GIBBSCAM® 14

Version 14 : September 2020

Broaching

CAMBRIO

Proprietary Notice

Contents

Introduction to Broaching

Broaching is used for high quality precision machining of non-circular shapes, such as hexagons or keyways. GibbsCAM supports both Linear (Oriented) and Rotary (Face or wobble) broaching

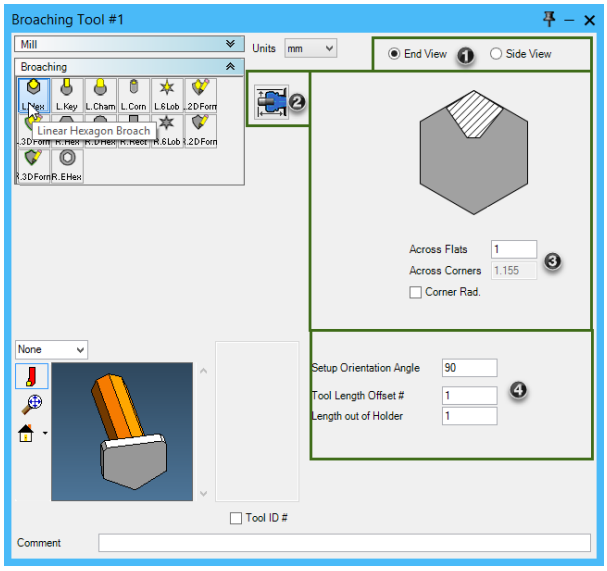
LicensingIf you have Turning, you automatically have **Face Broaching**. If you have **Advanced CS**, you automatically have **Oriented Broaching**.

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

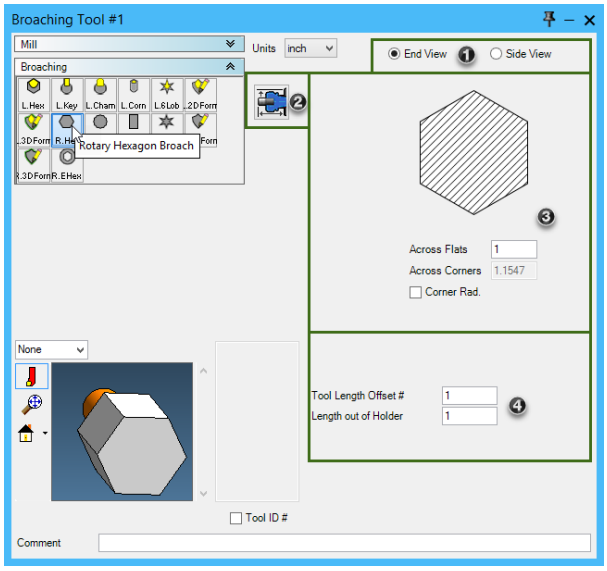
Tools for Broaching

Broaching Tool Dialogs

Linear Broaching Tool



Rotary Broaching Tool



Controls in Broaching Tool Dialogs

1. Enter the correct values in both the End and Side views of a broaching tool to fully define the tool.
2. Enter details of your Toolholder and Toolblock, if used. The system will calculate holder offsets using this data. For more information on offsets, see the see the *Mill* guide's discussion of Mill Tool Offset Data.

On turning machines, a broaching tool requires a special toolholder that allows it to rotate in sync with the workpiece. Different broaching tools are used for ID and OD broaching.

3. Define the Tooltype-specific values (see tools). The tool diagram and specifications depend on the tool type selected. The shaded sections of a tool diagram illustrate the cutting surfaces of a tool. The grey areas are non-cutting surfaces of the tool. If these surfaces come into contact with a part, the system draws this contact area in red during rendering to show interference.
4. Specify Linear/Rotary broaching general parameters as follows:

Setup Orientation Angle (Linear only)
Enter an orientation angle if required.

Tool Length Offset number
This number designates the numeric location in the machine where the Z offset amount is entered.

Length out of Holder
If there is a tool holder, this is the tool length measured from the front end of the holder.

If there is no holder, enter the tool length measured from the tool attachment point.

Parameters for 2D and 3D Form Tools

Nominal Diameter
Enter the overall diameter of the tool. If the shape is to be duplicated, this would be the diameter of the final tool.

Tip Offset Distance
This is the distance between the shank center and the tool tip.

Duplication Type
In order to create custom tools, the form tool shape can be repeated around a circle. Select the duplication required:

None
No duplication.

Full Circle
Creates equally spaced duplicates around a full 360 degrees. Enter the Total number of duplicates.

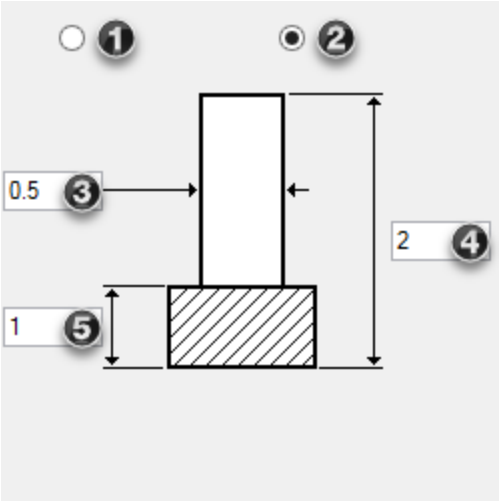
Partial Circle
Enter the number of duplicates.

Enter the angle between each shape. The angle can be positive or negative. A positive value duplicates in a counterclockwise direction, a negative value in a clockwise direction.

Broaching Tool Definition

To fully define a broaching tool, use the two radio buttons: One defines the side view of the tool. The other to define the bottom (cutting area of the tool).

Side and Shank Definition



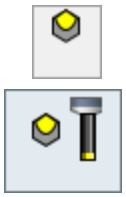
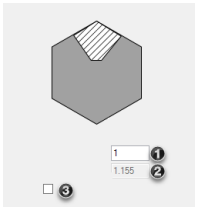
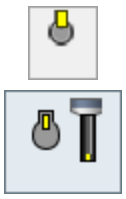
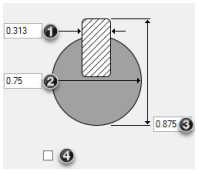
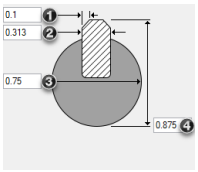
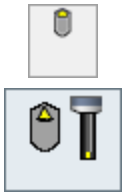
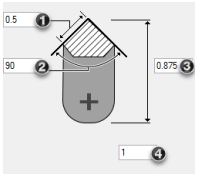
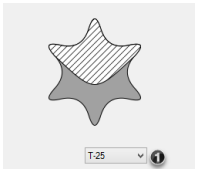
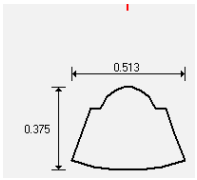
1. End view of broaching tool (tool-specific values
2. Side view of broaching tool
3. Shank diameter
4. Total tool length
5. Cut length

A broaching tool is created by extruding the cut end shape by the cut length amount along the tool axis. A cylindrical shank is then added. This dialog enables the shank and side view of the tool to be defined.

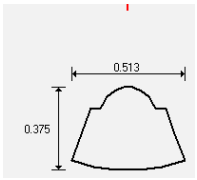
Linear Broaching Tools

GibbsCAM v14 supplies templates for seven linear broaching tools.

Cutting end definition for linear broaching tools

icons	*Name*	*End View*	*Notes*
	Linear Hexagon Broach		1. Dimension across Flats 2. Across Corners (calculated) 3. Check to add Corner radius
	Linear Keyway Broach		1. Width 2. Side to side diameter 3. Top to bottom Height 4. Check to add Corner radius
	Linear Chamfer Broach		1. Side length of Chamfer 2. Width of keyway 3. Side to side diameter 4. Top to bottom Height
	Linear Corner Broach		1. Length of edge 2. Included angle (degrees) 3. Top to bottom Height 4. Distance between shank center and Tool tip
	Linear Hexalobular Broach		1. Tool size
	Linear 2D Form Broach		Custom dimensions

Linear 3D Form Broach

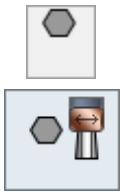
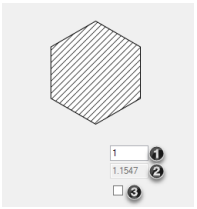
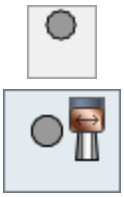
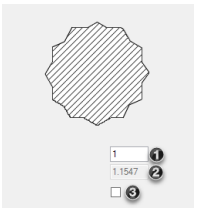
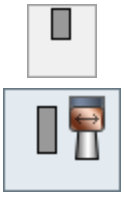
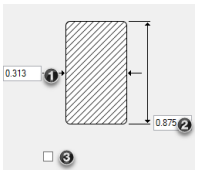
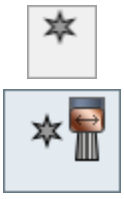
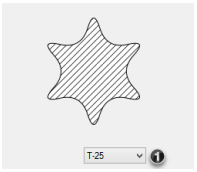
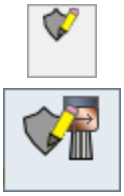
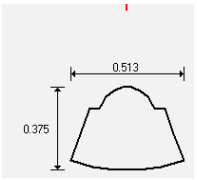
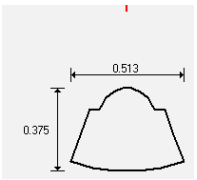


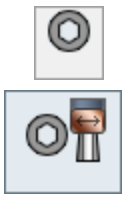
Custom dimensions

Rotary Broaching Tools

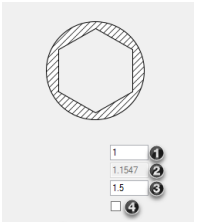
GibbsCAM v14 supplies templates for seven rotary broaching tools.

Cutting end definition for rotary broaching tools

icons	*Name*	*End View*	*Notes*
	Rotary Hexagon Broach		1. Diameter measured across flats 2. Across Corners (calculated) 3. Check to add Corner radius
	Rotary Double Hexagon Broach		1. Diameter measured across flats 2. Across Corners (calculated) 3. Check to add Corner radius
	Rotary Rectangle Broach		1. Side to side width 2. Top to bottom Height 3. Check to add Corner radius
	Rotary Hexalobular Broach		1. Tool size
	Rotary 2D Form Broach		Custom dimensions
	Rotary 3D Form Broach		Custom dimensions

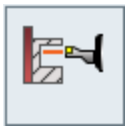


Rotary External Hexagon



1. Internal diameter measured across flats
2. Across corners (calculated)
3. External diameter
4. Check to add Corner radius

Broaching Processes

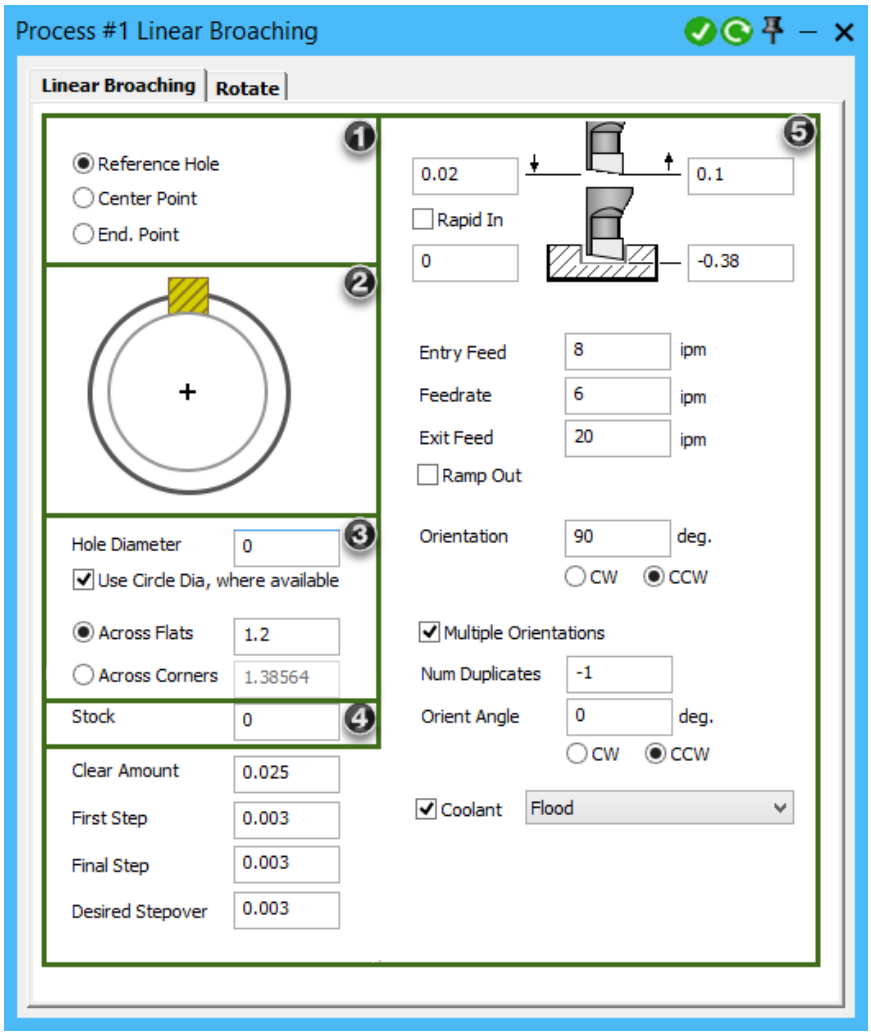


Linear Broaching Processes

Linear Broaching processes cut a small amount of material at a time by pushing the tool into the material. Many passes are usually required to create the desired shape.

Linear Broaching Process Diagram

When a Linear broaching function tile is combined with a linear tool from the Broaching tool list, a process dialog will appear. The exact dialog will differ with the tool selected and the strategy chosen. The illustration diagram will also change to provide information on input required for Clear Amount, First and Final Step, and Desired Stepover.



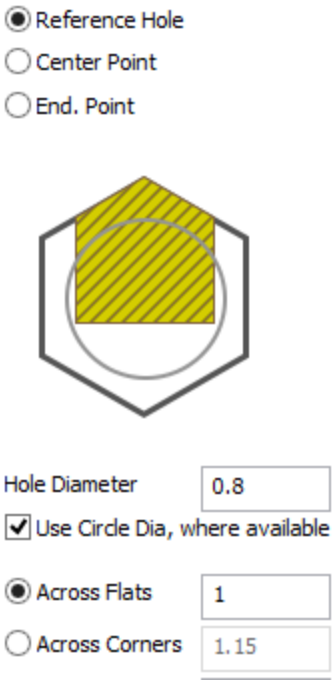
1. Strategy
2. *(illustration; varies with strategy)*
3. *(strategy-specific controls)*
4. Stock control
5. Controls common to all strategies

Strategies and Illustrations

Three strategies are provided for linear broaching in order to cater for different tool types and cutting scenarios.

Please view the sample files provided to see examples of broaching strategies.

Reference Hole



Reference hole options

Options and diagrams change with type of Tool selected.

Hole Diameter (All tools)
Enter the diameter of the start hole. This is used as a reference for the clearance and first pass positions.

Use Circle diameter where available (All tools)
Check this box to use the diameter of selected circles for the start hole diameter.

Adjust for Chord Height (Key, Cham)
When checked, adjusts the clearance position and first pass to allow for the chord height between the flat edge of the tool and the edge of the start hole. This is used for Keyway tools.

T Dimension (Key, Cham)
Distance from center to keyway flat.

Across Flats/Across Corners (Hex)
Used to calculate the final diameter.

Final Diameter (Com, Lob, Form tools)

Reference Hole Strategy

This is the preferred strategy for Linear Hexagon, Chamfer and Hexalobular tools, but it may also be used with Keyway tools. Your tool data should include a manufacturers' recommended pre-drilled hole dimension. This hole should be large enough to fit the tool into, but smaller than the final broached shape. We recommend that you draw a circle of this pre-drilled diameter.

First, create a process to pre-drill the hole, then create your linear broaching process. Select the circle for this process group. On the broaching process dialog, check the Use Circle Diameter where available option. If you did not create a circle, then select a point at the center of the hole. Make sure the Use Circle Diameter option is un-checked, and enter the pre-drilled diameter on the broaching dialog.

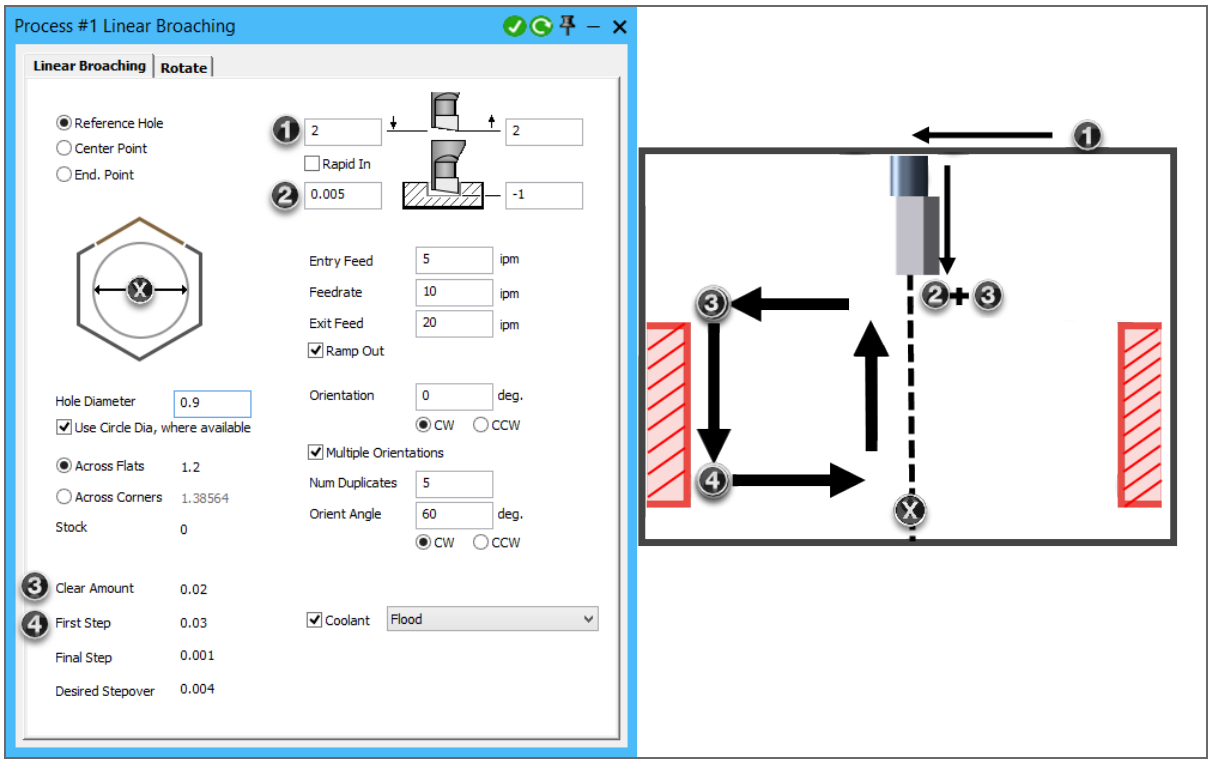
Next, specify the final size to be broached. For a hexagon tool this can be specified as the distance Across flats, or across the sharp corners. For other tools it will be the finished diameter.

The stock dimension can be used to adjust the final size: A negative stock amount will broach the part under size. A positive amount will cut it oversize.

The Clear Amount is the distance from the pre-drilled diameter where you can rapid to before starting to broach. A positive value will position the tool inside the pre-drilled hole.

The First Step value is the distance outside of the pre-drilled diameter where you want to make the first broaching stroke.

The Desired Stepover value is an incremental distance between each broaching stroke, and the Final Step value is the amount of material to be removed by the final stroke.



For the broaching process, the tool will move across to the hole center, rapid down to the Entry Clearance height Z (if it is above this value in Z). It will then move down to the start Z depth (either at feed or rapid, according to the Rapid In checkbox). This will be followed by a rapid move across to the Clear Amount (3).

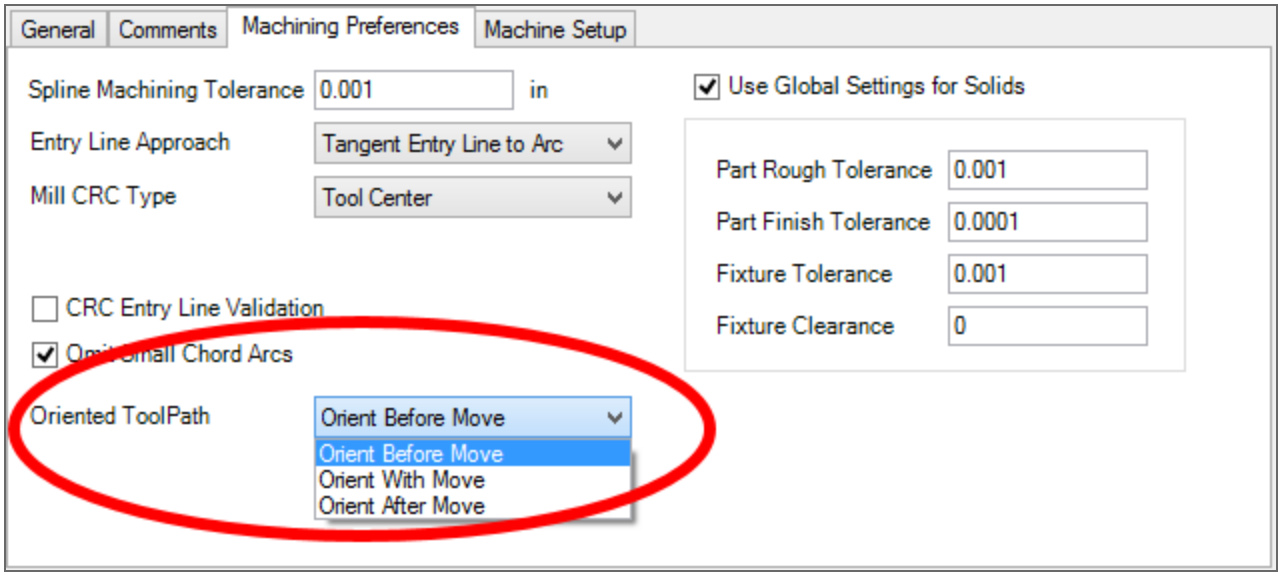
There will now be a series of 4 moves for each broaching stroke. It will then move to (2) the hole center and down to the (3) start position. It will complete the first cut (4) and return to the clear amount. The first cut is important as the hole may not be exactly the correct size. The next cut will be the Desired Stepover amount, the exact size of which will be calculated in conjunction with the first and final steps, dividing the distance and rounding it to make a whole number of steps. This stepover cut will be repeated until the tool reaches the final finishing cut just short of the Across Corners measurement of the final diameter, where it cuts the Final Step amount. After the final stroke, the tool will retract in Z to the exit clearance height.

The broaching process dialog requires an orientation angle, where zero degrees will result in the tool pointing along the X axis direction. The tool definition includes a setup angle. If this setup angle

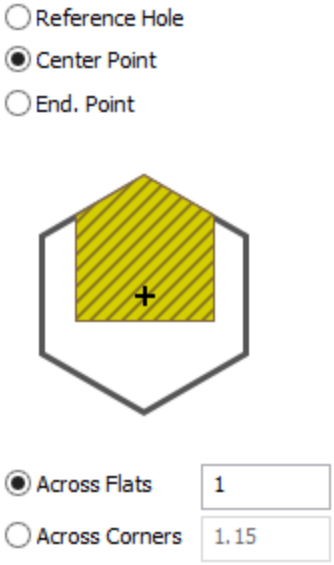
does not match the broaching orientation, then either the tool or the part will be rotated to achieve the required alignment. (Your MDD will control which is rotated.) If your machine does not have any axes that rotate around the Z direction, then the broaching orientation and tool setup angle must be the same, because no rotation is possible.

For machines that are able to rotate the tool relative to the part, you can define a series of broaching strokes at different orientations (for example, to create a complete hexagon). Check the Multiple Orientations option and enter the number of duplicates together with the required angle.

Note: When the toolpath requires a change of orientation, this rotation can be output either before a linear move, with a move, or after it. Your choice between these three options is set in the DCD **Machining Preferences** tab, using the Oriented Toolpath pull-down menu.



Center Point



Center point options

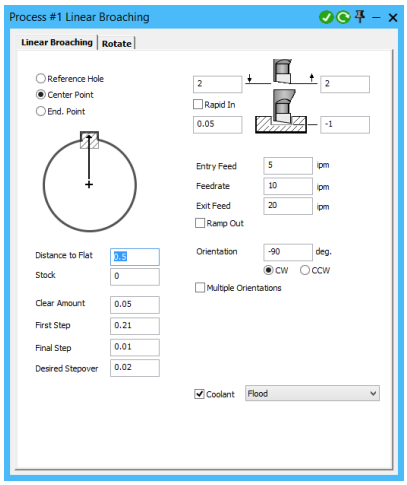
Across Flats/Corners (Hex)
Distance across flats/corners used to calculate the final diameter.

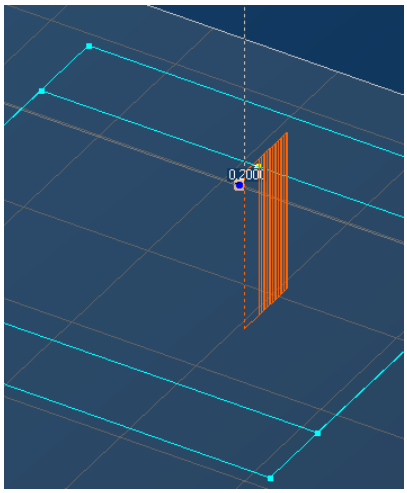
Distance to Flat (Key Cham)
Distance from center to keyway flat

Final diameter (Com, Lob, Form tools)

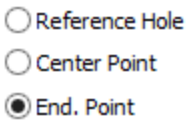
Center point strategy

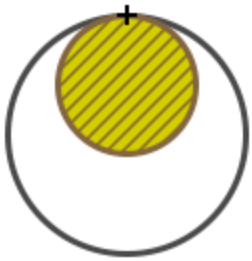
Center point strategy might possibly be more appropriate for Keyway tools as you may not be cutting into a circle. You pick the center point and the known distance is from this point to the top of the keyway.





End Point





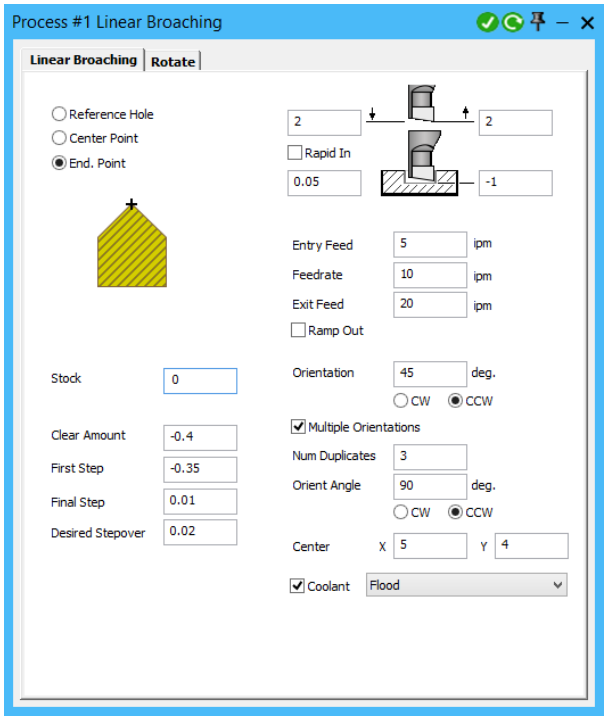
End point options

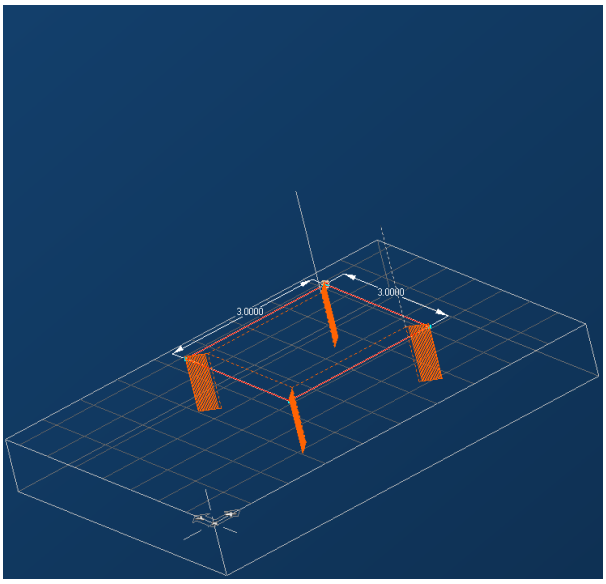
No tool-specific options required.

All dimensions entered would be negative as you are working back from the known end point.

End point Strategy

End point Strategy would be used primarily for cutting sharp corners with Corner Tools. Pick a point to be machined, enter the clear amount (negative distance) and the First Step, also a negative amount. Multiple Orientations can be programmed, but will only work with a regular shape, such as a square or hexagon, because a center must be defined as a rotation point. Below is an example of broaching using a corner tool.





Stock Controls

Stock
Amount of stock to leave after the final pass

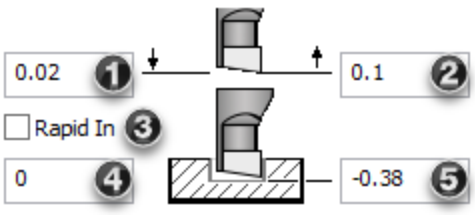
Clear Amount
Clearance position measured as a distance from the end point

First Step
Position of the first broaching pass measured as a distance from the end point

Final Step
Amount to remove by the final broaching pass.

Desired Stepover
Distance between each broaching pass.

Common Controls



1. Entry clearance plane
2. Exit clearance plane
3. Rapid in feature
4. Start position
5. End position

Entry clearance plane

Tool will move to this Z level before beginning the operation.

Exit clearance plane
Tool will move to this Z level when the operation is complete.

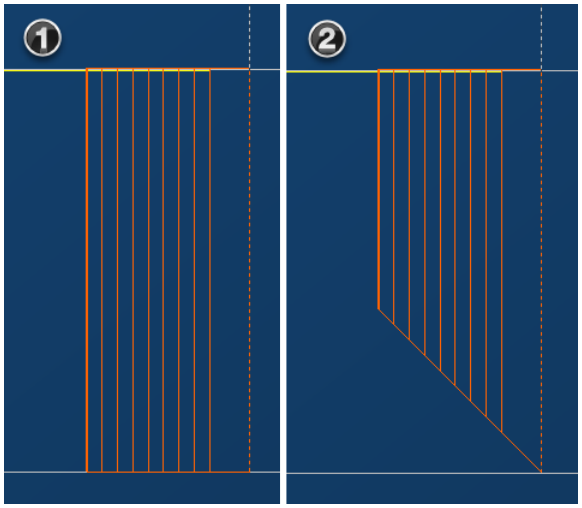
Rapid in feature
Check this box if you require the tool move to be a rapid instead of a feed move from the clearance area. Use caution with this option as the rapid move will be made directly into the part material.

Start/End position
The start and end positions are measured along the depth axis of the machining CS.

Entry/Exit Feed
Feed speed.

Feedrate
Broaching Feedrate per minute.

Ramp out
When checked, exit at 45 degrees.



1. Unchecked
2. Checked

Orientation
Tool orientation angle.

CW/CCW
Check to indicate if the orientation is measured clockwise (CW) or counterclockwise (CCW) from the 12 o'clock position.

Multiple orientations
If duplicate moves are required, check this box and indicate the orientation angle, and if this is measured CW or CCW from 12 o'clock.

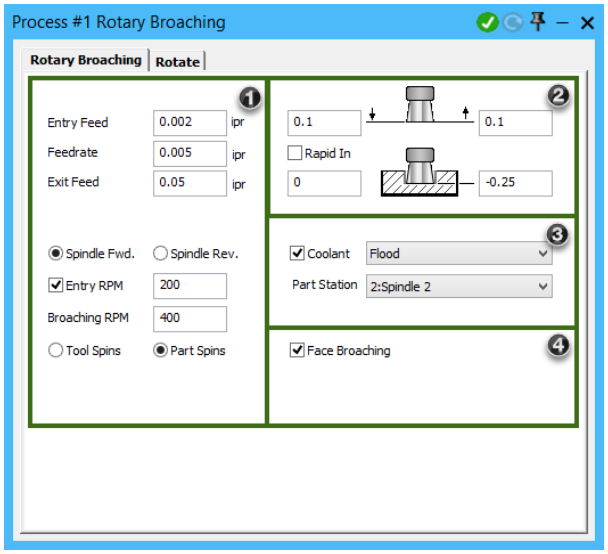
Rotary Broaching Process

In rotary broaching, a broaching tool is held in a toolholder that allows the tool to rotate inside it. The tool is also held with a slightly misaligned axis of rotation so as to create a slight wobble as it rotates.

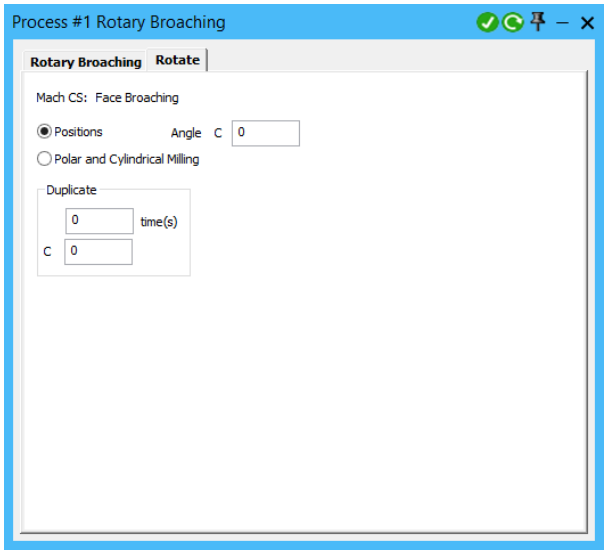
Internal or external broaching of a part is possible. Internal broaching requires a starting hole or opening in the workpiece so that the broaching tool can be pushed or pulled through. The broach is completed in just one pass. The entry point is totally random and the process is therefore best suited to cylindrical parts.

Rotary Broaching Process Dialog

Broaching tab



Rotary tab



1. Feeds and Speeds
2. Clearance Diagram
3. Machine controls
4. Face Broaching

For details, see the *Mill* guide's discussion of the **Rotate** tab.

When cutting an internal broach, a pilot hole should be pre-drilled. It is also advisable to cut a 90-degree countersink at the start of the hole that is greater than the across-point (OD) of the broach. If necessary the part should be made longer and faced off after broaching.

For external forms, a 90-degree chamfer should be added that is less than the ID of the feature being broached. Again, this can be faced off afterwards.

Feeds and Speeds

Entry/Exit Feed
Feedrate as defined in units per revolution.

Feedrate
Broaching Feedrate in units per revolution.

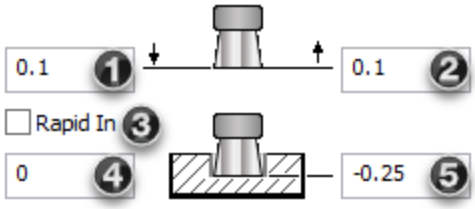
Spindle Fwd/Rev
Click the required radio button.

Entry RPM
Check to specify a different RPM for the entry spindle speed.

Broaching Spindle Speed

Tool/Part Spins
Check the radio button as required.

Clearance Diagram



1. Entry clearance plane
2. Exit clearance plane
3. Rapid in feature
4. Start position
5. End position

Entry clearance plane
Tool will move to this Z level before beginning the operation.

Exit clearance plane
Tool will move to this Z level when the operation is complete.

Rapid in feature
Check this box if you require the tool move to be a rapid instead of a feed move from the clearance area. Use caution with this option as the rapid move will be made directly into the part material.

Start/End position
The start and end positions, measured along the depth axis of the machining CS.

Machine controls

Coolant
Specify the coolant method, `Flood` or `Through spindle` if required.

Part Station (Appearance depends on MDD used)
Select the part station to machine on. Selection here will set all processes in the process list, because they must all machine the same part.

Machining CS (Appearance depends on MDD used)

Select the Coordinate System to be used when generating toolpath for this process. Output of all operations will be based on the specified CS. This CS will determine tool and rotary head orientation.

Face Broaching

Check box to create a Face Broaching process.

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

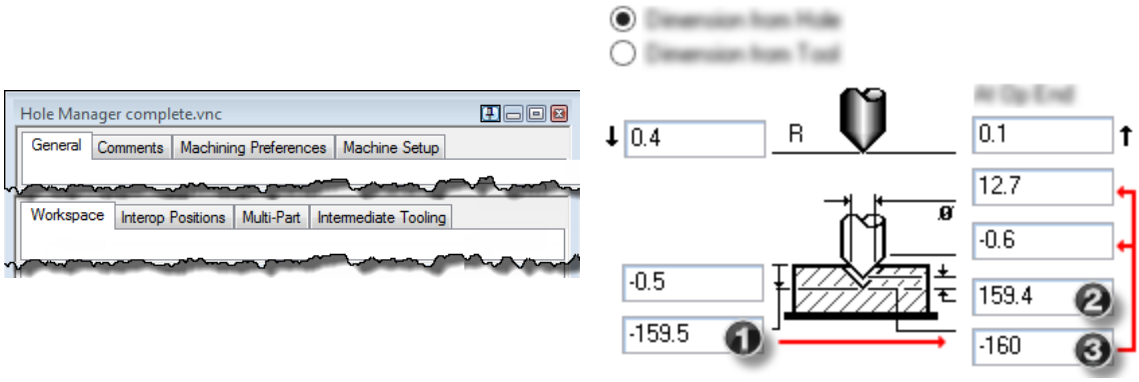
`Screen text`. Text with `this appearance` indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

`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.