
Version 14 : September 2020

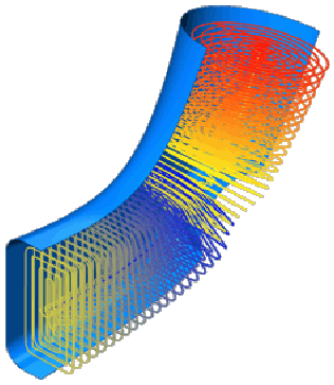
# 5-Axis Porting

# Proprietary Notice

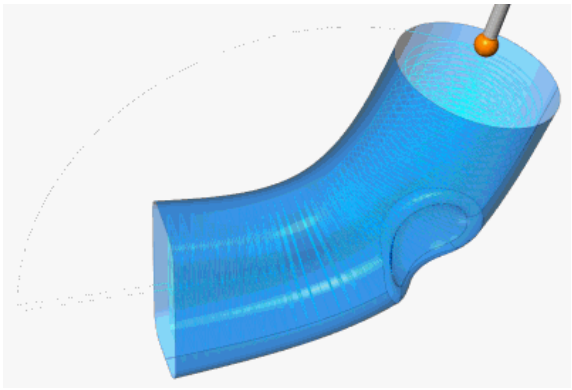
# Contents

# 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

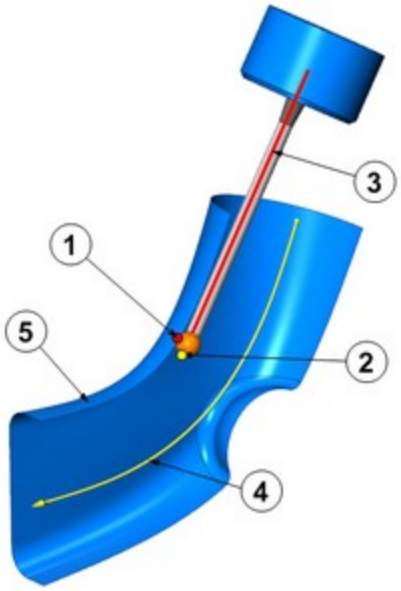
The sample part, `Porting Sample1 mm - Finish Around.vnc`, uses a lollipop with a tapered shaft. This is typical for port machining.

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.

## 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 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

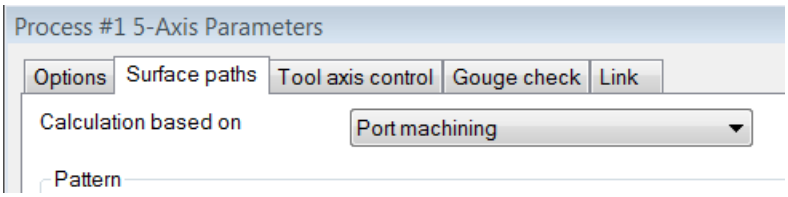
Within a 5-Axis process dialog, the following steps activate the user interface for Port machining:

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 7.

# 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 8.

- 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 11.

- 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 13.

# 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.

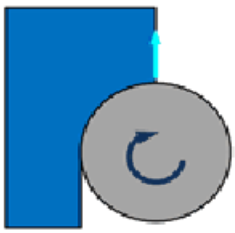
**Note:** We highly recommend using one of the three automatic settings: Midpoint, or especially Max. from top or Max. from bottom. Instead of spending hours determining how far a tool might reach into a part, you can have the system provide an answer automatically in just a few seconds.

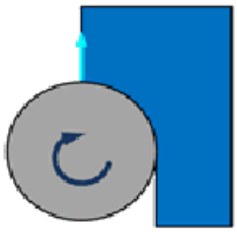
#### Top
For machining with the tool entering through the top aperture, specify the percentage of port length where the machining starts (usually 0%) and ends (usually around 50%).

#### 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.





| Conventional | Climb |
|---|---|

### 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 8), 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

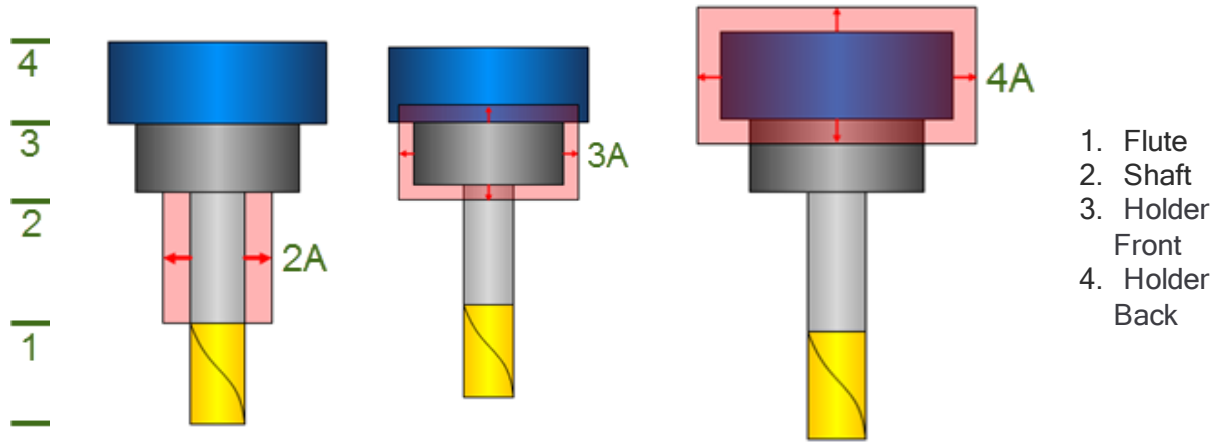
The controls presented in the 5-Axis Porting Link page provide a very simplified subset of the regular 5-Axis controls for gouge checking.

### 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 the5-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.

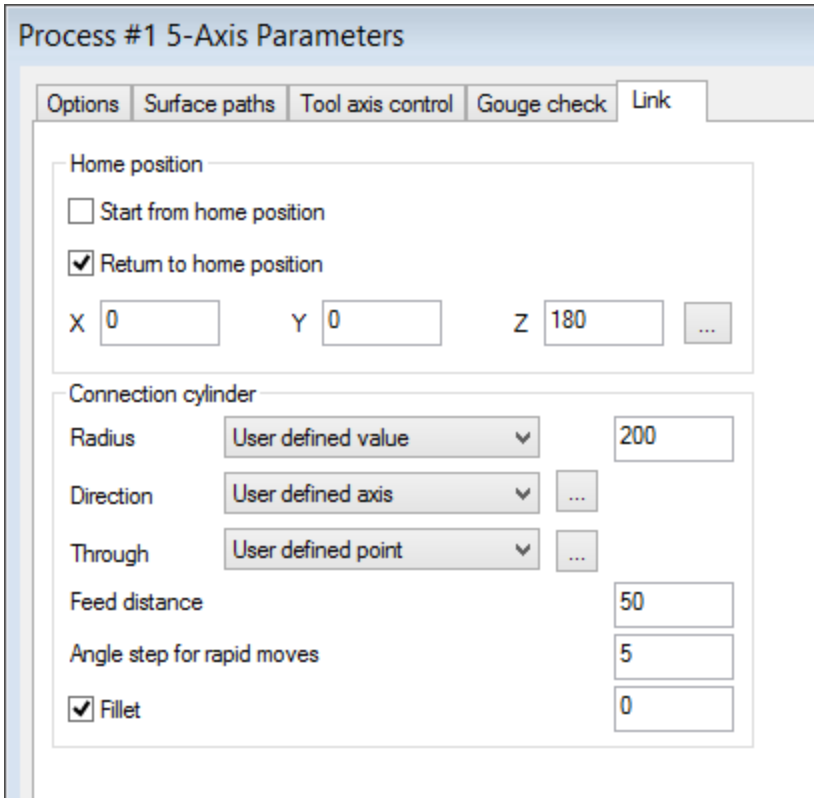
## Clearance Settings

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.




### 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.

# Using 5-Axis Porting

This part of the documentation covers the following areas:

- Getting Started with 5-Axis Porting – Provides general guidance on steps to take for programming a port-type part using 5-Axis Porting.
- "Sample Part: 5-Axis Porting" on page 16 – Provides specific examples of typical actions and settings for the sample part `Porting Sample1 mm.vnc`.

## Getting Started with 5-Axis Porting

After you open a porting part, choose a tool, create a 5-Axis process, and set material/feeds/speeds, usual practice is as follows.

1. First, in the **Surface Paths** page: Activate Calculation based on Port machining; under Pattern, choose Roughing. (Or, if you are performing rest machining from an existing body, choose Rest roughing.)
2. Next, in the **Part Definition** portion of the page: Define the machining surfaces and a value for Offset. Retain the default (selected) state of the Automatic spine checkbox. For rest roughing, also select a body forStock and supply a value for Stock offset.
3. In the **Area** portion of the page, keep the defaults, so as to output Both (top and bottom) and machine to the Midpoint between them.
4. In the **Surface quality** portion, enter a loose value for Cut tolerance for now, such as `0.1` mm or `0.005` inches. Later, you might need to tighten this to avoid gouging.

   Similarly, in the **Stepover** portion, enter fairly generous values at first, to reduce toolpath complexity. When you are close to completing the roughing operation, you will come back and change these to smaller values.
5. Next, select the **Tool Axis Control** tab and specify realistic values for the current tool. A value of `3` degrees is usually ample. Later, you might decrease this value in order to generate more tightly controlled toolpath.
6. Next, select the **Gouge Check** tab and specify clearance values for the current tool.
7. Generate the operation. Render it and inspect the preliminary results.
8. Refine settings as needed. For example:
   - If you misdefined any portion of the part or check surfaces, return to the **Surface Paths** page, click Machining surfaces, and `CTRL`-select to add or remove surfaces.
   - If gouging is occurring, try one or more of the following, depending on what you see:

- In **Surface Paths**, supply a tighter value for Cut tolerance and Stepover, and, if applicable, for Stock Offset.
- In **Tool Axis Control**, supply a smaller value for Maximum angle step.
- In **Gouge Check**, supply larger values for clearances. As needed, select the Check surfaces checkbox and specify other surfaces to check, using CTRL-select to add or remove surfaces.
- In **Link**, supply a more generous value for Clearance distance.

9. Repeat the previous step as needed, until you are pleased with the result.
10. After you complete your Roughing operation, create new operations with Pattern set to Finishing along or Fnishing around.

    For tools of smaller radius, be sure to adjust settings appropriately for feeds and speeds (in **Options**) and for offsets/clearances/tolerances.

# Sample Part: 5-Axis Porting

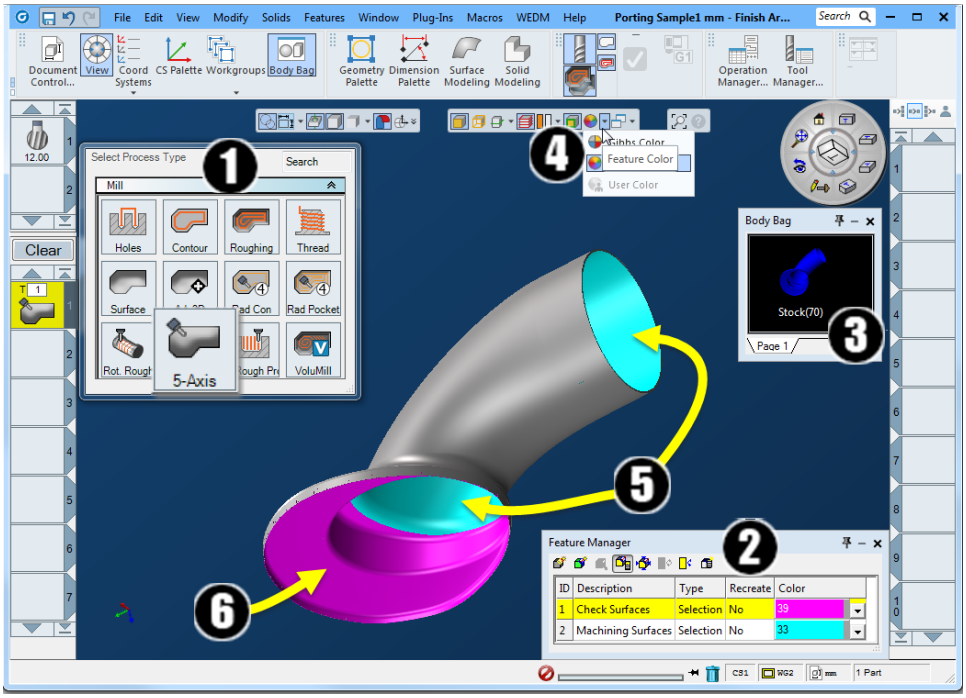
Sample parts are provided on the installation DVD. They can also be downloaded from https://online.gibbscam.com (login required).

For 5-Axis Porting, the sample part name is `Porting Sample1 mm.vnc`.



## Step A: Open and Explore the Sample Part

1. Start GibbsCAM, navigate to the folder containing sample parts, subfolder `5-Axis Porting\`, and open the sample part (`Porting Sample1 mm.vnc`).
2. Open the tool list, the Machining palette, and the Body Bag.
3. Temporarily change to Feature Color mode. 
4. Become acquainted with the part and its component surfaces.
5. When finished, change back to Gibbs Color mode.




1. Process types
2. Feature Manager: Feature #1 (cyan): Machining surfaces Feature #2 (pink): Check surfaces
3. Body Bag
4. Feature Color (icon)
5. Port surfaces
6. Check surfaces

# Step B: Create and Set Up a 5-Axis Porting Process

1. From the Tools List, drag the tile for Tool 1 onto the blank process tile. On the process selection palette, choose the 5-Axis process type.

   *Result:* The 5-Axis process dialog opens to the **Options** page.

2. In the **Options** page: Click Restore Defaults, and then click Material. In the Materials dialog, click Calc RPM and CalcFeed to populate values for feeds and speeds, and then close the Materials dialog.

3. Click the **Surface Paths** tab. In the pull-down menu for Calculation based on, change from Surfaces to Port machining. Notice how the tabs to the right change: Two tabs disappear (**Roughing** and **Utility**), and the three that remain contain a simplified and streamlined set of controls.

# Step C: Provide Initial Settings for 5-Axis Porting Processes

1. Explore the **Surface Paths** settings. For example:

   - Under Pattern, several strategies are available for Roughing and Finishing.

2. Before moving on to other tabs, set the following:

   - Pattern: Roughing

- Part definition / Machining surfaces: Click the ellipsis button (…). When prompted to select machining surfaces, click the corresponding row in Feature Manager.
- Area / Output type: Both
- Area / Machine to: Midpoint
- Sorting / Ramp angle: 5
- Surface quality/ Cut tolerance: 0.1

3. Click the **Tool axis control** tab and explore the settings. For now, specify a fairly large Maximum angle step of 3 degrees. Later, you might decrease this value in order to generate more tightly controlled toolpath.
4. Click the **Gouge check** tab and explore the settings. Before moving on to other tabs, set the following:
   - Holder back: 6 mm
   - Holder front: 6 mm
   - Shaft: 0.5 mm
   - Check surfaces: Click the ellipsis button (…). When prompted to select check surfaces, click the corresponding row in Feature Manager.
5. In the **Link** page, set the Clearance distance to 50 mm.

# Step D: Generate, Inspect, and Refine Initial Results

1. Generate the operation and render it. Rotate and zoom the preliminary results to inspect them closely.
2. Refine settings as needed. For example:
   - If gouging is occurring, try one or more of the following, depending on what you see:
     - In the**Surface paths** nd **Gouge check** pages, supply larger values for Stock to leave and offsets/clearances.
     - Under **Surface paths** / Surface quality, supply a tighter value for Cut tolerance.
     - In **Tool Axis Control**, supply a smaller value for Maximum angle step.
     - In **Link**, supply a more generous value for Clearance distance.

# Step E: Create Finishing Operations

1. After you complete your Roughing operations, create new operations with Machining set to Finishing around and then to Finishing along.

   For tools of smaller radius, be sure to adjust settings appropriately for feeds and speeds (in **Options**) and for offsets/clearances/tolerances.

# 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

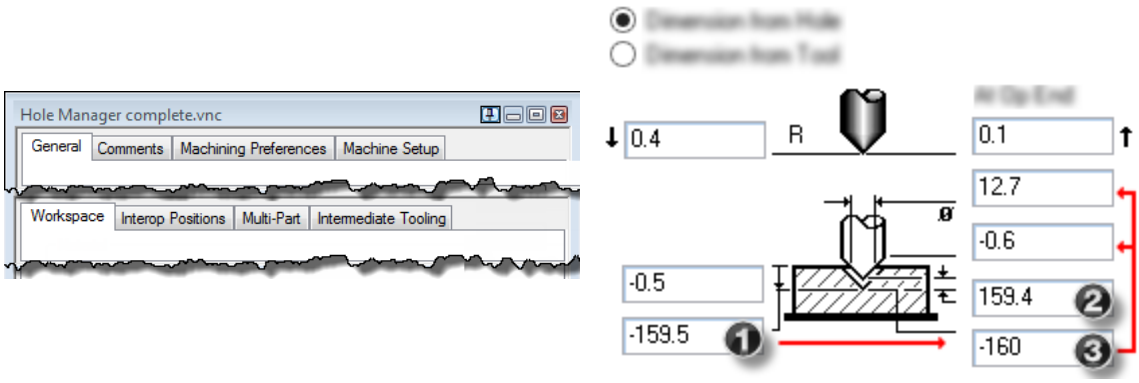
Screen text. Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

`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

## O



## P



## R



## S



## T