



GIBBSCAM 2026 CAM for
Production Machining

Version 2026 : September 2025

5-Axis Porting

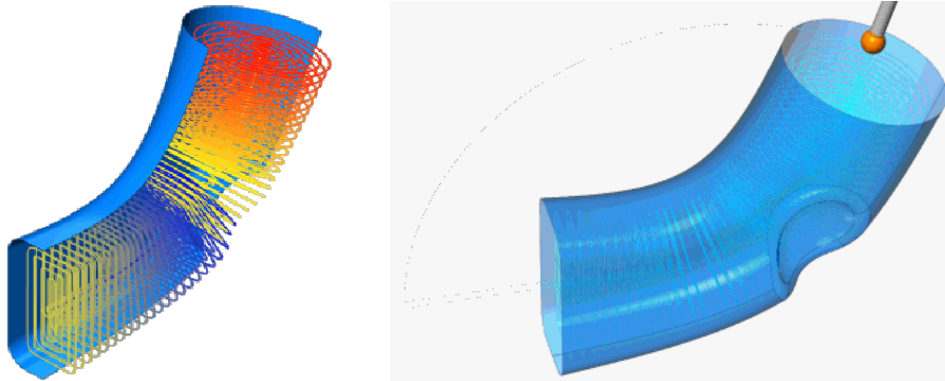


Contents

Introduction To 5-Axis Porting	3
Concepts and Terminology	3
Porting Compared to Other Calculation Strategies	4
Activating Port Machining Within 5-Axis	4
<hr/>	
Interface	5
Surface Paths tab for Port Machining	6
Pattern Settings	6
Part Definition Settings	6
Area Settings	7
Sorting Settings	7
Surface Quality Settings	7
Stepover Settings	8
Tool Axis Control for Port Machining	9
General Settings	9
Maching Angle Range Limitation Settings	9
Gouge Check tab for Port Machining	10
Link tab for Port Machining	11
Clearance Settings	11
Home Position Settings	12
<hr/>	
Conventions	13
Text	13
Graphics	13
<hr/>	
Links To Online Resources	14
<hr/>	
Index	15

Introduction to 5-Axis Porting

This document explains the use of the GibbsCAM 5-Axis Porting product. Port machining creates either rough or finished toolpath for port-type geometries, where a “tunnel” is created that connects and blends apertures of different shapes to one another.



Cutaway view

Machining of sample part

The sample part, `Porting Sample1 mm - Finish Around.vnc`, uses a lollipop with a tapered shaft. This is typical for port machining.

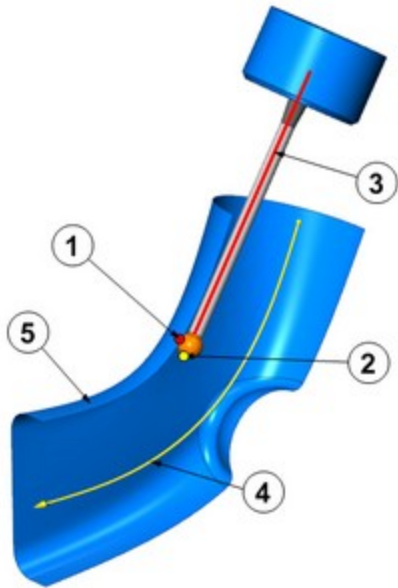
All members of the GibbsCAM 5-Axis family require 2.5D Solids or SolidSurfacer as prerequisite products, and a 4-axis/5-axis post processor. The current MDD must be of type Mill, Mill/Turn, or MTM.

Note: The capabilities and user interface described in this and other guides apply to GibbsCAM Industrial Edition with all product options licensed and active. GibbsCAM Viewer and GibbsCAM Student Edition provide a subset of the full functionality.

Concepts and Terminology

5-Axis Porting uses calculation based on triangle mesh to generate toolpath. It requires a part with an inner tunnel (**port**) composed of contiguous surfaces – **Machining surfaces**. This mesh of surfaces is bounded at the apertures by closed contours – a Top curve and Bottom curve.

The **spine** of the port is a space curve that smoothly connects the centers of the two closed contours as it runs approximately parallel to the surfaces. The spine is generated automatically, or you can select a curve in your part. In Roughing and in the Finishing **around** patterns, the cutter makes successive passes spiraling around the spine. In the Finishing **along** pattern, the cutter makes successive lengthwise passes.



1. **Contact point:** Point where the tool touches the port surface
2. **Cutter location point:** Point that is represented by the coordinates of the machine program
3. **Tool axis orientation:** Vector (I,J,K direction) of the tool axis
4. **Spine:** Curve that winds through the middle of the port
5. **Port surfaces:** Mesh of surfaces connecting the top contour to the bottom contour

Porting Compared to Other Calculation Strategies

Because 5-Axis Porting is designed for port-type parts and nothing else, there is no need to spend time creating, copying, slicing, or offsetting surfaces, or calculating curves and points, or the like. The user interface knows exactly the types of surfaces your part must have, and it supplies choices and parameters that are specific to ports.

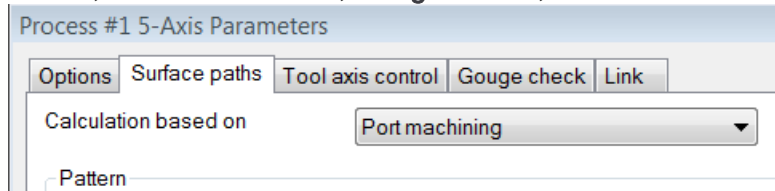
Activating Port Machining Within 5-Axis

Within a 5-Axis process dialog, the following steps activate the user interface for Port machining:

1. In the **Options** page, top pull-down menu: Select **General**.
 2. In the **Surface Paths** page, pull-down menu Calculation based on: Select **Port machining**.
- Result:* The settings offered by the Surface Paths page become specific to port machining, and other tabs in the process dialog are suppressed; see ["Interface" on page 5](#).

Interface

User interface controls for 5-Axis Porting are located in a process dialog with the following tabs: **Options**; **Surface Paths**; **Tool Axis Control**; **Gouge Check**; and **Link**.



- The **Options** page presents exactly the same controls as for all other 5-Axis process dialogs, such as feeds and speeds, coolant control, and patterns. Depending on your current MDD, it may also present controls for rotary duplication or for spindle. Note that the **Restore Defaults** button resets *all* 5-Axis controls, not just Porting. For a full discussion of the common settings in this section, with examples, see the [5-Axis](#) guide: [Options tab](#).

Output Port Machining Spine

When this option is selected, the curve calculated for the spine will be added to the part.

This is helpful for troubleshooting unexpected results from toolpath generation or from using the Automatic spine setting.

- The **Surface Paths** page, when Calculation based on is set to Port machining, presents controls for setting the machining type, surfaces, distances, toolpath cut direction, cut tolerances, and stepover parameters. For complete information, see [“Surface Paths tab for Port Machining” on page 6](#).
- The **Tool Axis Control** page, when Calculation based on is set to Port machining, provides a subset of the regular 5-Axis tool axis controls for tilt angles and gouge check controls for tool clearances. For complete information, see [“Tool Axis Control for Port Machining” on page 9](#).
- The **Link** page, when Calculation based on is set to Port machining, provides a subset of the regular 5-Axis controls for links between slices, links between layers, part clearance area, feed distance, and home position. For complete information, see [“Link tab for Port Machining” on page 11](#).

Surface Paths tab for Port Machining

The controls presented in the 5-Axis Porting Surface Paths page let you set general toolpath parameters for roughing and finishing operations.

Pattern Settings

The pull-down menu offers the following choices.

- **Roughing:** Use this for operations that remove large amounts of material.
- **Finishing along:** Use this to create fine passes axially (traveling parallel to the spine) along the length of the port.
- **Finishing around:** Use this to create fine passes radially (traveling perpendicular to the spine) spiraling around each slice of the port.
- **Rest Roughing:** Use this for operations that remove large amounts of rest material from a body you define as stock. With this choice, the Part definition section displays additional settings for stock and offset.

Part Definition Settings

The settings under Part definition provide the basic details of what is to be machined.

Machining surfaces

Clicking this button opens a dialog that lets you select the surfaces to be machined. Or, instead of a surface, you can select a facet body.

Offset


This value represents a virtual offset to the surfaces. The parameter enables you to specify the amount of material or stock allowance to remain on the surfaces. The offset can be understood as a 3-dimensional offset that expands the faces in all directions. For example, with an offset of 0.3 mm, the tool comes no closer than 0.3 mm to the selected faces.

Note: The offset from the surface is only as accurate as the machining accuracy (cut tolerance). That means that the offset can deviate with the selected tolerance. For example, with an offset of 0.1 mm and a cut tolerance of 0.1 mm, the real offset can go from 0.0 mm to 0.2 mm.

The offset will always be considered within gouge checking tool part clearances and the stock to leave value. All values will be added together, so in the end there is an overall clearance consisting of *offset + stock to leave + tool clearance*.

A positive value offset lets stock remain. A negative value lets the tool undercut the surface, and necessarily causes collisions.

Spine

Available only when the Automatic spine checkbox is **not** selected. The ellipsis button () opens a dialog that lets you select a curve to be used as the spine for the port.

Note that the **Options** tab provides a checkbox, **Output Port Machining Spine**, that allows you to output this curve as geometry in the workspace. This can be helpful in troubleshooting toolpath that does not match what you expected.

Automatic spine

When this checkbox is selected, the system will calculate the spine curve based on the machining surfaces.

Stock

Available only when the choice for **Pattern** is Rest roughing. The ellipsis button (...) opens a dialog that lets you select surfaces or a facet body to be used as stock.

Area Settings

The settings under **Area** give you fine control over cutting passes from each aperture.

Output type

You can choose whether to generate toolpath from the **Top** only, from the **Bottom** only, or from **Both top and bottom**. Top and bottom are determined by the chaining direction of the spine curve.

Machine to

You can choose whether the toolpath should extend to the **Midpoint** (about halfway between the start and of the spine), or to the maximum extent from top, or to the maximum extent from bottom, or according to parameters you specify for **Top** and **Bottom**.

Note: We highly recommend using one of the three automatic settings: **Midpoint**, or especially **Max. from top** or **Max. from bottom**. Instead of spending hours determining how far a tool might reach into a part, you can have the system provide an answer automatically in just a few seconds.

Top

For machining with the tool entering through the top aperture, specify the percentage of port length where the machining starts (usually 0%) and ends (usually around 50%).

Bottom

For machining with the tool entering through the bottom aperture, specify the percentage of port length where the machining starts (usually around 50%) and ends (usually 100%).

Sorting Settings

Direction for one way machining

Using **Conventional** cut direction, the tool proceeds along the drive surface biting from the inside.

Using **Climb** cut direction, the tool proceeds along the drive surface biting from the outside.



Ramp angle

Defines the angle of the ramp move from the end of one roughing pass to the start of the next.

Cutting method

Available only when **pattern** is **Finishing** along. Choose between **One way** and **Zigzag**.

Surface Quality Settings

Cut tolerance

Specify an overall tolerance for the entire toolpath. For a full discussion of the common settings in this section, with examples, see the [5-Axis](#) guide: [Surface Quality](#).



Tip: When generating trial toolpath for visualization, set a fairly loose tolerance value at first, such as 0.005 inches or 0.1 mm. Later, when you are satisfied with the toolpath for one segment, you can tighten the tolerance as needed.

Depending on the value for Cut tolerance, you will have many or relatively few points on the surface. This is especially true for round surfaces where you have more points because the toolpath always changes direction.

For more points on flat surfaces, select the Maximum distance checkbox.

Maximum distance

Smaller values will generate more points. Although the Cut tolerance is the same, you get more points on straight or flat surfaces because the distance is a maximum user-given distance to each other. For example, if this option is activated and the distance is set to 0.5mm, then at every 0.5mm (or less), a new toolpath position is calculated on the surface. If set, the value must be greater than 0.

Minimum distance

To prevent prolonged toolpath calculations generating many segments, you can select this checkbox and specify a minimum distance between points that are generated.

Stepover Settings

The stepover is the distance between two neighboring parallel cuts. For a full discussion of the common settings in this section, with examples, see the [5-Axis](#) guide: [Stepover](#).

Maximum stepover / Ridge height

The distance for the stepover can be defined directly, as a side step value (Maximum stepover). Alternatively, if you specify a value for maximum cusp height (Ridge height), the system automatically calculates a corresponding constant value for Maximum stepover.

Depth step

Available only when **Pattern** is set to one of the Roughing types. Specify a real number greater than 0. Each slice will be separated from the next by this amount.

Tool Axis Control for Port Machining

The controls presented in the 5-Axis Porting Tool Axis Control page provide a subset of the regular 5-Axis tool axis controls.

General Settings

Output Format

At this release, the only choice offered is 5-Axis.

Maximum angle step

This value sets the maximum allowed angle change between two adjacent toolpath positions. The calculation engine outputs 5-axis toolpath that contains the tool tip position and the tool axis vector. The tool axis vectors are not allowed to have an angle change more than the value specified here. Any number of degrees greater than 0 is a valid entry.



- Depending on the values for Cut tolerance and Maximum distance (see ["Surface Paths tab for Port Machining" on page 6](#)), there can be some toolpath positions where the angle step is less than this value.
- Decreasing the Maximum angle step generates more points; increasing the value generates fewer points.

Tool axis will ...

At this release, the only choice offered is Tilt automatically.

Maching Angle Range Limitation Settings

Machine angle limit

If your machine has limitations on tool angle, select this checkbox and enter the maximum angle that the tool is allowed to tilt for the specified Spindle direction. If this limit prevents the tool from reaching some areas of the toolpath, then that portion of the toolpath will be trimmed.

Spindle direction

This pull-down menu lets you choose a preset axis (X, Y, or Z). Or, instead, you can choose User defined direction and click the ellipsis button (...) to open a **Direction** dialog. This dialog allows you to enter XYZ values for the vector or, by clicking its (...) button, to select a line in the workspace.

Gouge Check tab for Port Machining

The controls presented in the 5-Axis Porting [Link](#) page provide a very simplified subset of the regular 5-Axis controls for gouge checking.

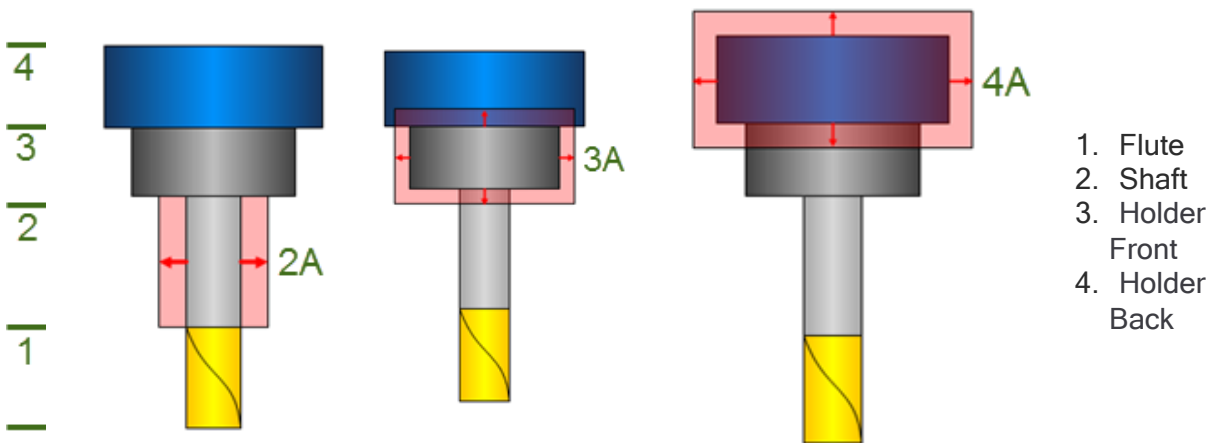
Tool clearances

The top three controls let you set cylindrical clearances for three portions of the tool.

Example: Cylindrical clearances.

For cylindrical clearances, you specify three linear values: One for the shaft diameter, one for the holder front diameter and length, and one for the holder back diameter and length.

Clearances - Cylindrical



2A = Shaft diameter clearance; 3A = Holder Front clearance; 4A = Holder Back clearance

The other controls let you specify the surfaces to guard against gouging.

Check against machining surfaces

Select this checkbox to guard the machining surfaces against gouging.

Check surfaces

When this checkbox is selected, the selector button () opens a dialog that allows you to select surfaces .

Link tab for Port Machining

The controls presented in the 5-Axis Porting Link page provide a simplified subset of the regular 5-Axis controls for controlling how the tool moves when it is not cutting.

Clearance Settings

The *Clearance area* is the area in which the tool can travel without hitting the work piece. For 5-Axis Porting, you set a value for *Clearance distance* and click *Clearance area* to define a cylindrical zone.

The tool moves at the rapid feedrate when at the clearance area. The tool is moved to its final orientation while moving from the clearance area to the rapid distance. After the tool is at the rapid distance, it has the correct orientation for the first cut.

Cylinder parallel to X, Y, Z, or to parallel to user-defined

This clearance area type has a cylindrical shape that can be aligned to the X, Y or Z axis. When defining this clearance area, ensure that it completely encloses the machining surfaces. The cylinder extends to infinity along the axis.

Process #1 5-Axis Parameters

Options | Surface paths | Tool axis control | Gouge check | **Link**

Home position

☐ Start from home position

☒ Return to home position

X 0 Y 0 Z 180 ...

Connection cylinder

Radius User defined value 200

Direction User defined axis ...

Through User defined point ...

Feed distance 50

Angle step for rapid moves 5

☒ Fillet 0

Radius

This is the main control for this clearance option. This value is the radius of the cylinder centered on the axis.

Through

to offset the axis of the cylinder so that it passes through a point other than X0,Y0,Z0, you can enter the coordinates of a point, or you can click the ellipsis button () and select a point in the workspace.



Because the cylinder is infinite in length, any value entered for a point along the parallel axis will be ignored. For example, if you choose the 3D point X+10, Y-5, Z+15 and the cylinder is parallel to Z, the Z value will be displayed but ignored.

Angle step for rapid moves

This parameter controls the length of the curved rapid moves that occur when moving along a non-planar clearance area such as a cylinder. The curved rapid moves are segmented into shorter line moves that do not exceed the angle step.

Home Position Settings


Start from home position

Select this checkbox to make the tool start from the home position you designate below.

Return to home position

Select this checkbox to make the tool return to the home position you designate below.

X/Y/Z

You can enter XYZ values for the home position, or you can click the ellipsis button () and then, in the **Select Home Position** dialog, select a point and click **OK** to return to the process dialog.

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

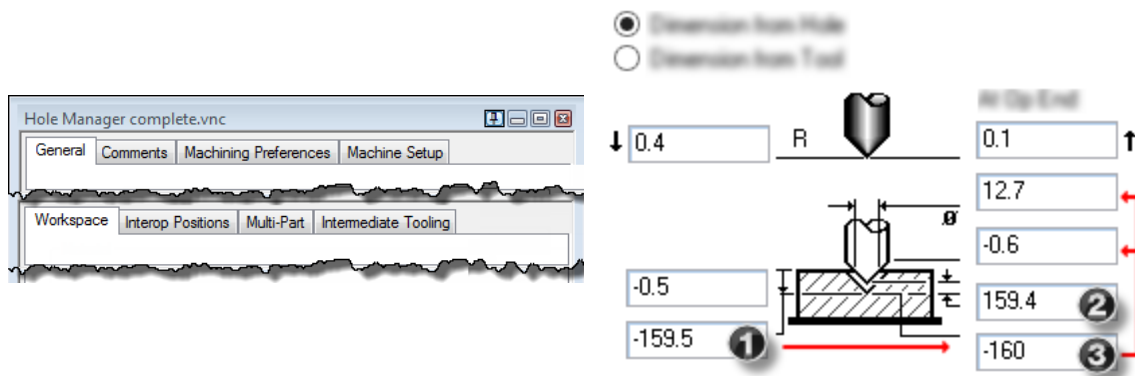
Screen text. Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

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.

Index

#

5-Axis Porting
 (illustrated) 3
 activating 4
 prerequisites 3
 purpose 3

A

angle limits (Tool Axis Control tab) 9
Angle step for rapid moves
 Cylinder clearance 12
Area, settings in Surface Paths 7
Automatic spine 6
 troubleshooting 5

B

Both, Area setting for Output type 7
Bottom (Surface Paths > Area) 7
Bottom, Area setting for Output type 7

C

Clearances (Link tab) 11
Climb Cut
 (illustrated) 7
Climb or Conventional (direction) 7
Conventional Cut
 (illustrated) 7
Conventional or Climb (direction) 7
Cut tolerance (Surface Quality setting) 7
Cylinder parallel to X, Y or Z, clearance 11

D

Depth step (Stepover setting) 8

F

Finishing along (Pattern setting) 6
Finishing around (Pattern setting) 6

G

Gouge Check tab for Port Machining 10

H

Home position (Link tab) 12

L

limits (Tool Axis Control tab) 9
Link tab for Port Machining 11

M

Machine angle limit (Tool Axis Control) 9
Machine to, Area settings 7
Machining surfaces 6
Material allowance
 offset 6
Max. from top, Area setting for Machine
 to 7
Maximum angle step
 for Tool axis control 9

Maximum distance (Surface Quality setting) 8

Maximum stepover (Part Definition tab) 8

Midpoint, Area setting for Machine to 7

Minimum distance (Surface Quality setting) 8

O

Offset

material allowance 6

stock allowance 6

Output format

for Tool axis control 9

P

Part definition, settings in Surface Paths 6

Pattern, settings in Surface Paths 6

Port machining

(illustrated) 3

activating 4

prerequisites 3

purpose 3

Port Machining tabs

Link 10-11

Surface Paths 6

Tool Axis Control 9

R

Radius

Cylinder clearance 11

Rest Roughing (Pattern setting) 6

Return to home position 12

Ridge height (Stepover setting) 8

Roughing (Pattern setting) 6

S

Spindle direction (Tool Axis Control) 9

Spine (Part Definition setting) 6

spine, automatic 6

troubleshooting 5

Start from home position 12

Stock (Part Definition setting) 7

Stock allowance

offset 6

Surface Paths tab, for Port Machining 6

Surfaces

machining 6

T

tabs, Port Machining interface

Gouge Check 10

Link 11

Surface Paths 6

Tool Axis Control 9

Through, cylinder clearance 11

Tool Axis Control for Port Machining 9

Tool axis vector 9

Top (Surface Paths > Area) 7

Top, Area setting for Output type 7

troubleshooting 5