



GIBBSCAM 2025 CAM for
Production Machining

Version 2025 : September 2024

Broaching



GIBBSCAM

Contents

| | |
|---|----|
| Introduction to Broaching | 3 |
| Tools for Broaching | 3 |
| Broaching Tool Dialogs | 3 |
| Controls in Broaching Tool Dialogs | 3 |
| Parameters for 2D and 3D Form Tools | 4 |
| Broaching Tool Definition | 5 |
| Side and Shank Definition | 5 |
| Linear Broaching Tools | 6 |
| Cutting end definition for linear broaching tools | 6 |
| Rotary Broaching Tools | 8 |
| Cutting end definition for rotary broaching tools | 8 |
| Broaching Processes | 10 |
| Linear Broaching Processes | 10 |
| Linear Broaching Process Diagram | 10 |
| Strategies and Illustrations | 11 |
| Reference Hole | 11 |
| Reference Hole Strategy | 11 |
| Center Point | 14 |
| Center point strategy | 14 |
| End Point | 15 |
| End point Strategy | 15 |
| Stock Controls | 16 |
| Common Controls | 16 |
| Rotary Broaching Process | 18 |
| Rotary Broaching Process Dialog | 18 |
| Feeds and Speeds | 19 |
| Clearance Diagram | 19 |
| Machine controls | 19 |
| Face Broaching | 20 |

| | |
|-------------------|----|
| Conventions | 21 |
| Text | 21 |
| Graphics | 21 |

| | |
|---------------------------------|----|
| Links To Online Resources | 22 |
|---------------------------------|----|

Introduction to Broaching

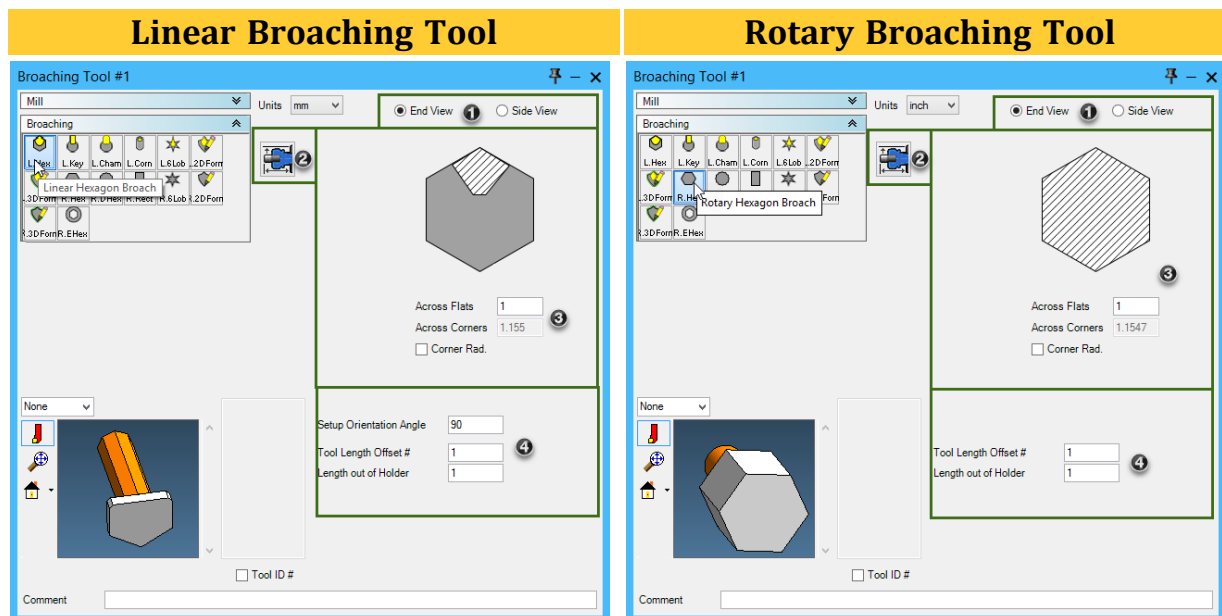
Broaching is used for high-quality precision machining of non-circular shapes, such as hexagons or keyways. GibbsCAM supports both Linear (Oriented) and Rotary (Face or wobble) broaching.

Licensing If 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



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 [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

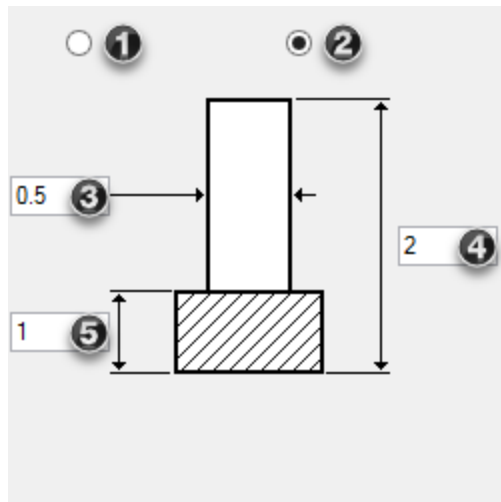
Enter the number of duplicates.

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




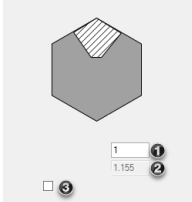


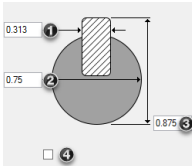


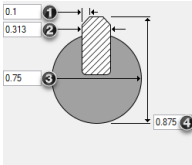


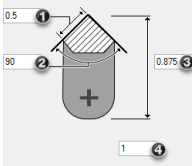
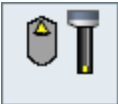

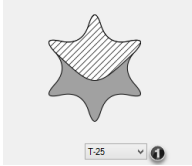


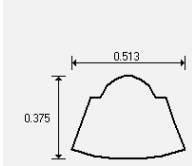

1. End view of broaching tool (tool-specific values)
2. Side view of broaching tool
3. Shank diameter
4. Total tool length
5. Cut length

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

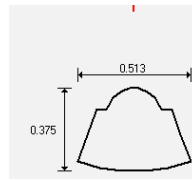
GibbsCAM v2025 supplies templates for seven linear broaching tools.

Cutting end definition for linear broaching tools

| icons | Name | End View | Notes |
|---|---------------------------|---|--|
|  | Linear Hexagon Broach |  | <ol style="list-style-type: none"> 1. Dimension across Flats 2. Across Corners (calculated) 3. Check to add Corner radius |
|  | | | |
|  | Linear Keyway Broach |  | <ol style="list-style-type: none"> 1. Width 2. Side to side diameter 3. Top to bottom Height 4. Check to add Corner radius |
|  | | | |
|  | Linear Chamfer Broach |  | <ol style="list-style-type: none"> 1. Side length of Chamfer 2. Width of keyway 3. Side to side diameter 4. Top to bottom Height |
|  | | | |
|  | Linear Corner Broach |  | <ol style="list-style-type: none"> 1. Length of edge 2. Included angle (degrees) 3. Top to bottom Height 4. Distance between shank center and Tool tip |
|  | | | |
|  | Linear Hexalobular Broach |  | <ol style="list-style-type: none"> 1. Tool size |
|  | | | |
|  | Linear 2D Form Broach |  | Custom dimensions |
|  | | | |



Linear 3D Form Broach



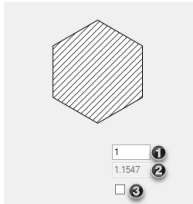


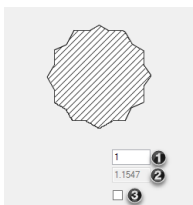


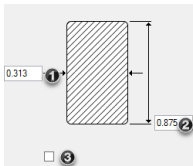


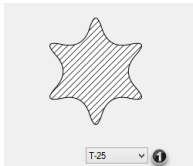


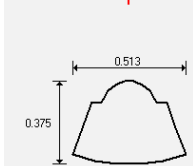


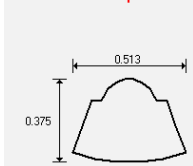


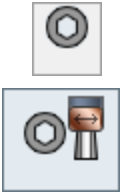
Custom dimensions

Rotary Broaching Tools

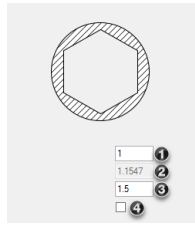
GibbsCAM v2025 supplies templates for seven rotary broaching tools.

Cutting end definition for rotary broaching tools

| icons | Name | End View | Notes |
|---|------------------------------|---|--|
|   | Rotary Hexagon Broach |  | <ol style="list-style-type: none"> 1. Diameter measured across flats 2. Across Corners (calculated) 3. Check to add Corner radius |
|   | Rotary Double Hexagon Broach |  | <ol style="list-style-type: none"> 1. Diameter measured across flats 2. Across Corners (calculated) 3. Check to add Corner radius |
|   | Rotary Rectangle Broach |  | <ol style="list-style-type: none"> 1. Side to side width 2. Top to bottom Height 3. Check to add Corner radius |
|   | Rotary Hexalobular Broach |  | <ol style="list-style-type: none"> 1. Tool size |
|   | Rotary 2D Form Broach |  | Custom dimensions |
|   | Rotary 3D Form Broach |  | Custom dimensions |

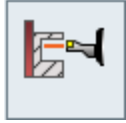


Rotary External
Hexagon



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.

Process #1 Linear Broaching

Linear Broaching | Rotate

1. Strategy

2. (illustration; varies with strategy)

3. (strategy-specific controls)

4. Stock control

5. Controls common to all strategies

Reference Hole
Center Point
End. Point

Hole Diameter 0
Use Circle Dia, where available

Across Flats 1.2
Across Corners 1.38564

Stock 0

Clear Amount 0.025
First Step 0.003
Final Step 0.003
Desired Stepover 0.003

0.02 0.1
Rapid In
0 -0.38

Entry Feed 8 ipm
Feedrate 6 ipm
Exit Feed 20 ipm
Ramp Out

Orientation 90 deg.
CW CCW

Multiple Orientations
Num Duplicates -1
Orient Angle 0 deg.
CW CCW

Coolant Flood


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
☐ Center Point
☐ End. Point



Hole Diameter

☒ Use Circle Dia, where available

☒ Across Flats
☐ Across Corners

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.

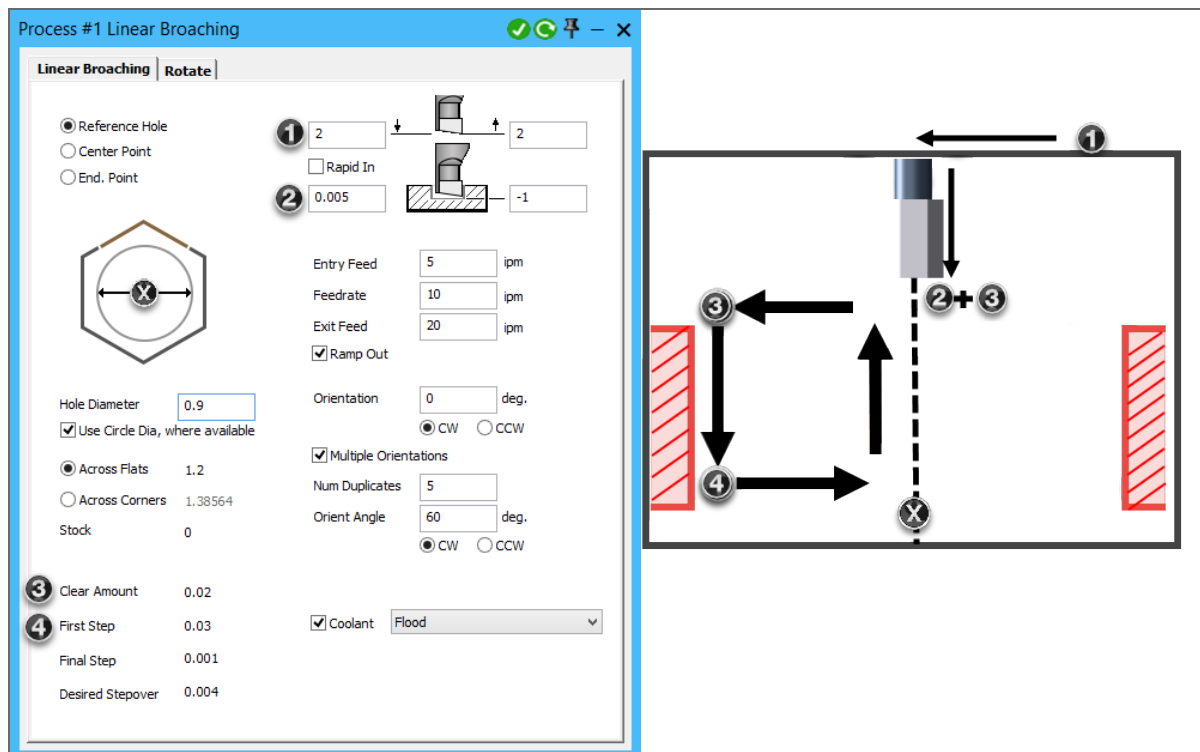
Next, specify the final size to be broached. For a hexagon tool this can be specified as the distance **Across flats**, or across the sharp corners. For other tools it will be the finished diameter.

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.



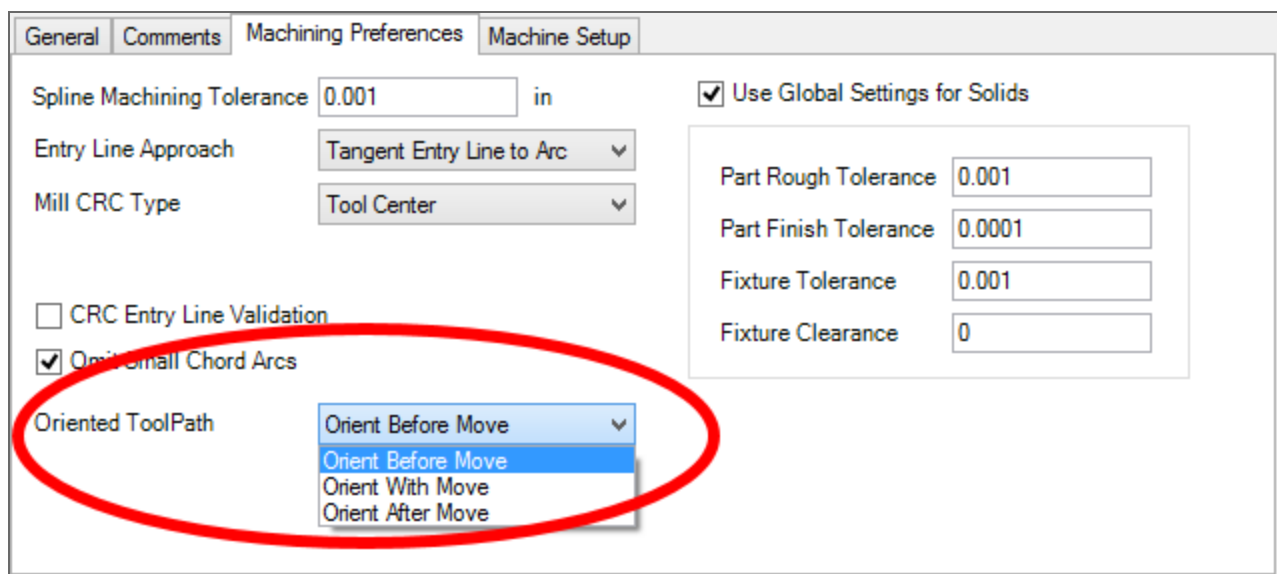
For the broaching process, the tool will move across to the hole center, rapid down to the Entry Clearance height Z (if it is above this value in Z). It will then move down to the start Z depth (either at feed or rapid, according to the **Rapid In** checkbox). This will be followed by a rapid move across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



The screenshot shows the 'Machining Preferences' tab in a software interface. The 'Oriented ToolPath' dropdown menu is open, displaying four options: 'Orient Before Move' (selected), 'Orient Before Move' (highlighted in blue), 'Orient With Move', and 'Orient After Move'. A red oval is drawn around the dropdown menu. Other settings visible include 'Spline Machining Tolerance' set to 0.001 in, 'Entry Line Approach' set to 'Tangent Entry Line to Arc', 'Mill CRC Type' set to 'Tool Center', and 'Use Global Settings for Solids' checked. Tolerance and clearance values for Part, Fixture, and Clearance are also shown.

| Setting | Value |
|----------------------------|-------------------------------------|
| Spline Machining Tolerance | 0.001 in |
| Entry Line Approach | Tangent Entry Line to Arc |
| Mill CRC Type | Tool Center |
| CRC Entry Line Validation | <input type="checkbox"/> |
| Orient Small Chord Arcs | <input checked="" type="checkbox"/> |
| Oriented ToolPath | Orient Before Move |
| Part Rough Tolerance | 0.001 |
| Part Finish Tolerance | 0.0001 |
| Fixture Tolerance | 0.001 |
| Fixture Clearance | 0 |

Center Point

- ☐ Reference Hole
☒ Center Point
☐ End. Point



- ☒ Across Flats 1
☐ Across Corners 1.15

Center point options

Across Flats/Corners (Hex)

Distance across flats/corners used to calculate the final diameter.

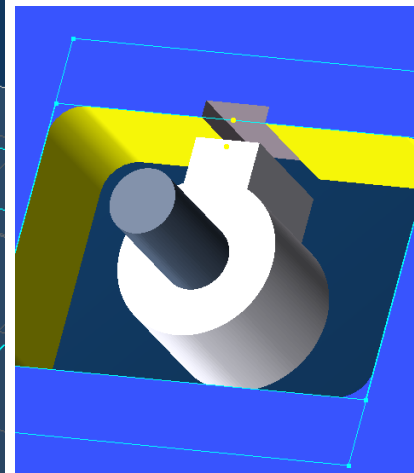
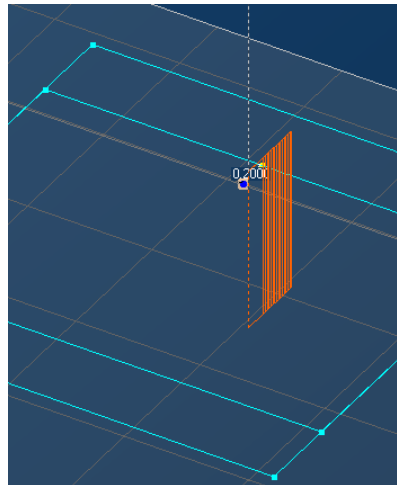
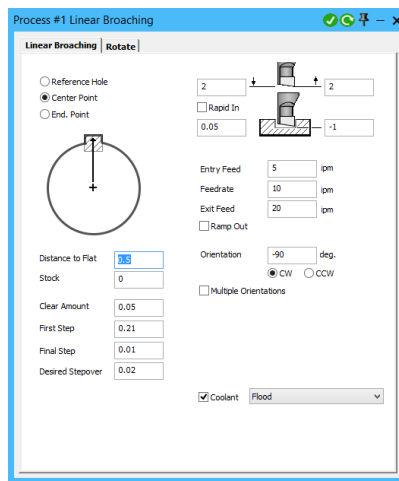
Distance to Flat (Key Cham)

Distance from center to keyway flat

Final diameter (Com, Lob, Form tools)

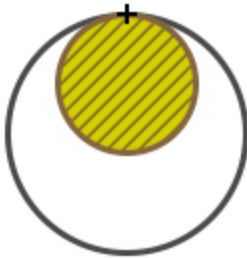
Center point strategy

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

- ☐ Reference Hole
- ☐ Center Point
- ☒ 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.

Process #1 Linear Broaching

Linear Broaching | Rotate

☐ Reference Hole
☐ Center Point
☒ End. Point

Stock: 0

Clear Amount: -0.4
First Step: -0.35
Final Step: 0.01
Desired Stepper: 0.02

Entry Feed: 5 ipm
Feedrate: 10 ipm
Exit Feed: 20 ipm
☐ Ramp Out

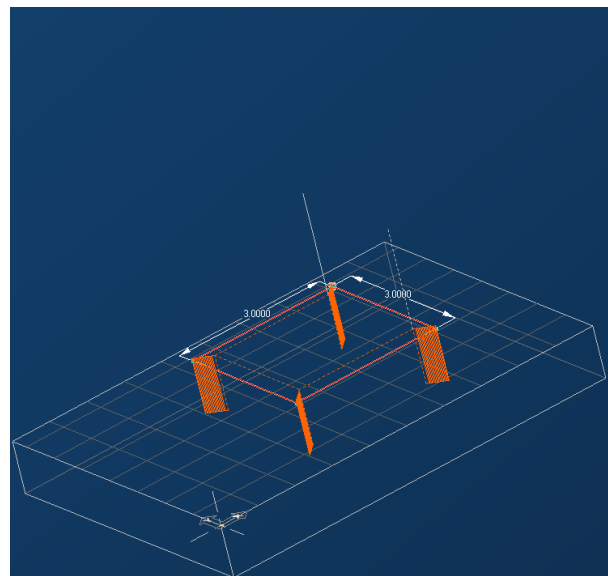
Orientation: 45 deg.
☐ CW ☒ CCW

☒ Multiple Orientations
Num Duplicates: 3
Orient Angle: 90 deg.
☐ CW ☒ CCW

Center: X 5 Y 4

☒ Coolant: Flood

2 2
0.05 -1



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

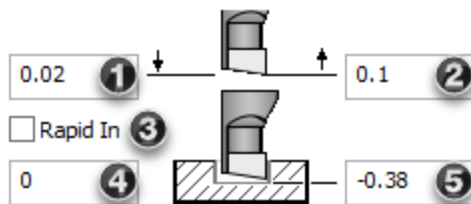
Final Step

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

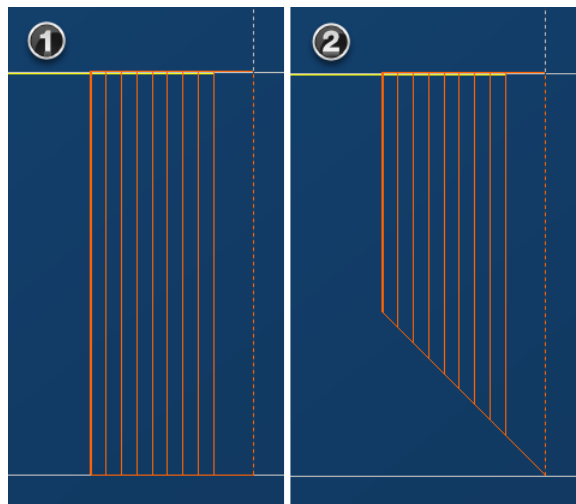
Feed speed.

Feedrate

Broaching Feedrate per minute.

Ramp out

When checked, exit at 45 degrees.



1. Unchecked
2. Checked

Orientation

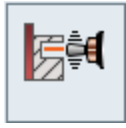
Tool orientation angle.

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

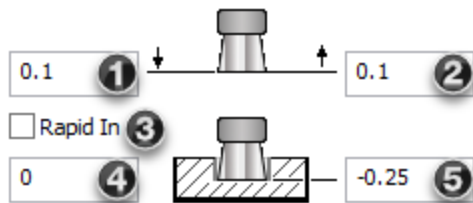
Check to specify a different RPM for the entry spindle speed.

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

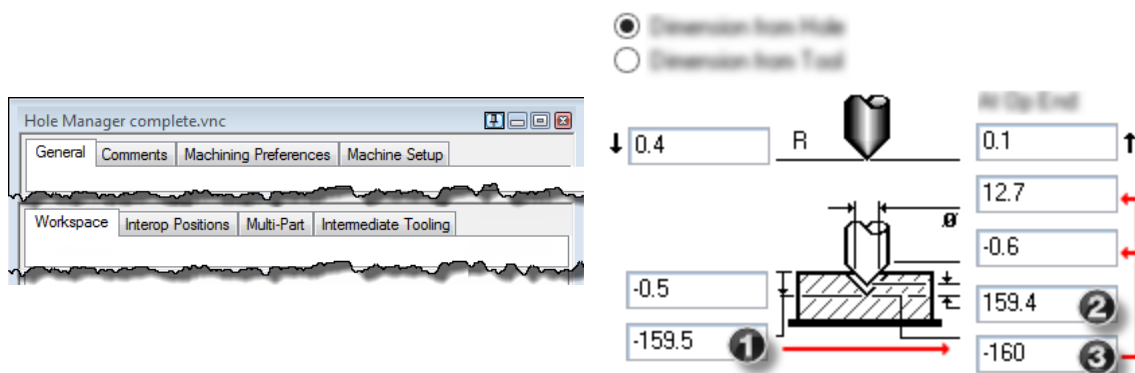
Screen text. Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

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.

Links to Online Resources

Please contact your reseller for support.

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