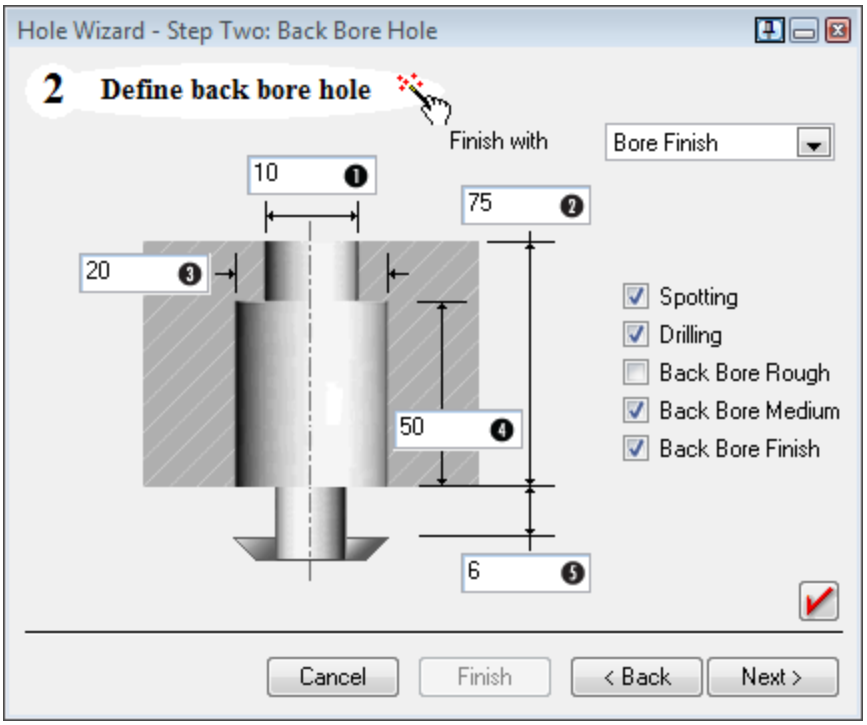
Over Travel Allowance
This is the distance past the end of the stock that the boring bar will travel.

Finish With
The options in this menu allow you to indicate which type of cycle to finish the bore with.

Chamfer Angle
This menu lists the chamfer angles that can be machined with the current tool list. The icon indicates that this value is a half angle, measured from vertical (e.g. 45=90˚ tip).

Processes
The process creation types will change with the hole dimensions. Each item may create multiple processes. Processes may be disabled, but at least one process must be selected. The process creation types will change with the hole dimensions. Each item may create multiple processes. Processes may be disabled, but at least one process must be selected. If you are boring a hole that has already been drilled, simply uncheck the Drilling and Spotting check boxes as needed before proceeding. A drilling function may or may not be needed based on the drilled hole. Bore Rough, Bore Medium, and Bore Finish all take one pass with a different boring tool, sized to take the cut amount specified in the Hole Data preferences. If no appropriate tools are found, then mill tools that can produce the same results will be used. For more information on the logic of each process, see “Appendix B: Advanced Hole Wizard Use” on page 81.

Back Bore Hole



1. Bore Drill Hole Diameter
2. Stock Depth
3. Bore Hole Diameter
4. Back Bore length
5. Over Travel Allowance

Bore Drill Hole Diameter
This specifies the diameter of the bore drill hole.

Stock Depth
This is the thickness of the stock.

Bore Hole Diameter
This specifies the finish diameter of the bored hole.

Back Bore Length
This is the length of the back bore.

Over Travel Allowance
This is the distance past the end of the stock that the boring bar will travel.

Finish With
The options in this menu allow you to indicate which type of cycle to finish the bore with.

Processes
The process creation types will change with the hole dimensions. Each item may create multiple processes. Processes may be disabled, but at least one process must be selected. If you are boring a hole that has already been drilled, simply uncheck the Drilling and Spotting check boxes as needed before proceeding. Back Bore Rough, Back Bore Medium, and Back Bore Finish all take one pass with a different boring tool, sized to take the cut amount you specified in your Hole Data, and to leave the material amount required by the next bore pass. Back Bore Rough will remove more material than Back Bore Medium, which is allowed to remove more material than Back Bore

Finish. You define these amounts in your Hole Data preferences. A drilling function may or may not be needed based on the drilled hole. For more information on the logic of each process, see “Appendix B: Advanced Hole Wizard Use” on page 81.

Appendix B: Advanced Hole Wizard Use

This section is meant as reference for advanced users needing to customize the Hole Wizard. The information is presented in a manner as user-friendly as possible. Despite this, the nature of the information is rather complex. An understanding of logic and basic math skills are needed.

How Does the Hole Wizard Work?

The Hole Wizard automates the process of making holes using logic in conjunction with your Knowledge Base. The parameters you set in the Hole Wizard Preferences dialog affect the way holes are made by the Hole Wizard. Even the tools available in the Tool List affect how the Hole Wizard makes a hole. There are many variables that the Hole Wizard takes into account when setting up a drilling process.

This section describes how the Hole Wizard selects tools and determines drilling process values. This detailed background information about Hole Wizard functionality is intended for advanced users. You can use this information to tailor Hole Wizard Preferences for your specific requirements.

How Does Logic Work?

Logic is a method of reasoning through connections. In order to understand tool selection and process creation you will need to have a fundamental understanding of logic. The details of tool selection and process creation are presented in a layout similar to an outline. There is a subject followed by relevant points and conclusions. Sometimes a point will bring up another point or subject. This is illustrated in the image below.

Subject	
	Point
	Conclusion
	Point
	Point
	Conclusion
	Point
	Conclusion

A simple example of a logical outline.

Tool Selection

Tool selection follows a certain criteria set up for each type of tool and how it is to be used. The selections are made based on hole size, Knowledge Base data, internal parameters and available tools. As selection progresses, tools are made, compared against each other and go through a process of elimination. The logic behind tool selection will be detailed for each type of tool process in this section. Tool processes include Center Drill, Drill, Drill Chamfer, Tap, Ream Medium, Ream Finish, Spot Face, Bore Rough, Bore Medium, Bore Finish, Bore Chamfer, Mill Rough, Mill Chamfer, Back Bore Rough, Back Bore Medium and Back Bore Finish.

Example of Tool Selection

This is a sample of how tool selection works and will help familiarize you with the layout of the description. Much of the selection is described following the system's logic pattern. The system works its way through the **Criteria** for each type of tool that matches the Tool Choices for the hole in question.

Current Process	This is the machining process for which we are trying to find a tool. An example is Drilling or Tapping.
Future Processes	These are additional processes that will be used to machine the hole. Future Processes have a direct impact on the tool selection logic. Just as you would probably not make a tool selection without considering what processes will happen after a specific hole, the Hole Wizard looks at what will happen later to figure out what tool is best to use.
Tool Choices	These are the types of tools that can be used for the process, that the Hole Wizard will be choosing from. Examples are Center Drill, Drill, and Counter Sink.
Usage	This is how the tool will be used. Examples are spotting, drilling and chamfering.

Criteria

The Criteria section details the tool selection logic. All of the conditions must be met for a tool to be used in this process. The criteria for some holes are very simple, such as a tool's diameter must be equal to the hole's diameter. Criteria for other holes is very complex. It is best to consider the Criteria a process of elimination. All of the criteria are read from top to bottom. If a tool's data matches the criteria, the system moves to the next item. If a tool does not match, it is no longer a candidate for use on the hole in question.

Below is an example of the criteria for a hole process. Text in the criteria that is in a light font represents items which may be found in the Hole Wizard preferences. The description for quoted items may be found in the Formulas and Glossary section.

tool draft angle = hole chamfer angle
tool flute length > Chamfer Depth + (Min. chamfer tool oversize * 2.0)
tool tip diameter < hole diameter - (Min. chamfer tool oversize * 2.0)

The above criteria is stating that the Hole Wizard is first looking for a tool whose draft angle is the same as the hole's chamfer angle. If there is no such tool, the search is over. A tool will need to be defined and the Hole Wizard will make a tool recommendation.

If such a tool is found then the Hole Wizard moves to the next criteria. Here the same tool's flute must be longer than the Chamfer Depth plus the Min. chamfer tool oversize multiplied by 2.0. Again, if no tool is found the search is over and the Hole Wizard makes a recommendation on the needed tool. However, if the tool does match the second criteria, the Hole Wizard moves on to the third criteria. This will continue until all criteria have been run through and all tools of the type needed are tested.

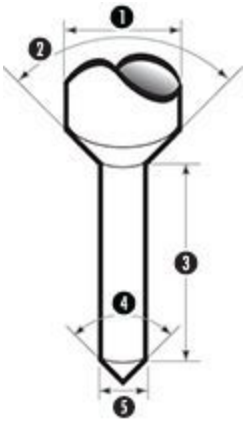
Best Use

If all of the criteria have been tested and more than one tool is a candidate the Hole Wizard will go through another process of elimination to pick which tool to use. This is a last resort of how to make the best choice out of several excellent options.

Center Drill Process 1

When the only process checked in Step Two of the Hole Wizard is Spotting the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a center drill. The center drill will be used to drill the hole and/or make a chamfer.

Current Process	Spotting
Future Processes	none
Tool Choices	center drill
Usage	drill &/or chamfer



1. Shank Diameter
2. Draft Angle
3. Flute Length
4. Tip Angle
5. Pilot Diameter

Elements of a Center Drill

Criteria

1. The tool pilot diameter must be equal to the hole diameter.
2. The tool tip angle must be equal to the hole tip angle if the specified hole has a tip angle specified.
3. The tool draft angle must equal the hole chamfer angle.
4. The tool flute length + Chamfer Depth must be less than or equal to the hole depth.

Best Use

Of all possible solutions, the tool with the longest flute length will be chosen. In the event of a tie, selection is based upon Standard Comparison.

Center Drill Process 2

When the checked processes in Step Two of the Hole Wizard are Spotting and Drilling, the following logic applies to tool selection. The Hole Wizard will be looking for a center drill or a spot drill to make this hole. The center drill is the preferred choice for this operation.

Current Process	Spotting
Future Processes	Drilling
Tool Choices	center drill, spot drill
Usage	spot &/or chamfer

Criteria

Center Drill

1. The tool draft angle must equal the hole chamfer angle.
2. The tool pilot diameter must be less than the hole diameter.
3. The tool shank diameter must be greater than the hole diameter + chamfer diameter + Min chamfer tool oversize * 2.0.
   [tool shank dia. > hole dia. + chamfer dia.+ (Min chamfer tool oversize * 2.0)]
4. The tool flute length + Chamfer Depth must be less than the hole depth.

Spot Drill

1. The tool tip angle * 0.5 must equal the hole chamfer angle.
2. The tool diameter must be greater than the hole diameter + chamfer diameter + Min chamfer tool oversize * 2.0.
   [tool shank dia. > hole dia. + chamfer dia.+ (Min chamfer tool oversize * 2.0)].
3. The Chamfer Depth must be less than the full hole depth (including tip where applicable).

Best Use

Choose the best center drill, otherwise choose spot drill.

1. The best pick is the tool with a diameter equal to Center drill preferred diam.
2. The tool with the largest diameter is best.
3. In the event of a tie, selection is based upon Standard Comparison.

Note that Min chamfer tool oversize is from the General tab, not the Bore tab in the Hole Data preferences.

Center Drill Process 3

When the process checked in Step Two of the Hole Wizard are Spotting, Drilling and Drill chamfering the following logic applies to tool selection. The Hole Wizard will be looking for a spot drill or center drill. The selected tool will be used to spot the hole.

Current Process	Spotting
Future Processes	Drilling, Drill Chamfering
Tool Choices	spot drill, center drill
Usage	spot

Criteria

Spot Drill

if tool diameter = Center drill preferred diam		
	then use this tool	
else tool diameter > [hole diameter + (Min chamfer tool oversize * 2.0)]		

Center Drill

if hole diameter > Center drill preferred diam.			
	if tool diameter = Center drill preferred diam		
		then use this tool	
	else hole diameter > tool diameter		
else hole diameter > tool diameter			

Best Use

1. Choose the best spot drill otherwise choose center drill.

   The best pick is the tool with a diameter equal to Center drill preferred diam.

   Spot Drill: Smallest of tool diameter - hole diameter + (Min chamfer tool oversize * 2.0)

   Center Drill: The tool with the largest diameter is best.

2. In the event of a tie, selection is based upon Standard Comparison.

Center Drill Process 4

When Spotting and Drill Chamfering are checked in Step Two of the Hole Wizard, the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a center drill. The center drill will be used to drill the hole and make a chamfer.

Current Process	Spotting
Future Processes	Drill Chamfering
Tool Choices	center drill
Usage	drill

Criteria

tool pilot diameter = hole diameter

tool flute length ≥ hole depth

tool tip angle = hole tip angle (only if the tip angle exists for the hole type selected)

Best Use

1. The tool with the longest flute length will be chosen.
2. In the event of a tie, selection is based upon Standard Comparison.

Drilling Process

When the only process checked in Step Two of the Hole Wizard is Drilling the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a drill.

Current Process	Drilling
Future Processes	none
Tool Choices	drill
Usage	drill

Drilling is divided into one or two passes depending on the size of the machining diameters.
One Pass Machining is used when the hole diameter is ≤ Two tool min. diameter
Two Pass Machining is used when the hole diameter is ≥ Two tool min. diameter

Criteria

Tool tip angle = hole tip angle (only if the tip angle exists for the hole type selected)

One Pass Machining

Drill, Tap, Ream & Bolt holes	hole diameter = tool diameter
Spot facing	tool diameter < hole diameter

Two Pass Machining

1st Drill pass	tool diameter ≤ hole diameter * (Two tool, first tool percent/100) (not to exceed the Two tool min. diameter)

2nd Drill pass	The same tool as selected for the Single pass machining.

For all drill tools a length check must be made:
(tool length - tool tip length) > drill hole depth

Best Use

Drill, Tap, Ream & Bolt holes	Selection is based upon Standard Comparison.
Spot facing	Select the tool with the largest diameter Selection is based upon Standard Comparison.

Drill Chamfering Process

When the only process checked in Step Two of the Hole Wizard is Drill Chamfering the following logic applies to tool selection. The Hole Wizard will be looking for a counter sink, spot drill, center drill or drill.

Current Process	Drill Chamfering
Future Processes	none
Tool Choices	counter sink, spot drill, center drill, drill
Usage	chamfer

Criteria

Counter Sink
Spot Drill
Drill

drill hole tap hole ream hole	tool diameter ≥ (hole diameter + chamfer diameter + Min. chamfer tool oversize * 2.0) Chamfer Depth < full hole depth (including tip where applicable) tool tip angle = hole chamfer angle * 2.0
spot face hole	tool diameter ≥ (hole diameter + chamfer diameter + Min. chamfer tool oversize * 2.0) Chamfer Depth < hole depth

	tool tip angle = hole chamfer angle * 2.0
bolt hole	tool diameter ≥ (hole diameter + chamfer diameter + Min. chamfer tool oversize * 2.0) hole top diameter ≥ tool diameter tool tip angle = hole chamfer angle * 2.0

Center Drill

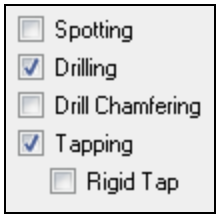
drill hole tap hole ream hole	tool shank diameter ≥ (hole diameter + chamfer dia. + Min. chamfer tool oversize * 2.0) tool pilot diameter < hole diameter Chamfer Depth < full hole depth (including tip where applicable) tool draft angle = hole chamfer angle
spot face hole	tool shank diameter ≥ (hole diameter + chamfer dia. + Min. chamfer tool oversize * 2.0) tool pilot diameter < hole diameter Chamfer Depth < hole depth tool draft angle = hole chamfer angle
bolt hole	tool shank diameter ≥ (hole diameter + chamfer dia. + Min. chamfer tool oversize * 2.0) hole top diameter ≥ tool shank diameter tool pilot diameter < hole diameter tool draft angle = hole chamfer angle

Best Use

1. Pick counter sink, Pick spot drill, pick center drill, Pick drill
2. In the event of a tie, selection is based upon Standard Comparison.

Note that Min chamfer tool oversize is from the General tab, not the Bore tab in the hole data preferences.

Tapping Process



When the only non-drilling process checked in Step Two of the Hole Wizard is Tapping, the following logic applies to tool selection. The Hole Wizard will be looking for a tap or rigid tap.

Current Process	Tapping
Future Processes	none
Tool Choices	tap, rigid tap
Usage	tap

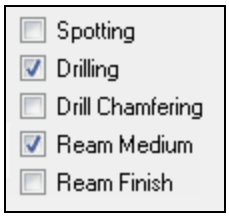
Criteria

If ***Rigid Tap*** is checked in the dialog then only rigid tap tools are evaluated, otherwise tap tools are evaluated.
tool diameter = hole diameter
tool pitch = hole pitch

Best Use

Selection is based upon Standard Comparison.

Ream Medium Process 1



When Ream Medium is the only non-drilling process that is checked in Step Two of the Hole Wizard the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a reamer.

Current Process	Ream Medium
Future Processes	none
Tool Choices	reamer
Usage	ream

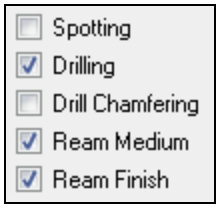
Criteria

tool diameter = hole diameter

Best Use

Selection is based upon Standard Comparison.

Ream Medium Process 2



When the only non-drilling process checked in Step Two of the Hole Wizard are Ream Medium and Ream finish the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a reamer.

Current Process	Ream Medium
Future Processes	Ream Finish
Tool Choices	reamer
Usage	ream

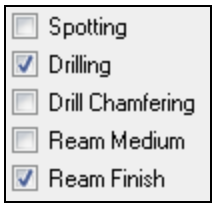
Criteria

Ream min. mach. stock ≤ [(ream hole diameter - tool diameter) / 2.0]
[(ream hole diameter - tool diameter) / 2.0] ≤ Ream max. mach. stock
tool diameter > drilled hole diameter (hole minor diameter)

Best Use

Selection is based upon Standard Comparison.

Ream Finish Process



When Ream Finish is the only non-drilling process that is checked in Step Two of the Hole Wizard the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a reamer.

Current Process	Ream Finish
Future Processes	none
Tool Choices	reamer
Usage	ream

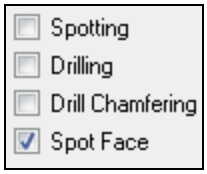
Criteria

tool diameter = hole diameter

Best Use

Selection is based upon Standard Comparison.

Spot Face Process



When Spot Face is the only non-drilling process that is checked in Step Two of the Hole Wizard the following logic applies to tool selection. The Hole Wizard will be looking for a spot face tool, finish end mill or rough end mill.

Current Process	Spot Face
Future Processes	none
Tool Choices	spot face, finish end mill, rough end mill
Usage	spot face

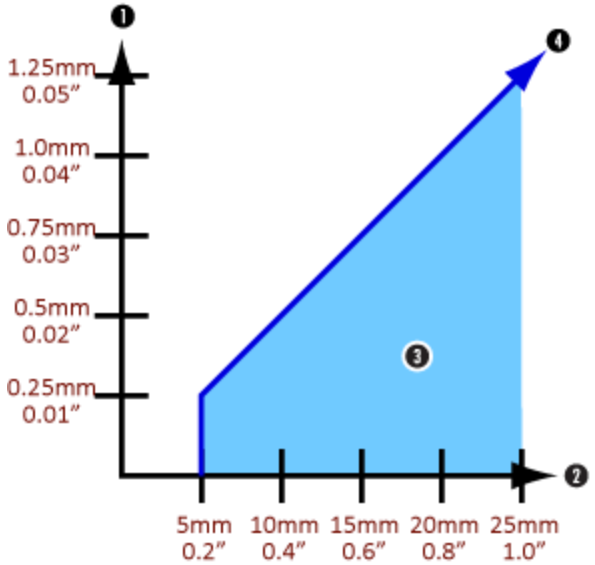
Criteria

tool diameter = spot face diameter

draft angle = 0

non-cutting diameter = 0

corner radius can be 5% of tool diameter with a minimum radius of 0.25mm or 0.01". The largest the corner radius may be is 5% of the tool's diameter, but the smallest the corner radius may be is 0.25mm or 0.01".



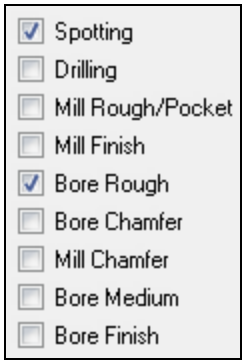
1. Corner Radius
2. Tool Size
3. CR ≤ 5% Tool Dia.
4. Max Corner Radius

Best Use

1. spot face
2. rEM
3. fEM

4. Selection is based upon Standard Comparison.

Bore Rough Process 1



When Bore Rough is the only non-drilling process that is checked in Step Two of the Hole Wizard the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a bore.

Current Process	Bore Rough
Future Processes	none
Tool Choices	bore
Usage	bore

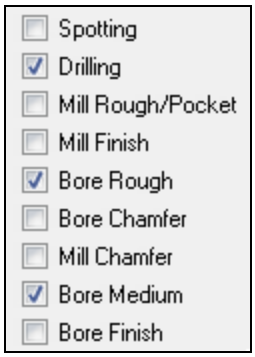
Criteria

tool diameter = hole diameter

Best Use

Selection is based upon Standard Comparison.

Bore Rough Process 2



When the only non-drilling processes checked in Step Two of the Hole Wizard are Bore Rough and Bore Medium the following logic applies to tool selection. The only type of tool that the Hole Wizard looks for is a bore.

Current Process	Bore Rough
Future Processes	Bore Medium
Tool Choices	bore
Usage	bore

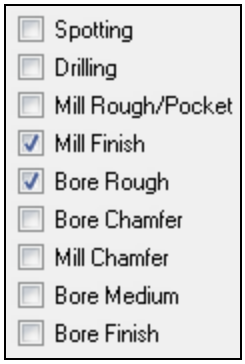
Criteria

(hole diameter - BMMaxAllowance * 2.0) ≤ tool diameter ≤ (hole diameter - BMMinAllowance * 2.0)

Best Use

Selection is based upon Standard Comparison.

Bore Rough Process 3



When the only non-drilling processes checked in Step Two of the Hole Wizard are Bore Rough and Bore Finish or Mill Finish, the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a bore.

Current Process	Bore Rough
Future Processes	Bore Finish OR Mill Bore finish
Tool Choices	bore
Usage	bore

Criteria

(hole diameter - BFMinAllowance * 2.0) ≤ tool diameter ≤ (hole diameter - BFMinAllowance * 2.0)

Best Use

Selection is based upon Standard Comparison.

Bore Rough Process 4

When the non-drilling processes checked in Step Two of the Hole Wizard are Bore Rough, Bore Medium and either Bore Finish *or* Mill Finish are checked in Step Two of the Hole Wizard, the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a center bore.

Current Process	Bore Rough
Future Processes	Bore medium & Bore Finish OR Mill Bore finish
Tool Choices	bore
Usage	bore

Criteria

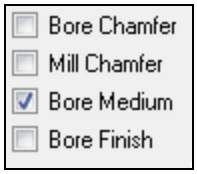
First find the tool that will be used in the Bore Medium process:
(hole diameter - BFMaxAllowance * 2.0) ≤ tool diameter ≤ (hole diameter - BFMinAllowance * 2.0)
Pick bore medium tool with the largest diameter

	If bore medium tool found:	
		bore med. tool dia. - BFMaxAllowance * 2.0) ≤ tool dia. ≤ (bore med. tool dia. - BFMinAllowance * 2.0)
	else	
		(assume an end mill will be used for Bore Medium)
		removed diameter = hole diameter - (BFMinAllowance * 2.0)
		removed dia. - (BFMinAllowance * 2.0) ≤ tool dia. < removed dia. - (BFMinAllowance * 2.0)

Best Use

Selection is based upon Standard Comparison.

Bore Medium Process 1



When Bore Medium is the only non-drilling process that is checked in Step Two of the Hole Wizard the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a bore.

Current Process	Bore Medium
Future Processes	none
Tool Choices	bore
Usage	bore

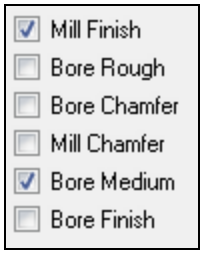
Criteria

tool diameter = hole diameter

Best Use

Selection is based upon Standard Comparison.

Bore Medium Process 2



When the non-drilling processes checked in Step Two of the Hole Wizard are Bore Medium and either Bore Finish *or* Mill Finish, the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a bore.

Current Process	Bore Medium
Future Processes	Bore Finish OR Mill Bore Finish
Tool Choices	bore
Usage	bore

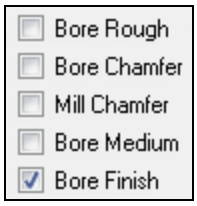
Criteria

(hole diameter - BMMaxAllowance * 2.0) ≤ tool diameter ≤ (hole diameter -BMMinAllowance * 2.0)

Best Use

Selection is based upon Standard Comparison.

Bore Finish Process



When Bore Finish is the only non-drilling process that is checked in Step Two of the Hole Wizard the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a bore.

Current Process	Bore Finish
Future Processes	none
Tool Choices	bore
Usage	bore

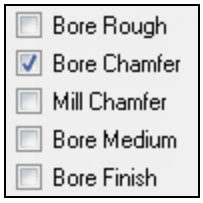
Criteria

tool diameter = hole diameter

Best Use

Selection is based upon Standard Comparison.

Bore Chamfer Process



When the Bore Chamfer process is checked in Step Two of the Hole Wizard the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a counter sink.

Current Process	Bore Chamfer
Future Processes	none
Tool Choices	counter sink
Usage	bore chamfer

Criteria

low ≤ tool diameter ≤ high

low = hole diameter + chamfer diameter + Min. chamfer tool oversize * 2.0

high = hole diameter + chamfer diameter + Max. chamfer tool oversize * 2.0

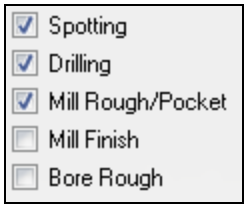
(tool tip angle * 0.5) = hole chamfer angle

Best Use

Selection is based upon Standard Comparison.

Note that Min chamfer tool oversize and Max chamfer tool oversize are from the Bore tab, not the General tab in the hole data preferences.

Mill Rough Process



When Mill Rough/Pocket is the only non-drilling process that is checked in Step Two of the Hole Wizard the following logic applies to tool selection. The Hole Wizard will be looking for a rough end mill or a finish end mill. The end will be used as a substitute for a bore.

Current Process	Mill Rough/Pocket
Future Processes	none
Tool Choices	rough end mill, finish end mill
Usage	bore substitute

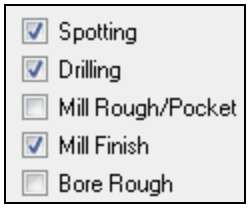
Criteria

tool diameter < pre-drilled hole diameter

Best Use

Selection is based upon Standard Comparison.

Mill Finish Process



When Mill Finish is the only non-drilling process that is checked in Step Two of the Hole Wizard the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a finish end mill. The end will be used as a substitute for a bore.

Current Process	Mill Finish
Future Processes	none
Tool Choices	finish end mill
Usage	bore substitute

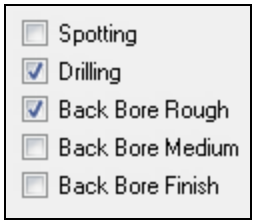
Criteria

tool diameter < [hole diameter - (BMMaxAllowance * 2.0)]
if Separate finish end mill was checked then Mill Finish tool ¦ Mill Rough tool

Best Use

Selection is based upon Standard Comparison.

Back Bore Rough Process 1



When the only non-drilling process checked in Step Two of the Hole Wizard is Back Bore Rough the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a back bore.

Current Process	Back Bore Rough
Future Processes	none
Tool Choices	back bore
Usage	back bore

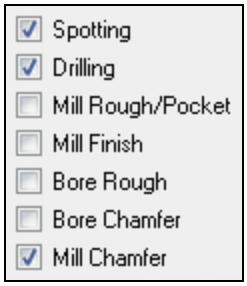
Criteria

tool diameter = hole diameter

Best Use

Selection is based upon Standard Comparison.

Mill Chamfer Process



When the Mill Chamfer process is checked in Step Two of the Hole Wizard, the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a finish end mill. The end mill will be used as a substitute for a bore chamfer.

Current Process	Mill Chamfer
Future Processes	none
Tool Choices	finish end mill
Usage	bore chamfer substitute

Criteria

tool draft angle = hole chamfer angle

tool flute length > Chamfer Depth + (Min. chamfer tool oversize * 2.0)

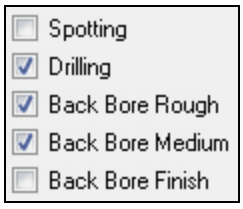
tool tip diameter < [hole diameter - (Min. chamfer tool oversize * 2.0)]

Best Use

Selection is based upon Standard Comparison.

Note that Min chamfer tool oversize is from the Bore tab, not the General tab in the hole data preferences.

Back Bore Rough Process 2



When the only non-drilling process checked in Step Two of the Hole Wizard are Back Bore Rough and Back Bore Medium the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a back bore.

Current Process	Back Bore Rough

Future Processes	Back Bore Medium
Tool Choices	back bore
Usage	back bore

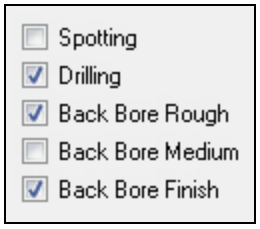
Criteria

(hole diameter - BMMaxAllowance * 2.0) ≤ tool diameter ≤ (hole diameter - BMMinAllowance * 2.0)

Best Use

Selection is based upon Standard Comparison.

Back Bore Rough Process 3



When the only non-drilling process checked in Step Two of the Hole Wizard are Back Bore Rough and Back Bore Finish the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a back bore.

Current Process	Back Bore Rough
Future Processes	Back Bore Finish
Tool Choices	back bore
Usage	back bore

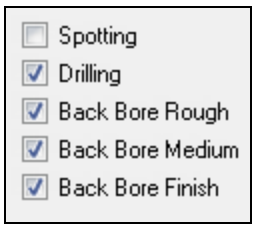
Criteria

(hole diameter - BMMaxAllowance * 2.0) ≤ tool diameter ≤ (hole diameter - BMMinAllowance * 2.0)

Best Use

Selection is based upon Standard Comparison.

Back Bore Rough Process 4



When the non-drilling process checked in Step Two of the Hole Wizard are Back Bore Rough, Back Bore Medium and Back Bore Finish the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a back bore.

Current Process	Back Bore Rough
Future Processes	Back Bore Medium & Back Bore Finish
Tool Choices	back bore
Usage	back bore

Criteria

First find the bore tool that will be used in the Bore Medium process.

(hole diameter - BMMaxAllowance * 2.0) ≤ bore medium tool ≤ (hole dia. - BMMinAllowance * 2.0)

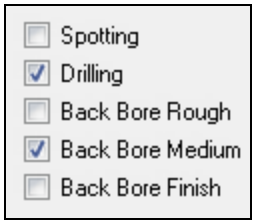
Pick the bore medium tool with the largest diameter

	If bore medium tool found	
		bore med. tool dia. - (BMMaxAllowance * 2.0) ≤ tool dia ≤ bore med. tool dia. - (BMMinAllowance * 2.0)
	else	
		(assume an end mill will be used for the bore medium)
		removed dia = hole diameter - (BMMinAllowance * 2.0)
		removed dia - (BMMaxAllowance * 2.0) ≤ tool dia ≤ removed dia - (BMMinAllowance * 2.0)

Best Use

Selection is based upon Standard Comparison.

Back Bore Medium Process 1



When the only non-drilling process checked in Step Two of the Hole Wizard is Back Bore Medium the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a back bore.

Current Process	Back Bore Medium
Future Processes	none
Tool Choices	back bore
Usage	back bore

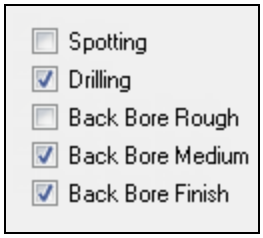
Criteria

tool diameter = hole diameter

Best Use

Selection is based upon Standard Comparison.

Back Bore Medium Process 2



When the only non-drilling process checked in Step Two of the Hole Wizard are Back Bore Medium and Back Bore Finish the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a back bore.

Current Process	Back Bore Medium
Future Processes	Back Bore Finish
Tool Choices	back bore
Usage	back bore

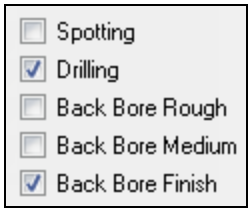
Criteria

(hole diameter - BMMaxAllowance * 2.0) ≤ tool diameter ≤ (hole diameter - BMMinAllowance * 2.0)

Best Use

Selection is based upon Standard Comparison.

Back Bore Finish Process



When the only non-drilling process checked in Step Two of the Hole Wizard is Back Bore Finish the following logic applies to tool selection. The only type of tool that the Hole Wizard will be looking for is a back bore.

Current Process	Back Bore Finish
Future Processes	none
Tool Choices	back bore
Usage	back bore

Criteria

tool diameter = hole diameter

Best Use

Selection is based upon Standard Comparison.

Process Creation

This section will document process creation. The section is intended to provide insight into how the Hole Wizard creates a process and sets the processes' values. Just as with tool selection, process creation is done through pre-defined logic and the Knowledge Base but is influenced by the results of tool selection. We will present you with the actual formulas the Hole Wizard uses to make the processes.

Reading and understanding how Hole Wizard creates processes and sets the values automatically is very helpful but can be difficult. All of the information follows a logical flow you will need to become familiar with if you want to understand what is going on inside the Hole Wizard.

There are four elements that make up the description of how the Hole Wizard sets up a process. The first item presented is the name of an element in the Hole Process dialog, the Process Item. This is the item we are going to explain. An example would be the Entry Clearance value, which consists of the label and text box. The Process Items are listed alphabetically.

The second item is a Group Header. Group Headers are variables that influence how a Process Item is defined. This item is telling you that there are multiple options that are contained in the group and you need to pick one. Each Process Item may have several Group Headers to accurately describe all of the options and variables that influence the Hole Wizard. Group Headers include the type of hole being made, tool types and Entry/Exit Cycles. Think of the Group Headers as questions that need to be answered in order to understand the Process Item.

The third item is a Group Item. Group Items are related to each other but are different choices. Group Items may include the type of hole being made, tool types or Entry/Exit Cycles. So, if you are presented with a list of hole types and you are making a Spot Face hole you would follow this item's logic instead of a bore hole's logic. Think of the Group Items as possible answers to the Group Header's question.

The last type of item you will find is a Formula. The formula mathematically describes how the Hole Wizard comes up with the value for a Process Item. The formula can be thought of as what the question and answer really mean. This is what you've been trying to get to in order to understand the Hole Wizard's logic.

Contained within formulas are several typefaces and items which need further definition. Often these items are cross references. How the items are presented will provide an answer to their purpose.

- {Bracketed} - An item in brackets specifies where the previous item in the formula may be found. Stroke Over {Step Two} says that the referred item (Stroke Over) may be found in Step Two of the Hole Wizard.

- ***Italicized*** - An item in italics is a reference to another process item. When you see ***Surface Z*** you should refer back to that item. That item has an impact on the formula.
- "Quoted" - A quoted item is a link referring to a formula or an item that needs a definition. A glossary and set of formulas may be found at the end of this section on Formulas and Glossary.

Following is the basic layout of the Hole Wizard's logic for each Process Item. Since there can be multiple responses to a question, we need to find the correct question and answer that will give us what we need - how a Process Item's values are set. In each Process Item it is very likely that there will be more than one set of Group Headers and Group Items. A Group Header may have four Group Items, each of which may have one or more indented Group Headers that ask yet more questions. These items become indented to help you keep them organized. Items that line up on the left are a part of the same Group Header, they are all possible answers to the same question. You always work down through the Process item, always trying to get to an indented item. Following this method you will soon reach a formula for the Process Item.

Approach Angle

The Approach Angle will be set to 90° when setting up a Mill Chamfer, Mill Finish or Mill Rough/Pocket process. This item is not used in any other processes.

Bore Diameter

Hole Type	Formula
Mill Chamfer Mill Finish Mill Rough/Pocket	90
Otherwise:	unused

Bore Diameter is used by Mill Chamfer, Mill Finish and Mill Rough/Pocket processes only.

Hole Type	Formula
Mill Chamfer Mill Finish	Bore D + C
Mill Rough/Pocket	Bore D
Otherwise	unused

Clearance

The Clearance value is applied only when the Entry/Exit Cycle is Peck, Full Out. The clearance value is taken from the Hole Data Preferences.

Entry/Exit Cycle Peck, Full Out	Formula Re-entry clearance {HDP}
Otherwise	unused

Clearance Amount

The Clearance Amount value is applied only if a Mill Chamfer, Mill Finish or Mill Rough/Pocket process is being used. The value is taken from the Hole Data Preferences.

Hole Type	Formula
Mill Chamfer Mill Finish Mill Rough/Pocket	Rapid clearance amount {HDP}
Otherwise	unused

Clearance Diameter

The Clearance Diameter value is applied only if a Mill Chamfer, Mill Finish or Mill Rough/Pocket process is being used.

Hole Type	Formula
Mill Chamfer Mill Finish Mill Rough/Pocket	Drill D
Otherwise	unused

Climb/Conventional

Climb or Conventional cut is applied only if a Mill Chamfer, Mill Finish or Mill Rough/Pocket process is being used. The default is to Climb cut.

Hole Type	Formula
Mill Chamfer Mill Finish Mill Rough/Pocket	Climb
Otherwise	unused

Coolant

The coolant is on by default and set to Flood.

Flood

Cutter Radius Compensation On

CRC is applied only if a Mill Chamfer, Mill Finish or Mill Rough/Pocket process is being used and the Hole Data Preferences specifies CRC is to be used.

Hole Type	Formula
Mill Chamfer Mill Finish Mill Rough/Pocket	On if CRC check box is checked {HDP}
Otherwise	unused

Cut Feed

A Cut Feed value is applied only if a Mill Chamfer, Mill Finish or Mill Rough/Pocket process is being created. The value is taken from the Feed process.

Hole Type	Formula
Mill Chamfer Mill Finish Mill Rough/Pocket	*Feed*
Otherwise	unused

Cut Width

A Cut Width value is applied only if a Mill Chamfer, Mill Finish or Mill Rough/Pocket process is being created. The value is equal to the sum of the Hole Data Preferences Cut Width value divided by 100 multiplied by the tool diameter. The value will be 0 for a Mill Chamfer or Mill Finish process.

Hole Type	Formula
Mill Chamfer Mill Finish	0
Mill Rough/Pocket	[(Cut width {HDP} / 100) * tool diameter]

Hole Type	Formula
Otherwise	unused

Desired Z Step

The Desired Z Step value is applied only if a Mill Chamfer, Mill Finish or Mill Rough/Pocket process is being created.

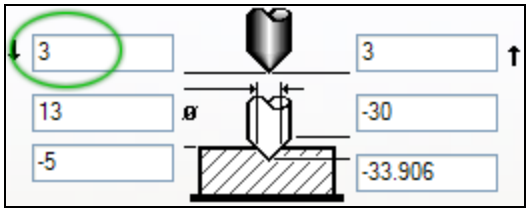
Hole Type	Formula
Mill Chamfer Mill Finish	L
Mill Rough/Pocket	[(Z Step {HDP} / 100) * tool diameter]
Otherwise	unused

Dwell

The Dwell value defaults to whatever value was last used. A new Dwell value is determined only if a Feed In - Rapid Out Entry/Exit Cycle is used for Drill Chamfering, Bore Chamfering or Spot Facing.

Entry/Exit Cycle	Group Header	Group Items	Formula
FI-RO	Hole Type	Drill Chamfer	(Drill Dwell amount (revs) {HDP} * 60) / RPM
		Bore Chamfer Spot Face	(Spot Face Dwell amount (revs) {HDP} * 60) / RPM
Otherwise			Last default

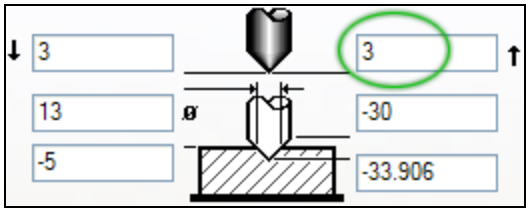
Entry Clearance



The Entry Clearance value is equal to the Clearance Height set in Step Three of the Hole Wizard plus the results of running the *Surface Z* process creation.

Surface Z + Clearance Height {Step Three}

Exit Clearance

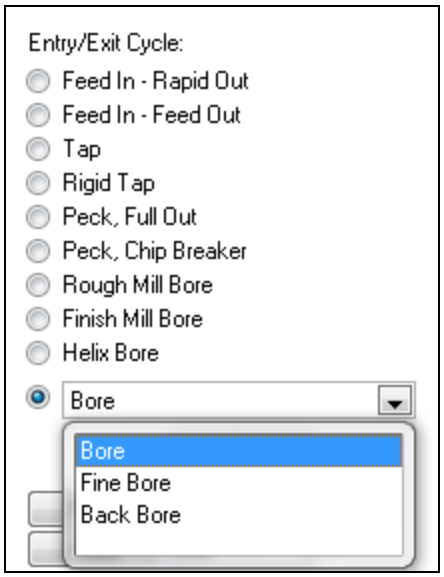


The Exit Clearance value is equal to the Clearance Height set in Step Three of the Hole Wizard plus the results of running the *Surface Z* process creation.

Surface Z + Clearance Height {Step Three}

Entry/Exit Cycle

The Entry/Exit Cycle used is determined by the type of hole being created. Some of the results are straight forward - a Back Bored hole is being made so the Entry/Exit Cycle is set to Back Bore.



Process Type	Tool	Query	Result	Query	Result	Cycle
Back Bore Finish Back Bore Medium Back Bore Rough						Back Bore
Bore Chamfer Ream Finish Ream Medium						FI-FO
Bore Finish						Fine Bore

Process Type	Tool	Query	Result	Query	Result	Cycle
Bore Medium Bore Rough						Bore
Drill		(L / D * 100) > Max. depth for FI-RO {HDP}	No			FI-RO
			Yes	Entry/Exit Cycle = Full Out	Yes	Peck, Full Out
					No	Peck, Chip Breaker
Drill Chamfer Spot Face						FI-RO
Mill Chamfer Mill Finish						Finish Mill Bore
Mill Rough/Pocket						Rough Mill Bore
Spotting	drill	(L / D * 100) > Max. depth for FI-RO {HDP}	No			FI-RO
			Yes	Entry/Exit Cycle = Full Out	Yes	Peck, Full Out
					No	Peck, Chip Breaker
	other					FI-RO
Tapping		Rigid Tap {Step Two}	Yes			Rigid Tap
			No			Tap

Feed

The Feed is automatically calculated in the same manner as clicking on the Feed button in the Drill process dialog. This value is based on the material used.

Finish Entry/Exit 90°

The size of the 90° Radius used for Finish Entry/Exit moves is determined only if a Mill Chamfer, Mill Finish or Mill Rough/Pocket process is being created. The value will be 0 for a Mill Rough/Pocket process. For a Mill Chamfer or Mill Finish process the value is equal to the sum of the Hole Data Preferences Approach Exit radius value divided by 100, multiplied by the tool diameter.

Hole Type	Formula
Mill Chamfer: Mill Finish	[(Approach Exit radius {HDP} / 100) * tool diameter]
Mill Rough/Pocket	0
Otherwise	unused

Finish Entry/Exit Min. Line

The size of the Min. Line used for Finish Entry/Exit moves is determined only if a Mill Chamfer, Mill Finish or Mill Rough/Pocket process is being created. The value will be 0 for a Mill Rough/Pocket process. For a Mill Chamfer or Mill Finish process the value is equal to the sum of the Hole Data Preferences Approach Exit line value divided by 100, multiplied by the tool diameter.

Hole Type	Formula
Mill Chamfer Mill Finish	[(Approach Exit line {HDP} / 100) * tool diameter]
Mill Rough/Pocket	0
Otherwise	unused

Overlap

The Overlap size is determined only if a Mill Chamfer, Mill Finish or Mill Rough/Pocket process is being created. The value will be 0 for a Mill Rough/Pocket process. For a Mill Chamfer or Mill Finish process the value is equal to the sum of the Hole Data Preferences Overlap value divided by 100, multiplied by the tool diameter.

Hole Type	Formula
Mill Chamfer Mill Finish	[(Overlap {HDP} / 100) * tool diameter]
Mill Rough/Pocket	0
Otherwise	unused

Peck

A Peck values is determined only if the Entry/Exit Cycle is set to Peck, Chip Breaker or Peck, Full Out. The Peck value is equal to the hole's diameter multiplied by the Peck Depth set in the Hole Data Preferences.

Entry/Exit Cycle	Formula
Peck, Chip Breaker Peck, Full Out	D * Peck depth {HDP}
Otherwise	unused

Positive Approach

Positive Approach is not active by default for all processes.

false

Retract

A Retract value is determined only when the Entry/Exit Cycle is set to Peck, Chip Breaker. The value is equal to the hole's diameter multiplied by the Retract Amount set in the Hole Data Preferences.

Entry/Exit Cycle	Formula
Peck, Chip Breaker	D * Retract Amount {HDP}
Otherwise	unused

Retract to Z

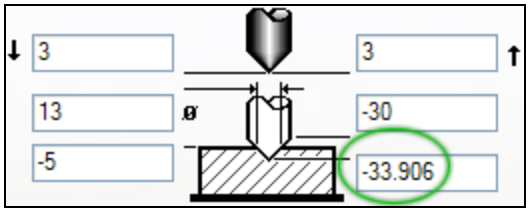
The value for the second Retract to Z option is equal to the Clearance Height set in Step Three of the Hole Wizard plus the results of determining the *Surface Z.*

Surface Z + Clearance Height {Step Three}

RPM

The RPM is automatically calculated in the same manner as clicking on the RPM button in the Drill process dialog. This value is based on the material used.

Sharp Tip Z



The Sharp Tip Z value is the most complex of the process creation items. The method for determining the Sharp Tip Z value first depends on the type of process being made. Secondly, the type of tool being used and/or the type of hole being made can influence which formula is used. Lastly, whether the process will be chamfering with the tool tip or not will influence which formula is used to determine the Sharp Tip Z. The Surface Z process is common to all process types. This value will need to be calculated first for any process type.

Process Type	Tool Type	Hole Type	Query	Result	Formula
Back Bore Finish Back Bore Medium Back Bore Rough		Back Bore			***Surface Z*** + (L + Tip lead {Tool dialog} + Stroke Over {Step Two})
		Bore			***Surface Z*** + (L + Tip lead {Tool dialog})
		Bore, Thru			***Surface Z*** + (Stock size + Tip lead {Tool dialog} + Stroke Over {Step Two})
Bore Finish Bore Medium Bore Rough		Bore			***Surface Z*** - (L + Tip lead {Tool dialog})
		Bore, Thru			***Surface Z*** - (Stock size + Tip lead {Tool dialog} + Stroke Over {Step Two})
		Back Bore			***Surface Z*** - (L +

Process Type	Tool Type	Hole Type	Query	Result	Formula Tip lead {Tool dialog} + Stroke Over {Step Two})
Drill		Back Bore Bore, Thru			*Surface Z*- (taper depth + L + Stroke Over {Step Two})
		Spot Face			*Surface Z*- (L - 0.5)
		Otherwise			*Surface Z*- (taper depth + L)
Bore Chamfer Drill Chamfer Mill Chamfer	Center Drill	Bolt Hole	Chamfering with tip	No	*Surface Z*- (tool flute length + taper depth + Chamfer Depth + SFD)
				Yes	*Surface Z*- (Chamfer Depth (PD=0) + SFD)
		Otherwise	Chamfering with tip	No	*Surface Z*- (tool flute length + taper depth + Chamfer Depth)
				Yes	*Surface Z*- Chamfer Depth (PD=0)
	Counter Sink	Bolt Hole			*Surface Z*- (Chamfer Depth + SFD)
		Otherwise			*Surface Z*- Chamfer Depth
	Drill Spot Drill	Bolt Hole			*Surface Z*- (Chamfer Depth (PD=0) + SFD)

Process Type	Tool Type	Hole Type	Query	Result	Formula
		Otherwise			***Surface Z*** - Chamfer Depth (PD=0)
	Finish End Mill	Bolt Hole			***Surface Z*** - (EM taper depth + Min. chamfer tool oversize {HDP} + SFD)
	Otherwise				***Surface Z*** - (EM taper depth + Min. chamfer tool oversize {HDP})
Mill FinishMill Rough/Pocket		Bore, Thru			*Surface Z* - Stock size
		Otherwise			*Surface Z* - L
Ream FinishReam MediumTapping					*Surface Z* - (Tap depth {Step Two} + Tip lead {Tool dialog})
Spotting	drill				***Surface Z*** - (taper depth + L)
	drill &/or chamfer	Bolt Hole			***Surface Z*** - (taper depth + L + SFD)
		Otherwise			***Surface Z*** - (taper depth + L)
	spot				***Surface Z*** - Preferred spot depth {HDP}
	spot &/or chamfer	Bolt Hole	Center Drill		***Surface Z*** - (tool flute length + taper depth + Chamfer Depth + SFD)

Process Type	Tool Type	Hole Type	Query	Result	Formula
			Otherwise		*Surface Z* - (Chamfer Depth + SFD)
		Otherwise	Center Drill		*Surface Z* - (tool flute length + taper depth + Chamfer Depth)
			Otherwise		*Surface Z* - Chamfer Depth
Spot Face					*Surface Z* - Spot Face Depth {Step Two}

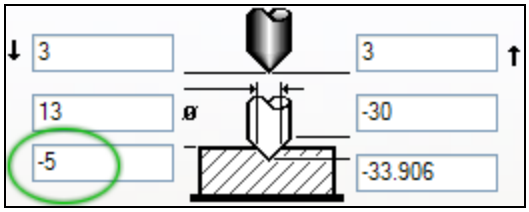
Spring Passes

Hole Type	Query	Result	Formula
Mill Chamfer Mill Finish	Finish stock to remove > Max. fin. Cut, no spring {HDP}	Yes	1
		No	0
Mill Rough/Pocket			0
Otherwise			unused

Stock

Hole Type	Formula
Mill Chamfer Mill Finish	0
Mill Rough/Pocket	Finish stock to remove
Otherwise	unused

Surface Z



Group Header	Group Items	Formula
Process Type	Back Bore Finish Back Bore Medium Back Bore Rough	Stock surface value {Step Three} - Stock size - Stroke Over {Step Two}
	Otherwise	Stock surface value {Step Three}

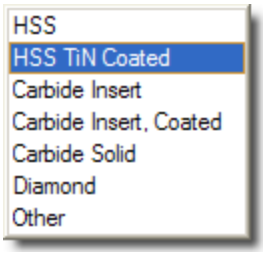
Tap%

Entry/Exit Cycle	Formula
Tap	Tap feed rate percent {HDP}
Rigid Tap	100%
Otherwise	unused

Formulas and Glossary

a	1/2 angle
	If (hole diameter <= Min. dia. for allow. 2 {HDP})
BFMaxAllowance	Bore finish max allow 1 {HDP}
	else Bore finish max allow 2 {HDP}
	If (hole diameter <= Min. dia. for allow. 2 {HDP})
BFMinAllowance	Bore finish max allow 1 {HDP}
	else Bore finish max allow 2 {HDP}

	If (hole diameter <= Min. dia. for allow. 2 {HDP})
BMMaxAllowance	Bore medium max allow 1 {HDP}
	else Bore medium max allow 2 {HDP}
	If (hole diameter <= Min. dia. for allow. 2 {HDP})
BMMinAllowance	Bore medium max allow 1 {HDP}
	else Bore medium max allow 2 {HDP}
C	Chamfer * 2.0
CD	C+ D ([Chamfer * 2.0] + [Hole diameter])
	The Chamfer Depth is the distance the tool needs to descend to create the desired chamfer.
Chamfer Depth	[(hole diameter + chamfer diameter – pilot diameter) / 2.0] * tan (90 – chamfer angle) which can also be described as: (CD-PD)/2 * tan(90-a)
D	Hole diameter
EM taper depth	= C/2 * tan(90-a)
Finish stock to remove	D < Min dia. for Big Bore {HDP}? **Yes** = Small Bore Stock Allowance: Finish maximum {HDP} **No** = Big Bore Stock Allowance: Finish maximum {HDP}
HDP	Hole Data Preferences
L	Length
Last default	Whatever value last used for that item
PD	Tool pilot diameter
SFD	Spot Face diameter
Standard Comparison	Assuming all else is equal, this is the final decision maker in which tool will be used in a process. It works through three steps. As soon as one tool is left, that is the one to be used. The steps are: 1) Check the tool material. The best material is used. HSS is lowest, Diamond or Other is highest. The tool material pull-down menu shows all of the choices, from least desirable to best. If the tools are made of the same material the check progresses to Step Two. 2) Check if the tool was used in the previous operation. If neither of the tools fits this criteria we



then move on to Step Three.

3) Use the lowest tool number. This means basically you've got two or more tools that qualify and we'll just "pick one".

taper depth = D/2 * tan(90-a)

Hole Type determination

GibbsCAM recognizes Hole Features from other systems, such as SOLIDWORKS, Solid Edge, Inventor, and Catia. When holes are imported, they are mapped to hole types as specified in the following tables. Holes that cannot be mapped will be labeled as "Compound" holes, and the Compound Hole Editor must be used to modify them.

Throughout:

- If TPI/Pitch is not mentioned, it is undefined.
- Chamfer width should be taken from the same segment as chamfer angle.
- Depth is always measured to the end of the last segment (including tip).
- Full Depth is always measured to the end of the last segment (excluding tip).

Drill	One segment	Two segments (chamfered)
Segment 1	Straight, non-threaded "Drill"	Tapered, non-threaded
Segment 2		Straight, non-threaded "Drill"
1.Diameter	Segment 1	Segment 2.
2.Diameter 2		
3.Mid Depth		
4. Chamfer Angle		Segment 1 taper angle

Tap	One segment	Two segments	Two segments (non-chamfered)	Three segments
Segment 1	Threaded	Tapered	Threaded	Tapered, non-threaded
Segment 2		Threaded	Straight	Threaded
Segment 3				Straight, non-threaded "Drill"
1.Diameter	Segment 1	Segment 2	Segment 2	Segment 3
2.Diameter 2			Segment 1	Segment 2
3.Mid Depth			End of Segment 1	End of Segment 2
4. Chamfer Angle		Segment 1 taper angle		Segment 1 taper angle
5.TPI/Pitch	TPI/Pitch of Segment 1	TPI/Pitch of Segment 2	TPI/Pitch of Segment 1	TPI/Pitch of Segment 2

Ream	One segment	Two segments	Two segments (non-chamfered)	Three segments

Segment 1	Non-threaded "Ream"	Tapered, non-threaded "Ream"	Non-threaded "Ream"	Tapered, non-threaded "Ream"
Segment 2		Non-threaded "Ream"	Straight, non-threaded "Drill"	Non threaded "Ream"
Segment 3				Straight, non-threaded "Drill"
1.Diameter	Segment 1	Segment 2	Segment 2	Segment 3
2.Diameter 2			Segment 1	Segment 2
3.Mid Depth			End of Segment 1	End of Segment 2
4. Chamfer Angle		Segment 1 taper angle		Segment 1 taper angle

Bolt hole	Two segments	Three segments
Segment 1	Non threaded, straight "Drill"	Non-threaded, straight "Drill"
Segment 2	Non-threaded, straight "Drill"	Tapered, non-threaded
Segment 3		Non-threaded, straight "Drill"
1.Diameter	Segment 2	Segment 3
2.Diameter 2	Segment 1	Segment 1
3.Mid Depth	End of Segment 1	End of Segment 1
4. Chamfer Angle		Segment 2 taper angle.

The Diameter of the first segment must be larger than the diameter of the last segment.

Spot Face	Two segments
Segment 1	Non-threaded, tapered
2nd segment	Non-threaded, straight "Drill"
1.Diameter	Segment 2
2.Diameter 2	
3.Mid Depth	
4. Chamfer Angle	Segment 1 taper angle

Bore	Two segments	Three segments
Segment 1	Straight, non-threaded "Bore"	Tapered, not threaded
Segment 2	Straight, non-threaded "Drill"	straight, non-threaded "Bore"
Segment 3		Straight, non-threaded "Drill"

1.Diameter	Segment 2 diameter	Segment 3 diameter
2.Diameter 2	Segment 1 diameter	Segment 2 diameter
3.Mid Depth	End of Segment 1	End of Segment 2
4. Chamfer Angle		Segment 1 taper angle.

Bore Through	One segment	Two segments
Segment 1	Straight, non-threaded “Bore”	Tapered, not threaded.
Segment 2		Straight, non-threaded “Bore”
1.Diameter	Segment 1 diameter	Segment 2 diameter
2.Diameter 2		
3.Mid Depth		
4. Chamfer Angle		Segment 1 taper angle.

Back Bore	Two segments
Segment 1	Straight and non-threaded
Segment 2	Straight, non-threaded “Bore” The diameter of the first segment must be LESS than the diameter of the second segment.
1.Diameter	Segment 2 diameter
2.Diameter 2	Segment 1 diameter
3.Mid Depth	End of Segment 1
4. Chamfer Angle	

Compound	Anything not met by one of the above definitions

Conventions

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

Text

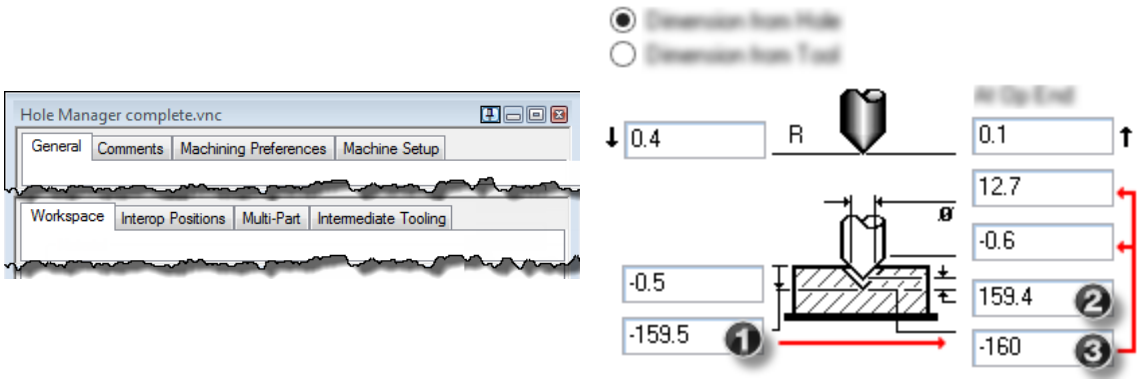
Screen text. Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

`Keystroke/Mouse`. Text with `this appearance` indicates a keystroke or mouse action, such as `Ctrl+C` or `right-click`.

`Code`. Text with `this appearance` indicates computer code, such as lines in a macro or a block of G-code.

Graphics

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



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

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

Links to Online Resources

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

Index

D

E

F

G

H

I

K

L

M

N

O

P

R

S

T

U

W

Z