Version 14 : September 2020

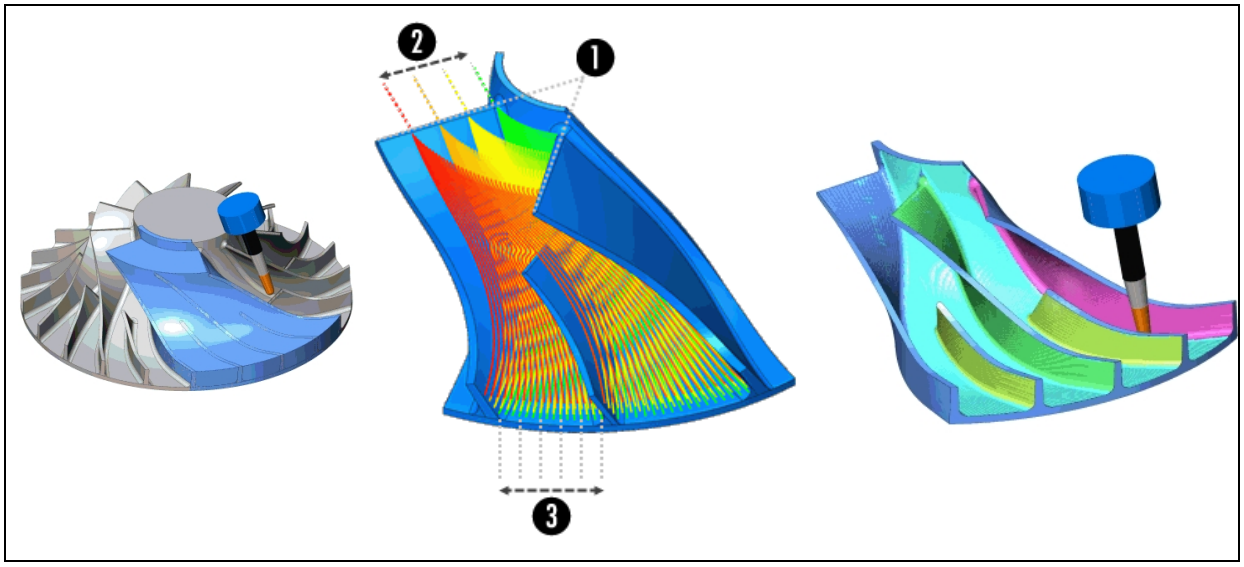
5-Axis-MultiBlade

Proprietary Notice

Contents

Introduction to 5-Axis MultiBlade

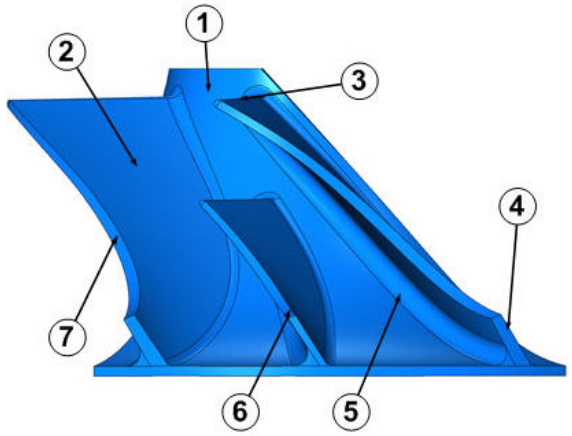
This document explains the use of the GibbsCAM 5-Axis MultiBlade product. MultiBlade is the preferred solution for machining impellers, blisks (bladed disks), and the like – parts that compress or transport fluids.



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.

MultiBlade Terminology

MultiBlade generates toolpath only for parts that contain one inner ***hub*** surface, one outer ***shroud*** surface, ***blades*** (whose ***leading edge*** and ***trailing edge*** extend from hub to shroud), and ***fillets*** on each blade.



Each ***segment*** includes a left blade, a right blade, and the items between them.

1. ***Hub*** surface, sometimes also called *floor*.
2. Main ***blade***. The illustration shows the right side of the left main blade for this segment.
3. ***Leading edge*** of the right main blade
4. ***Trailing edge*** of the right main blade
5. ***Fillet*** connecting the hub and the left side of the right main blade
6. ***Splitter*** blade between the left and right main blades
7. ***Shroud*** surface

Activating MultiBlade Within 5-Axis

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

1. In the **Options** tab, top pull-down menu: Select General.
2. In the **Surface Paths** tab, pull-down menu Calculation based on: Select Multiblade parts.

Result: The tabs in the process dialog change to the following: Options, Surface Paths, Part Definition, Tool Axis Control, Link, and Edges.

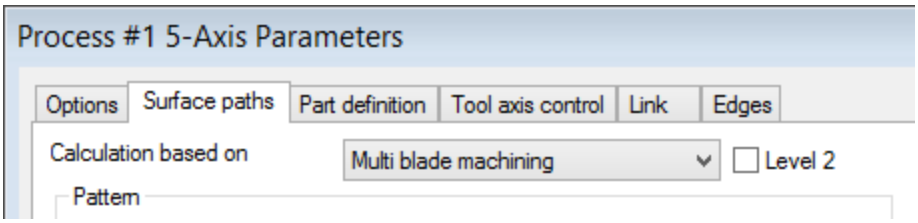
MultiBlade Compared to Other Calculation Strategies

Because MultiBlade is designed for impeller-type parts and nothing else, there is no need to spend time extracting/untrimming floor surfaces or cross-sections, creating/copying/slicing/offsetting surfaces, separating blades, and so forth.

The MultiBlade user interface knows exactly the types of surfaces your part must have, and it supplies choices and parameters specific to impeller surfaces – blades arranged in a radially symmetric pattern sitting on a central hub, machined using a sphere-tip tool (Sphere mill, Lollipop mill, or Taper mill).

Interface

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



- 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 MultiBlade. For complete information, see *5-Axis* guide's information on the Options tab.
- The **Surface Paths** page, when Calculation based on is set to MultiBlade, presents controls for setting the machining type and strategy, toolpath sorting method (such as one-way/spiral/zig-zag) and cut direction, and parameters for toolpath layers and slices. For complete information, see "Surface Paths tab for MultiBlade" on page 7.
- The **Part Definition** page presents controls for specifying the surfaces and segments to be machined and for setting the machining quality. For complete information, see "Part Definition tab" on page 12.
- The **Tool Axis Control** page, when Calculation based on is set to MultiBlade, 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 tab for MultiBlade" on page 15.
- The **Link** page, when Calculation based on is set to MultiBlade, 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 MultiBlade" on page 18.
- The **Edges** page presents controls for handling surfaces for the leading edge (fluid intake) and trailing edge (exit) surfaces. For complete information, see "Edges tab" on page 19.

Surface Paths tab for MultiBlade

The controls presented in the MultiBlade Surface Paths page let you set general toolpath parameters for roughing and finishing operations, such as sorting and layers/slices.

Some settings on this tab have an effect on other portions of the interface. For example, if you choose roughing or hub finishing, the Contour options are suppressed. Similarly, the **Part**

Definitions tab lets you specify Shroud parameters only if you choose a roughing or blade finishing for strategy that includes the shroud.

Advanced
Available only if you are licensed for 5-Axis MultiBlade Level 2. When this checkbox is selected, the interface changes in the following ways:

- On the **Surface Paths** page:
  - Under Pattern, the Machining pull-down menu offers Fillet finishing.
  - For roughing operations, under Rest material, you can specify Avoid incomplete layers or Rough all layers.
- On the **Part Definition** page:
  - You can define stock and stock offset.
  - Under Segments, you can specify several sorting options.
  - Under Quality, you can control smoothing parameters.
- On the **Tool Axis Control** page: Under Limits, you can specify a machine angle limit and maximum angle step.
- On the **Edges** page: Under Edge rolling, you can specify whether and how to trim the leading edge and trailing edge.

Pattern Settings

Machining
This pull-down menu offers the following choices.

- Roughing: Use this for operations that remove large amounts of material from shroud to hub, using one of the options for Strategy.
- Blade finishing: Use this for fine operations that finish main blades and splitters, using options for Strategy and Contour.
- Hub finishing: Use this for fine operations that finish the hub surface.
- Fillet finishing: Use this for fine operations that finish the fillets between hub and blades, using one of the options for Contour and specific settings for blade side and hub side described in "Fillet Finishing" on page 12. *Available only if you are licensed for 5-Axis MultiBlade Level 2.*

Strategy
This pull-down menu, when available, offers the following choices.

- Offset from hub: Each layer will be approximately parallel to the hub (inner) surface. For this strategy, the shroud (outer) surface need not be a surface of revolution. Because the hub and shroud surfaces are usually not parallel, layers that are offset from the hub will start to intersect the shroud at some point. In this case, slices will be trimmed away so that they do not reach over.

- Offset from shroud: Each layer will be approximately parallel to the shroud surface, which must be a surface of revolution. Because the hub and shroud surfaces are usually not parallel, layers that are offset from the shroud will start to intersect the hub at some point. In this case, slices will be extended until they reach the hub surface edge.
- Morph between shroud and hub: Layers will be interpolated to blend between inner and outer surfaces, which must both be surfaces of revolution: layers closest to the hub will approximate hub offsets, and layers closest to the shroud will approximate shroud offsets. In the Area section, you can specify where the morphing starts and ends. In this case, cuts are neither trimmed nor extended. Although this choice works well in most cases, chip load may diminish on the trailing-edge side for blades whose leading edge is significantly longer than the trailing edge.

Tip: Thin blades can benefit from extra passes. For example, for the outermost 60%, you might want to create a roughing operation using Offset from shroud followed by a blade finishing operation. Then, for the remaining material, create a roughing operation using Offset from hub followed by another blade finishing operation.

Contour Settings

The settings under Contour, for some finishing patterns, let you limit the extent of the toolpath generated for the surfaces and segments specified in the Part Definition page.

This pull-down menu, when available, offers the following choices.

- Full: Toolpath will be generated for all corresponding surfaces.
- Full (trim trailing edge): Toolpath will be generated for all corresponding surfaces except the trailing edge (exit) surfaces.
- Full (trim trailing/leading edges): Toolpath will be generated for all corresponding surfaces except the trailing edge (exit) and leading edge (fluid intake) surfaces.
- Left side: Toolpath will be generated only for the left sides of surfaces you specify in Part Definition.
- Right side: Toolpath will be generated only for the right sides of surfaces you specify in Part Definition.
- Pocket (Blade finishing only): For each segment, toolpath will be generated only for the inner sides of surfaces you specify in Part Definition.

Sorting Settings

The settings under Sorting allow you to control the cutter direction along the first slice and how it changes direction with each subsequent slice, using the settings under Links between slices.

Method

This pull-down menu offers the following choices.

- One way, start from leading edge: Each cutter pass will move in the same direction, always proceeding from the leading edge downward and outward to the trailing edge.
- One way, start from trailing edge: Each cutter pass will move in the same direction, always proceeding from the trailing edge upward and inward to the leading edge.
- Zig-zag, start from leading edge: The cutter will first proceed from the leading edge downward and outward to the trailing edge. Then it will step over and continue machining in opposite direction.
- Zig-zag, start from trailing edge: The cutter will first proceed from the trailing edge upward and inward to the leading edge. Then it will step over and continue machining in the opposite direction.

Warning: If no zig-zag path can be generated, a one-way path will be generated instead.

Ordering

This pull-down menu, when available, offers the following choices.

- Left to right: The first slice will be on the farthest left of the segment. Each subsequent slice will be farther to the right.
- Right to left: The first slice will be on the farthest right of the segment. Each subsequent slice will be farther to the left.
- From center away: The first slice will be in the center of the area to be machined. Subsequent slices will alternate sides proceeding away from the center. For zig-zag approaches, you also specify Climb or Conventional.

Cut direction: Climb / Conventional

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.

Layers Settings

The settings under Layers (for roughing or for blade finishing) allow you to specify either how many outer-to-inner passes are made from shroud to hub, or else the depth-of-cut for each pass.

Tip: To visualize a trial toolpath most quickly and easily, set By maximum number =1 at first. Later, when you are satisfied with other settings and ready to generate final toolpath, you can use a more realistic setting for Layers.

By maximum number

Specify the maximum number of layers to cut from shroud to hub.

By maximum distance / Distance

Specify the maximum depth-of-cut for each layer.

Slices Settings

The settings under Slices (for roughing or for hub finishing) allow you to specify either how many edge-to-edge passes are made between blades, or else the stepover distance from one slice to the next.

By maximum number
Specify the maximum number of slices to cut from one blade to the next, considering splitters as blades.

By maximum distance
Specify the maximum stepover from each slice to the next.

First Slice Settings

The settings under First slice (for roughing only) allow you to specify different settings when the cutter enters a new layer for the first time.

Number of intermediate slices
Specify the number of passes to take when cutting the first slice.

First slice feedrate %
Optionally, you can specify a slower feedrate for the first slice, as a percentage of the general feedrate set in the **Options** page.

Area Settings

The settings under Area become available for roughing or for blade finishing when using the strategy Morph between shroud and hub.

Start at(%)
Specify the percentage of blade length from the hub where the morphing starts.

End at(%)
Specify the percentage of blade length from the shroud where the morphing ends.

Rest material

Available only if you are licensed for 5-Axis MultiBlade Level 2.

The options under Rest material (for roughing only) allow you to choose between the following.

- Avoid incomplete layers: Choose this option if you want to skip layers that were incompletely machined in the previous pass.
- Rough all layers: Choose this option if you want to machine all layers, even those that were only partly machined in the previous pass.

Fillet Finishing

Available only if you are licensed for 5-Axis MultiBlade Level 2. Use this pattern to generate a finishing toolpath on the fillet area between hub and blade. The system finds the fillet surfaces automatically. You can define the area to be machined for the hub and blade sides of the fillet independently or together.

Blade-Side Settings

Area

The settings under Area allow you to how the blade-side portion of the fillet will be machined.

- By number of cuts: With this option, you specify how many blade-side cuts to make and, under Both Sides, a stepover distance.

  #### Number of cuts

  Specify the number of blade-side cuts for the fillet.

- By big tool diameter: With this option, you use the tool diameter under Both Sides to determine which blade-side fillet areas were unreachable by the roughing tool, and you can also specify blade overlap.

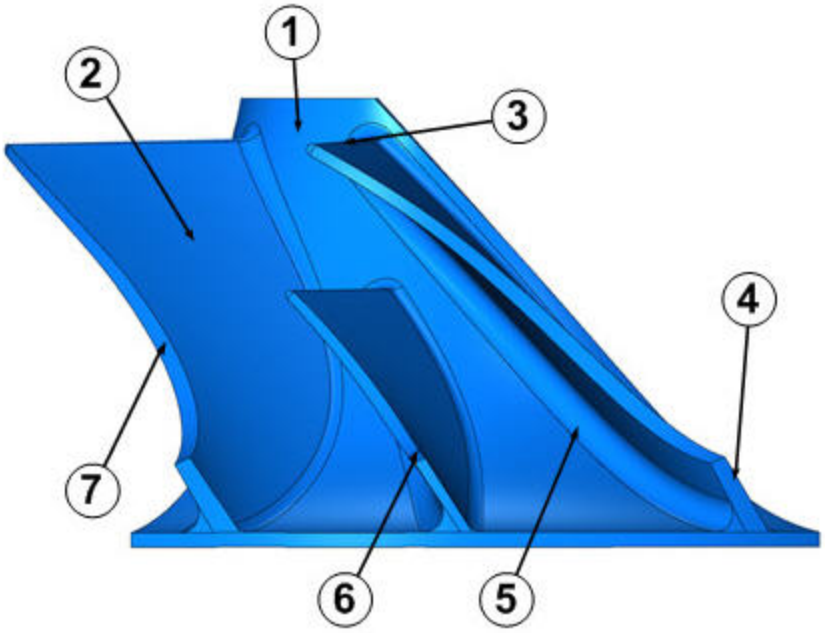
  #### Blade overlap

  #### By maximum number

Part Definition tab

The controls presented in the MultiBlade Part Definition page let you specify the surfaces and segments to be machined and set the machining quality.

Part Definition Settings



Blades,splitter,fillets / Stock to leave

Blades, such as main blades (2) and splitters (6), are usually free-form surfaces with a double curve. Fillets (5) are part of the blade. Each blade has a leading edge (3) and a trailing edge (4): the leading edge is the suction side for the transported fluid, and is usually rounded; the trailing edge is the exhaust side, usually corresponding to the stock geometry.

For roughing and hub finishing, you specify the left main blade and right main blade for the segment, and optionally one or more splitter blades between them. For blade finishing, you specify only the blade or splitter you want to machine. For roughing or blade finishing, you must also specify all associated fillets.

To specify main blades (2,3,4), splitter blades (6), and fillets (5): Click the ellipsis button (...); then, in the **Select Surfaces** dialog, select a surface (or `CTRL`-select multiple surfaces) in the workspace, and click `OK` to return to the process dialog. Enter a value for the amount of stock to remain unmachined.

Hub / Stock to leave

The hub surface (1), also called the floor surface, is the inner surface, and must be a surface of revolution. The blades and splitters sit upon it.

To specify the hub (1): Click the ellipsis button and, in the **Select Hub Surfaces** dialog, select any inner surface that adjoins both a leading-edge fillet and a trailing-edge fillet (you do not need to select all surfaces of the entire 360-degree hub, but you must also select the "collar" above the leading-edge fillets if it exists), and then click `OK`. Enter a value for the amount of stock to remain unmachined.

Shroud / Stock to leave
The shroud surface (7) is the outer surface. It is usually the same as revolved-surface stock, but it can be a free-form surface if you are roughing using Offset from hub strategy.

To specify the shroud (7), click the ellipsis button and, in the **Select Shroud Surfaces** dialog, select any one of the small outer surfaces that adjoins both a leading edge and a trailing edge (you do not need to select all surfaces of the entire 360-degree shroud), and then click OK. Enter a value for the amount of stock to remain unmachined.

Check surfaces / Clearance
Select this checkbox if you want to specify a clearance value for one or more additional check surfaces. For example, when you are finishing a splitter, you might specify blade faces as check surfaces.

Stock definition / Stock offset
Select this checkbox if you want to specify a body for rest material. Wherever no stock is defined, no toolpath will be created. *Available only if you are licensed for 5-Axis MultiBlade Level 2.*

Rotation axis
If the system cannot automatically detect the axis of rotation, you can define it yourself.

- For rotation axis: Select User defined and click the ellipsis button (…); then, in the **Rotary Axis Selection** dialog, specify XYZ values for the orientation vector (or click the ellipsis button, select a line, and click OK) and then click OK.
- For rotation axis base point: Click the ellipsis button (…); then, in the **Rotary Base Point** dialog, specify XYZ values for the base point (or click the ellipsis button, select a point, and click OK) and then click OK.

Number of segments
Specify the total number of segments in the impeller or blisk.

Segments Settings
A segment is a portion of the part from one main blade to another. For example, if a part consists of eight segments, it has eight main blades.

Machine
Specify how many segments should be machined by the current operation.

Tip: To visualize a trial toolpath most quickly and easily, set Determined number =1 at first. Later, when you are satisfied with the toolpath for one segment, you can increase the number or change the setting to All.

Start angle
To specify the start position of the toolpath, either identify the segment number (At segment) or supply an angle (User-defined).

Direction
When you are machining multiple segments, you can specify whether to proceed clockwise or counterclockwise. (This is independent of the slice ordering *within* a single segment; see "Sorting

Settings" on page 9.)

Sort by

*Available only if you are licensed for 5-Axis MultiBlade Level 2.*When you are machining multiple segments, you can specify:

- Complete segment: To machine the slices in all layers in the current segment before proceeding to the next segment.
- Layer: To machine all slices in the current layer in all segments before proceeding to the next layer.
- Slice: To machine one slice in the current layer in all segments before proceeding to the next slice.

Quality Settings

Machining tolerance

Specify an overall tolerance for the entire toolpath.

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 to a more realistic value.

Warning: Setting too loose a tolerance can cause toolpath calculation to fail, especially for fillet finishing.

Splitter flowline smoothing

Available only if you are licensed for 5-Axis MultiBlade Level 2. Drag the slider or use the arrow keys to specify how much to smooth the toolpath as it moves aside for the splitter. The lowest setting of 0% would leave less stock, but it can create jerky toolpath with acute angles. A high setting, such as 20%, would create much smoother toolpath that leaves more stock at the leading-edge fillet of the splitter.

Tool axis smoothing

Available only if you are licensed for 5-Axis MultiBlade Level 2. Drag the slider or use the arrow keys to specify how much to smooth the variations in tool axis tilting. The object is to minimize tilting without gouging the part.

Tool Axis Control tab for MultiBlade

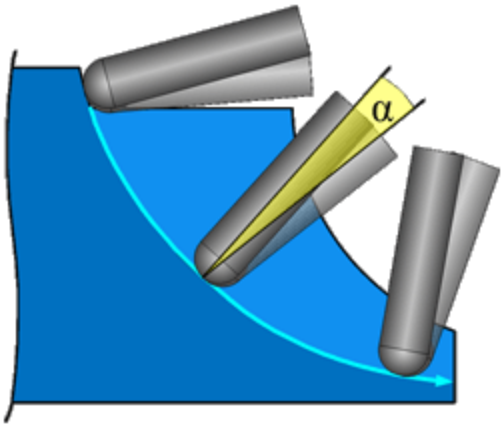
The controls presented in the MultiBlade Tool Axis Control page provide a rearranged subset of the regular 5-Axis tool axis controls for tilt angles and gouge check controls for tool clearances.

Tilting Settings

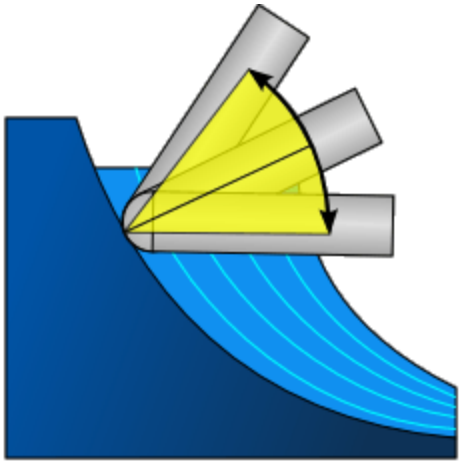
The settings under Tilting specify tool axis angles with respect to the hub normal vector.

Strategy

At this release, the only strategy offered is Global lead angle.



Global lead angle



Dynamic lead angle

When you use Dynamic lead angle as your tilting strategy, you can specify values for:

- Preferred lead angle (same as Global lead angle): Specify a preferred forward tilt angle for the tool, relative to the hub surface, in the cutting direction.
- Minimum lead angle: The minimum amount that the tool axis can tilt forward.
- Maximum lead angle: The maximum amount that the tool axis can tilt forward.

As the illustration indicates, these parameters let you vary lead angles as needed for machining in cramped locations.

Side tilt angle

Specify the maximum angle for the tool to tilt to the side of the cutting direction, towards the blades, to avoid gouging. At 0 degrees, the tool will be oriented perpendicular to the hub surface.

Tilt around toolpath

Available only for roughing.

Limits Settings

Available only if you are licensed for 5-Axis MultiBlade Level 2.

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. If this limit prevents the tool from reaching some areas of the toolpath, then that portion of the toolpath will be trimmed.

Maximum angle step

This value sets the maximum allowed angle change between two adjacent toolpath positions; it must be greater than 0 degrees. The toolpath calculation will prevent any tool axis vectors from having an angle change greater than the value specified.

Maximum angle step for rapid moves

This value sets the maximum allowed angle change between two adjacent rapid moves.

Tip: Decreasing the value for Maximum angle step generates more points; increasing it generates fewer points.

Warning: Collisions are checked only *at* tool positions, not between positions. Therefore, if Maximum angle step is set to a large value, the system might not detect some collisions between positions. If this occurs, use a smaller value here.

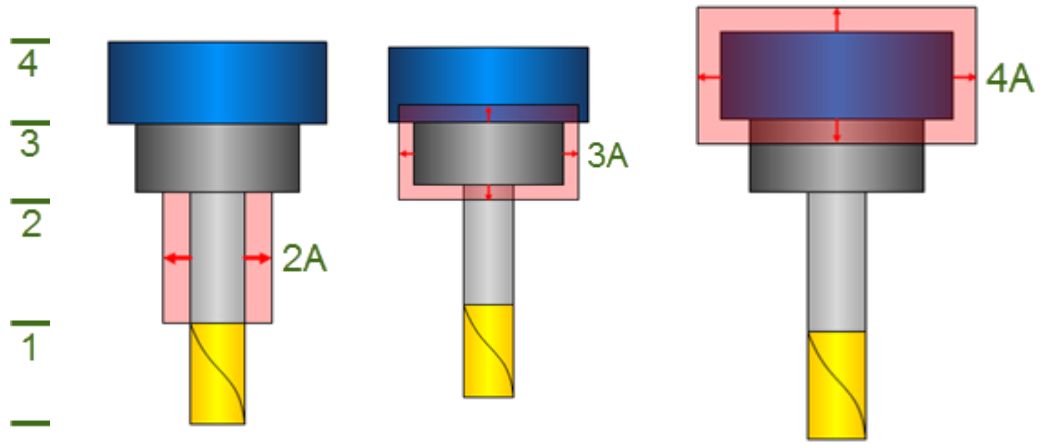
Clearances Settings

The system looks at the holder back, holder front, and shaft as simple cylinders, no matter what the shape actually looks like. These clearances are a virtual stock added to the diameters of your holder back, holder front, and shaft.

If the surfaces specified in the **Part Definition** tab have a value set for Stock to leave, then the clearance and Stock to leave values are added together to keep the holder front away from the part by that distance. For example, if the holder front clearance is 0.2 and you set Stock to leave to 0.5 on the surfaces, then the holder front is not allowed to come closer than 0.2 + 0.5 = 0.7 to the part.

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

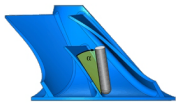


1. Flute
2. Shaft
3. Holder Front
4. Holder Back

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

Clearance angle

Specify a minimum clearance angle between blade and tool.



α = Clearance angle between blade and tool

Link tab for MultiBlade

The controls presented in the MultiBlade Link page provide a subset of the regular 5-Axis controls for links between slices and links between layers, in addition to parameters for part clearance area, feed distance, and home position.

Automatic
If you want to keep the default system values (recommended), keep this checkbox selected. Clearing the checkbox allows you specify nondefault settings for all of the following.

Links Between Cuts/Slices

The choices for Link between cuts and Links between slices (available only when By maximum number is greater than 1 under **Surface Paths** > Layers) refer to settings farther below on the **Link** page. Each slice is a cut approximately parallel to the blades, proceeding from one edge to the other. Each layer is a series of cuts where the entire set proceeds inward from shroud to hub.

If you clear the Automatic checkbox, choose an option from the pull-down menu to specify how to link from one cut (or slice) to the next:

- Choose Direct line to proceed directly from the end of one cut to the start of the next, optionally with a plunge arc.
- Choose Blend spline to blend the lead-out of the last cut into the lead-in of the next, optionally with a plunge arc.
- Choose Feed distance to retract by the value specified in the Distances settings described below, optionally with a plunge arc.
- Choose Clearance to retract to the location implied by the Clearance settings described below, optionally with a plunge arc.

Clearance Settings

Use
Choose whether to use a cylindrical or spherical clearance envelope around the part.

Sphere / Cylinder radius
Specify a value measured from the rotary center of the part.

Sphere center height
Specify a value the places the center of the sphere at the center of the part (height=0) or above or below it (positive or negative).

Autodetect dimension and position
By default, this checkbox is selected, causing the system to calculate the radius. If you clear the checkbox, manually specify a value for Cylinder radius.

Distances Settings

Feed distance
Specify how far to retract the tool from the drive surface before approaching the next cut.

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.

Edges tab

The controls presented in the MultiBlade Edges page give you control over the tool behavior at the leading edge and trailing edge of the blade and the trailing edge of the splitter. Usual practice is to use Edge rolling to trim the toolpath back from the edge by at least a half tool radius, and then to use Edge extension to extend the toolpath slightly past the edge.

Edge Rolling Settings

Edge rolling
Full (without trimming): The tool will roll around the entire leading/trailing edge over to the other side of the blade. *Available only if you are licensed for 5-Axis MultiBlade Level 2.*

Auto (trim by tool radius): Toolpath will be trimmed when the radius of the leading edge or trailing edge exceeds the tool radius. *Available only if you are licensed for 5-Axis MultiBlade Level 2.*

Trimmed by length
Specify how far back to trim the toolpath at the leading edge and trailing edge.

Trimmed by angle *Available only if you are licensed for 5-Axis MultiBlade Level 2.*
Imagine the leading edge and trailing edge extended in their natural directions. Take this vector and specify an angle away from it in direction of the cutting side. Trimming occurs when the angle you specify is reached.

Edge Extension Settings

Tangential
Tangential extension means that the toolpath is extended in the cutting direction.

Radial
Radial extension means that the toolpath is extended at the leading edge directly towards the center of rotation, and extended at the trailing edge directly away from it.

Edge Tilting Settings

Available only if you are licensed for 5-Axis MultiBlade Level 2.

Keep tilt angle within distance

Using 5-Axis MultiBlade

This part of the documentation covers the following areas:

- Getting Started with MultiBlade – Provides general guidance on steps to take for programming an impeller or blisk using 5-Axis MultiBlade.
- "Sample Part: MultiBlade Impeller" on page 23 – Provides specific examples of typical actions and settings for the sample part `MultiBlade Sample Impeller1 mm.vnc`.

Getting Started with MultiBlade

After you open an impeller or blisk part, choose a sphere-tip tool (Sphere mill, Lollipop, or Taper mill), 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 MultiBlade; under Pattern, choose Roughing and Morph between shroud and hub (with one exception: if the shroud of your part is not a revolved surface, your choice for Strategy must be Offset from hub). You can come back later and change your choices after you see results.

   Under Layers, specify By maximum number = `1` for now. When you are close to completing the Roughing operation, you will come back and change this to a realistic value.

2. Next, in the **Part Definition** page: Define the drive surfaces for one segment (hub, blades+fillets, and so forth) and specify the total Number of segments in the part.

   Under Segments, for Machine, set Determined number = `1` for now (you will come back and change this).

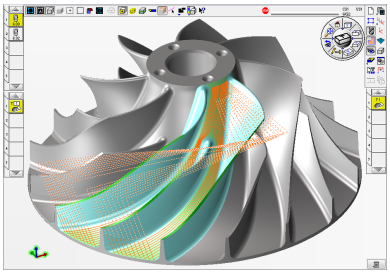
   Under Quality, enter a loose value for Machining tolerance for now, such as `0.1` mm or `0.005` inches. (You may need to tighten this to avoid gouging.) When you are close to completing the Roughing operation, you will come back and change this to a realistic value.

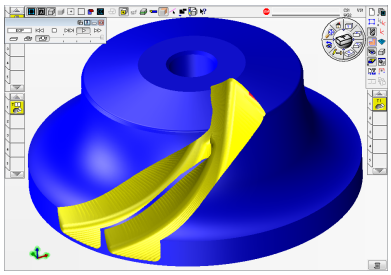
   If you are performing rest machining from an existing body, select Stock definition, choose the stock surfaces, and supply a value for Stock offset. *Available only if you are licensed for 5-Axis MultiBlade Level 2.*

3. Next, in the **Tool Axis Control** page: Under Clearances, specify realistic values for the current tool.

   For now, under Limits, specify a fairly large Maximum angle step of `3` or `5` degrees. Later, you might decrease this value in order to generate more tightly controlled toolpath.

4. Generate the operation (one segment, one layer, several slices). Render it and inspect the preliminary results.



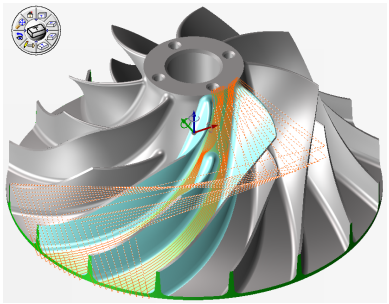


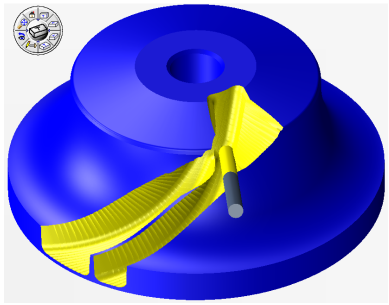
5. Refine settings as needed. For example:

   - If you misdefined any portion of the part or check surfaces, return to the **Part Definition** tab and CTRL-select to add or remove surfaces. For example, the hub should also include the small "collar" surface just above the leading edges.
   - If gouging is occurring, try one or more of the following, depending on what you see:
     - In the **Part Definition** and **Tool Axis Control** pages, supply larger values for Stock to leave and offsets/clearances.
     - In **Part Definition**, supply a tighter value for Machining tolerance.
     - In **Tool Axis Control**, supply a smaller value for Maximum angle step.
     - In **Link**, supply a more generous value for Clearance or Feed distance.

6. Repeat the previous step as needed, generating and rendering toolpath for one layer of one segment, until you are pleased with the result. If appropriate, experiment with one or more of the following:

   - In **Edges**: Supply values for Edge Rolling (to trim back toolpath) and Edge Extension (to smoothly extend toolpath), fine-tuning tool behavior at the leading edges and trailing edges of the blades and splitters.
   - In **Part Definition**: Under Quality, try adjusting the sliders for smoothing. *Available only if you are licensed for 5-Axis MultiBlade Level 2.*
   - In **Surface Paths**: Try out different choices under Sorting.
   - In **Surface Paths**: Try out different values under Slices and possibly Area.



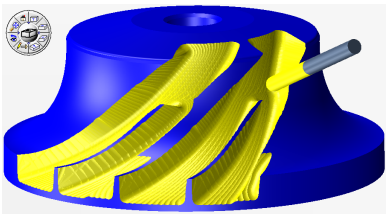


7. When you are fully satisfied with the toolpath for one layer of one segment: In **Surface Paths**, under Layers, increase By maximum number) to 2 or 3. In **Link**, choose a method for Links between layers. Regenerate the toolpath. Experiment as needed with other values and methods.

8. In **Part Definition**, under Segments, set Determined number to 2 or 3, and specify choices for Start angle and Direction. Regenerate the toolpath. When you are satisfied, increase Determined number to equal the value for Number of segments.

   You might also want to experiment with choices for Sort by to see which one maximizes the production for a particular setup. *Available only if you are licensed for 5-Axis MultiBlade Level 2.*

9. As needed, in **Surface Paths**, try out realistic values under Layers and Slices, possibly adjusting other settings such as offsets/clearances/tolerances in **Part Definition**, clearances in **Tool Axis Control** and **Link**, and choices for linking between and slices and layers and sorting them.



Because multiple passes (under Layers) and low values for Machining tolerance increase CPU usage, these are usually the last parameters you will change as you complete your Roughing operation.

10. After you complete your Roughing operation, create new operations with Machining set to Blade finishing and then to Hub finishing.

    Also create operations for Fillet finishing as needed. *Available only if you are licensed for 5-Axis MultiBlade Level 2*. You will need to use a tool of much smaller radius, of course, and you will also need to set a tighter value for Machining tolerance (in **Part Definition**, under Quality).

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

Sample Part: MultiBlade Impeller

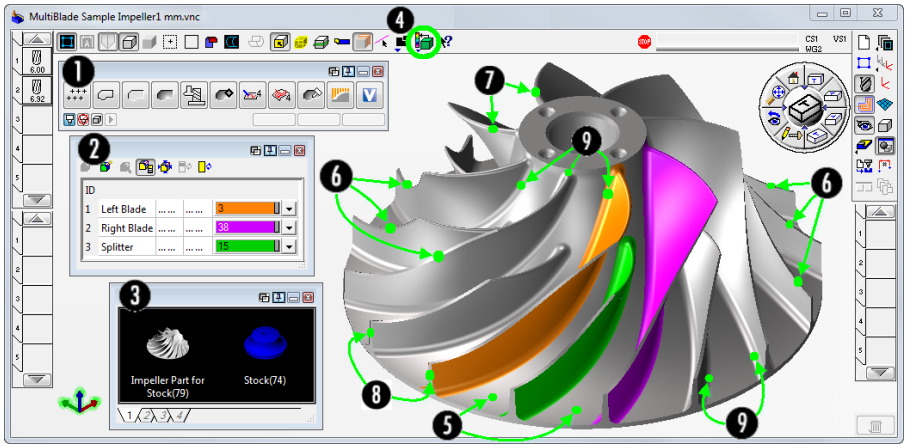
Sample parts are provided on the installation DVD. They can also be downloaded from https://online.gibbscam.com (login required).

For 5-Axis MultiBlade, the sample part name is `MultiBlade Sample Impeller1 mm.vnc`.

- Step A: Open and Explore the Sample Part
- "Step B: Create and Set Up a 5-Axis MultiBlade Process" on page 24
- "Step C: Provide Initial Settings for MultiBlade Processes" on page 25
- "Step D: Generate, Inspect, and Refine Initial Results" on page 26

Step A: Open and Explore the Sample Part

1. Start GibbsCAM, navigate to the folder containing sample parts, subfolder `5-Axis MultiBlade\`, and open the sample part (`MultiBlade Sample Impeller1 mm.vnc`.).
2. Open the tool list, the Machining palette, the Body Bag, and Feature Manager.
3. Temporarily change to Feature Color mode.
4. Become acquainted with the part and its component surfaces: hub, shroud, main blades and splitter blades, fillets, and so forth.
5. When finished, change back to Gibbs Color mode.



1. Machining palette
2. Feature Manager:
   Feature #1 (orange): left blade
   Feature #2 (pink): right blade
   Feature #3 (green): splitter
3. Body Bag
4. Feature Color (icon)
5. Hub surface
6. Shroud surface
7. Leading edges of blades
8. Trailing edges of blades
9. Fillets joining hub and blades

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

1. From the Tools List, drag the tile for Tool 1 onto the blank process tile. From the Machining palette, drag the 5-Axis function tile onto the same process tile.

   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 Multi blade parts. Notice how the tabs to the right change: Three tabs disappear (**Gouge Check**, **Roughing**, and **Utility**), and one new tab is added (**Edges**).

Step C: Provide Initial Settings for MultiBlade Processes

1. Explore the **Surface Paths** settings. For example:
   - Under Pattern, several strategies are available for Roughing and Blade finishing, and you can specify choices and values for Layers.
   - For Roughing and Hub finishing, you can specify choices and values for Slices.
   - Fillet finishing offers a completely different set of parameters. *Available only if you are licensed for 5-Axis MultiBlade Level 2.*
2. Before moving on to other tabs, set the following:
   - Pattern / Machining: Roughing
   - Pattern / Strategy: Morph between shroud and hub
   - Layers / By maximum number: 1
     Although this is an unrealistic value, it simplifies visualization and greatly reduces the time required to generate toolpath. Later, you will come back and set this to a more realistic value.
3. Click the **Part Definition** tab and explore the settings. Before moving on to other tabs, set the following:
   - Blades,splitter,fillets: Click the ellipsis button (…). When prompted to select impeller blade surfaces, click all three of the lines in Feature Manager to select the surfaces associated with the two main blades (Left Blade and Right blade) and splitter blade, and then click OK. The blade surfaces are displayed in pale blue. Set an appropriate value for Stock to leave, such 0.2 mm.

Tip: When using Feature Manager as a shortcut for selecting surfaces, double-check the list shown in the Select [...] Surfaces dialog box to be sure it includes items of type Face only and does not accidentally include curves or points.

   - Hub: Click the ellipsis button (…). When prompted to select impeller hub surfaces, click the large surface between any two blades, and CTRL-click the small “collar” surface just above the leading edges of the main blades. Because the hub is a surface of revolution, the system is now acquainted with the entirety of the hub and its fillets with the blades. The hub

surfaces are displayed in pale green. Set an appropriate value for Stock to leave, such 0.2 mm.

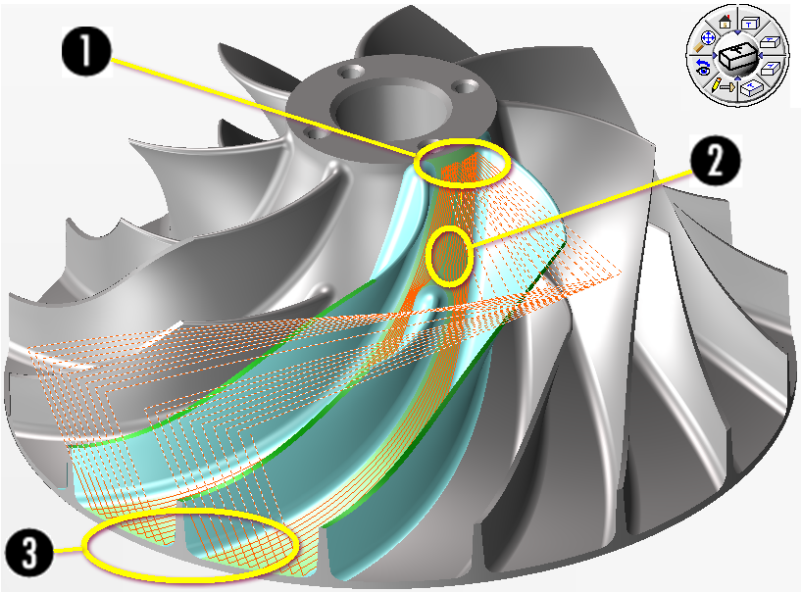
- Shroud: Click the ellipsis button (...). When prompted to select impeller shroud surfaces, click any one of the thin outer surfaces connecting leading edge to trailing edge. Because the shroud is a surface of revolution, the system is now acquainted with the entirety of the shroud and its intersection with leading and trailing edges of all blades. The shroud surfaces are displayed in dark green.
- Number of segments: Set to 8 for this part, which has 8-fold symmetry.
- Segments / Machine: Choose Determined number and set to 1 for now. Later, you will machine multiple segments and experiment with direction and sorting.
- Quality / Machining tolerance: 0.1 mm
  Although this is an unrealistic value, it reduces the time required to generate toolpath. Later, you will come back and set this to a more realistic value.

4. Click the **Tool Axis Control** tab and explore the settings. Under Clearances, specify realistic values for the current tool, such as:
   - Shaft: 0.2 mm
   - Holder front: 0.5 mm
   - Holder back: 2 mm
   - Clearance angle: 0 degrees
5. Next, in the **Tool Axis Control** page: Under Clearances, specify realistic values for the current tool.

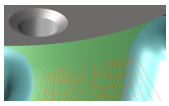
   For now, under Limits, specify a fairly large Maximum angle step of 3 or 5 degrees. Later, you might decrease this value in order to generate more tightly controlled toolpath.

Step D: Generate, Inspect, and Refine Initial Results

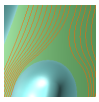
1. Generate the operation and render it. Rotate and zoom the preliminary results to inspect them closely.



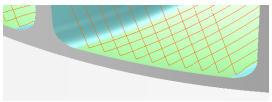
1. Trimmed toolpath at leading edge should extend beyond the top of the hub.
   In the **Edges** page, increase Edge extension for Leading edge.

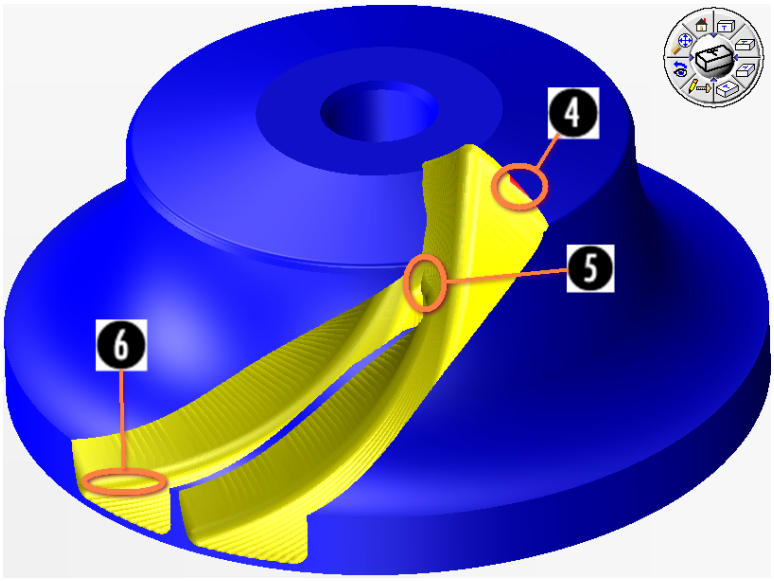


2. Where toolpath divides for splitter, adjustments may be needed.
   In the **Part Definition** page, adjust the Splitter flowline smoothing.



3. Trimmed toolpath at trailing edge should extend beyond the outermost hub.In the **Edges** page, increase Edge extension for Trailing edge.





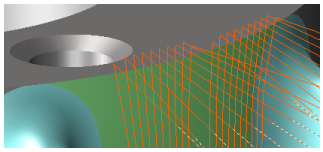
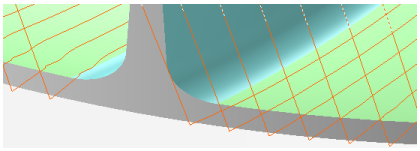
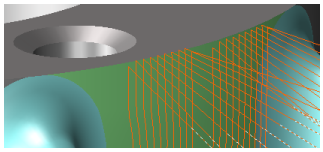
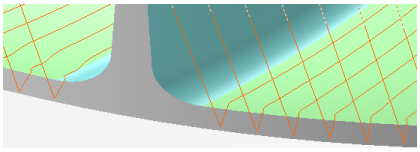
4. No gouging should occur.>
   In the **Tool Axis Control** page, tighten the values under Limits and Clearances. In the **Part Definition** page, tighten the Machining tolerance.

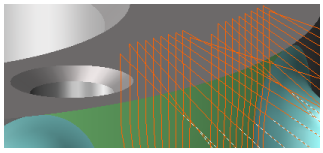
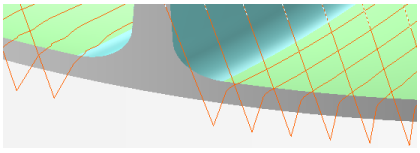
5. Fillet at leading edge of splitter should be rounded.
   After you complete the Roughing operation, you will create operations for Blade finishing and for Fillet finishing.

6. Fillets near trailing edge of blades may need attention.
   After you complete the Roughing operation, you will create an operation for Fillet finishing.

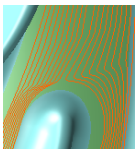
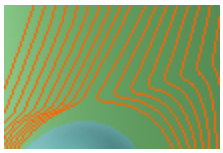
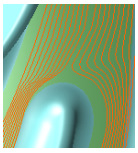
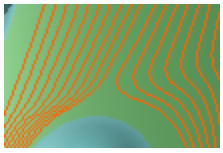
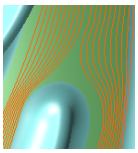
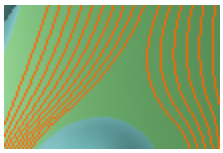
7. Refine settings as needed. For example:

   - If toolpath does not extend beyond the leading edge and trailing edge of the hub: In **Edges**, supply positive values for Edge extension.

Leading Edge	Trailing Edge	Edge Extension
		Tangential 3mm
		Radial 2mm

		Tangential 1mm and Radial 3mm

- To adjust toolpath as it divides approaching the splitter, use the Splitter flowline smoothing slider in the **Part Definition** page:
  - For closest approach to the fillet, *decrease* the smoothing.
  - To reduce jerky acute angles, *increase* the smoothing.

Fillet	Close up	Splitter flowline smoothing
		Splitter flowline smoothing: 0.70%
		Splitter flowline smoothing: 3.00%
		Splitter flowline smoothing: 20.0%

- If gouging is occurring, try one or more of the following, depending on what you see:
  - In the **Part Definition** and **Tool Axis Control** pages, supply larger values for Stock to leave and offsets/clearances.
  - In **Part Definition**, supply a tighter value for Machining tolerance.
  - In **Tool Axis Control**, supply a smaller value for Maximum angle step.
  - In **Link**, supply a more generous value for Clearance or Feed distance.

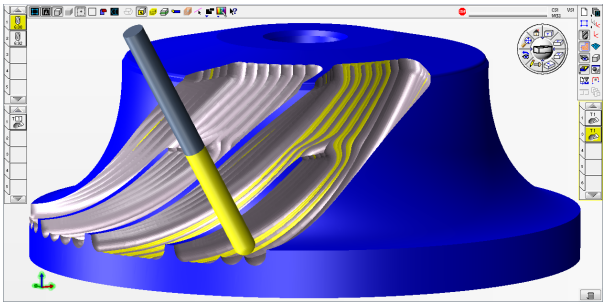
Step E: Increase Layers and Segments, with Sorting

1. When you are fully satisfied with the toolpath for one layer of one segment:
   - In **Surface Paths**, under Layers, increase By maximum number) to 2 or 3.
   - In **Part Definition**, under Segments, increase Determined number) to 2 or 3.
   - In **Link**, choose a method for Links between layers.

2. Regenerate the toolpath. Experiment with other methods for the following:

   - In **Part Definition**, under Segments, see the effect of trying a different method for Sort by. Maybe also try changing Direction.
   - In **Surface Paths**, under Sorting, experiment with different methods for starting and joining the slices and for ordering the slices.
   - In **Link**, try out other choices for Links between slices and Links between layers.
   - Try out the idea of taking two or more roughing passes, where the first one leaves stock on the blades/splitters/fillets, uses a looser Machining tolerance and fewer slices, and machines only part of the area. In the last roughing pass, use a large number of slices and specify Zig zag as the sorting method.

Sample Options: Two Successive Roughing Operations



For both operations:
Pattern / Machining = Roughing;
Segments / Machine Determined number = 2
Start At segment = 1;
Links: Automatic

1. **(silver):**
Layers / By maximum distance = 2 mm
Slices / By maximum number = 4
Area / Start at = 60%;
Part definition / Stock to leave = 0.4 mm
Quality / Machining tolerance = 0.1 mm

2. **(yellow):**
Layers / By maximum distance = 5 mm
Slices / By maximum number = 10
Sorting / Method: Zig-zag
Part definition / Stock to leave = 0.1 mm
Segments / Sort by: Layer
Quality / Machining tolerance = 0.05 mm

3. Regenerate the toolpath as needed. When you are satisfied with your choices, generate toolpath for the entire part with realistic values:

   - In **Part Definition**, under Segments, change Machine to All and decrease Machining tolerance to 0.01 mm.
   - In **Surface Paths**, under Layers, specify a realistic value for this part, such as By maximum number = 15 layers total, or By maximum distance = 1 mm.

Because multiple passes (under Layers) and low values for Machining tolerance increase CPU usage, these are usually the last parameters you will change as you complete your Roughing operation.

Step F: Create Finishing Operations

1. After you complete your Roughing operation, create new operations with Machining set to Blade finishing and then to Hub finishing.
2. Also create operations for Fillet finishing as needed. *Available only if you are licensed for 5-Axis MultiBlade Level 2.* You will need to use a tool of much smaller radius, of course, and you will also need to set a tighter value for Machining tolerance (in **Part Definition**, under Quality).

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

Troubleshooting / FAQs

This part of the documentation is intended to answer questions of the following types.

Why Can't I ...?

Question. Why can't I see or activate MultiBlade?
Licensing. 5-Axis MultiBlade is licensed separately from 5-Axis. If you do not have a license for 5-Axis MultiBlade, you will not see it as an option in the regular 5-Axis interface.

Question. Why can't I see or use 5-Axis MultiBlade Level 2 controls?
Licensing. 5-Axis MultiBlade Level 2 is licensed separately from 5-Axis MultiBlade Level 1. If you do not have a license for 5-Axis MultiBlade Level 2, you will not see its capabilities within the MultiBlade interface.
Checkbox. If you are licensed for 5-Axis MultiBlade Level 2, a checkbox (Advanced) appears near the top of the Surface Paths page. If this checkbox is not selected, you will not see the Level 2 capabilities within the MultiBlade interface.

Question. Why can't I use all tool types?
Tool Types. 5-Axis MultiBlade supports only sphere-tip tools: Sphere mills, Lollipop mills, and Taper mills.

Question. Why can't I see balloons or tooltips for MultiBlade controls?
Balloons. Instead of short balloon text, MultiBlade provides context-sensitive help: On the main palette, click the question mark (or, on the Help menu, select Help On Item) and then click the MultiBlade control to summon documentation for the interface item.
Tooltips. Tooltips are provided only for toolbars across the top of dialog boxes. The process dialog for MultiBlade provides all controls within the tabbed pages without launching other dialogs.

How Do I ...?

General information on using 5-Axis MultiBlade and the sample part is supplied in "Using 5-Axis MultiBlade" on page 21.

Question. How do I ...?
5-Axis MultiBlade is a new product, and this section will be expanded as we learn more about commonly asked questions.

Why Did It ...?

Question. Why did it take so long to load the part file?
Question. Why did the part file become so much bigger?
Most impeller and blisk parts, including the sample part provided for MultiBlade, make extensive use of faceting. This saves CPU time at the expense of file size. To save space, the sample part is saved without facets (3.5MB file size). Loading it requires a longer time to expand it to the faceted

form (30MB). Depending on your preferences (File > Preferences > File tab > Save Size), you save the model either as Large (the default) to minimize load time, or as Minimal (without facets) to minimizing size on disk.

Question. Why did it take so long to generate toolpath?

Processing time depends most heavily on number of layers and on overall machining tolerance. When you are planning the part programming, you can greatly reduce CPU usage by temporarily setting Layers (By maximum number) to 1 and Quality to use a loose value for Machining tolerance. The table shows how dramatically this can reduce CPU cost.

Time	Layers	Quality
186	By max number = 10	Machining tolerance = 0.01 mm
24	By max number = 10	Machining tolerance = 0.1 mm
16	By max number = 1	Machining tolerance = 0.01 mm
2	By max number = 1	Machining tolerance = 0.1 mm

In the **Part Definition** page, values for Stock to leave and Quality > Smoothing have little or no effect on CPU usage. But when you machine multiple layers, the following can also reduce CPU time to a minor extent:

- Segments / Machine. To minimize CPU time, use Determined number = 1.
- Limits / Maximum angle step. To minimize CPU time, use a larger value, such as 3 degrees.

Question. Why did it generate One way toolpath when I specified Zig zag?

When you specify Zig zag and the system cannot satisfy all parameters for tolerances, stock, tool tilting, angle limits, and so forth, it will first try to generate One way toolpath if it can do so. If it cannot generate One way toolpath, it will display an error in the 5-Axis Task Manager.

Question. Why did context-sensitive help (Help > On Item) display the wrong topic?

The scope of the context is the control you click, such as a checkbox, option, or key-in area. No general context is associated with items like page background or tab names.

Question. Why did it generate toolpath only near the leading edges?

Perhaps the Clearance angle (Tool Axis Control tab) is too large, preventing the tool from entering tight regions.

Question. Why did it generate toolpath on only one segment half, between the right splitter surface and the left blade surface?

Question. Why did ...?

5-Axis MultiBlade is a new product, and this section will be expanded as we learn more about commonly asked questions.

Where Is Information on ...?

You can find information within this document using the ToC (Contents) and Index. From within MultiBlade, you can use context-sensitive context-sensitive help: On the main palette, click the question mark (or, on the Help menu, select Help On Item) and then click the MultiBlade control to summon documentation for the interface item.

Question. Where is information on ...?

5-Axis MultiBlade is a new product, and this section will be expanded as we learn more about commonly asked questions.

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

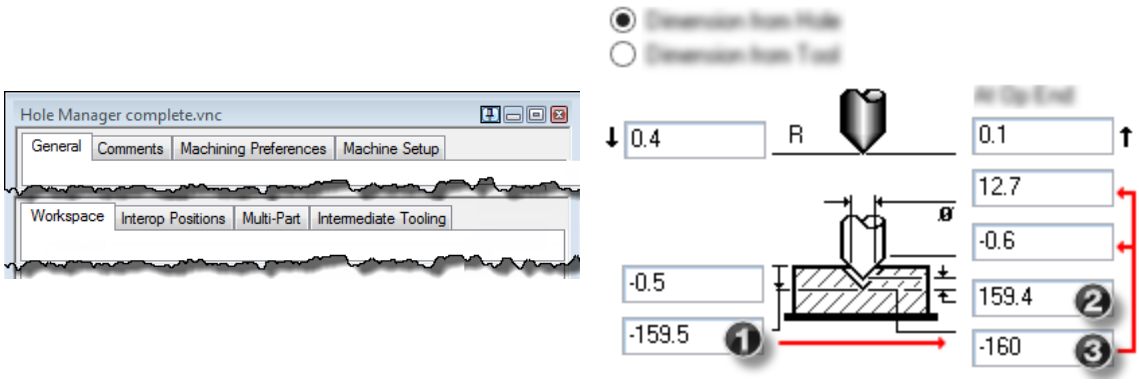
`Screen text`. Text with `this appearance` indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

`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

F

G

H

I

K

L

M

N

O

P

Q

R

S

T

Z