What's New in GibbsCAM 2026 — BETA

Note: Do NOT use BETA version in production!

Contents

Highlights

Some of the most important enhancements in GibbsCAM 2026 include the following:

- **GibbsCAM Viewer**
  For details, see "GibbsCAM Viewer" on page 6.
- **Improved UI for B-Axis Turning**
  For details, see "Variable-B Turning: Improved UI" on page 8.
- **Mill>Contour**: Deburr/chamfer of solid part edges
  For details, see "For technical details, such as running Industrial and Viewer on the same workstation, PowerOpts, and NLO considerations, see the ***Installation*** guide.
- Milling" on page 6.
- **Sim>Settings** > Program Time: Option for Sim replay time to match machining time
  For details, see "Visualization and User Interface" on page 10.

This "Highlights" section is just a small subset of the enhancements in GibbsCAM 2026. The next section ("Overview") provides a complete list.

Overview of What's New in GibbsCAM 2026

The significant enhancement to **Milling** in GibbsCAM 2026 is:

- Automated deburring/chamfering of solid part edges

For details, see "For technical details, such as running Industrial and Viewer on the same workstation, PowerOpts, and NLO considerations, see the ***Installation*** guide.

Milling" on page 6.

Important enhancements to **Turning** include:

- Variable-B Turning: Improved user interface
- Collision-avoidance material calculation for multi-sided tools (e.g., CoroTurn Prime)
- MTM Utility Ops: Extended Move Toolgroup for tool subpositions

For details, see "Turning" on page 8.

Enhancements to **Visualization and User Interface** include:

- Undo/Redo: Visualization and navigation through Undo/Redo history, listed by action type
- Drag-and-drop and custom png or jpg image to represent a DCD's machinetool (MDD)
- Preference for displaying MDD list sorted alphabetically by name
- Efficient Shrink Wrap with cylindrical stock
- Modify>Rotate: New command for nonplanar (out-of-plane) rotations
- View>Visibility>Measure Measure/analyze/display min|max distance between elements
- Sim>Settings Update Control: Options for Sim replay proportional to machining time
- Sim(Op,Mach): Visualization: Optional metallic Rendering appearance

For details, see "Visualization and User Interface" on page 10.

Improvements to **Solids** include:

- Improved engine for Advanced 3D
- Modify > Scale for solids and sheets
- Modify > Rotate for nonplanar rotations

For details, see "Solids" on page 15.

Improvements to **Tooling** include:

- Tool List: Spreadsheet-like list view
- Tool List: Filter by toolgroup
- Tool Usage Reporter
- Support for non-round inserts in VoluTurn

For details, see "Tooling" on page 18.

Miscellaneous improvements and changes include:

- Updated system requirements (Microsoft desupport of Windows 10)
- Updated third-party-library support
- Change to default GibbsCAM paths
- New/modified plug-ins:
  - Auto CS Create: Option to automatically create CSes from holes, cylinders, and cones
  - New plug-in: TMiQ for AI-based tool management
  - New plug-in: ToolsUnited tool library
  - New plug-in: MachiningCloud tool library
  - Changed workflow and UI for up2parts autoCAM plug-in
- Machine Manager: Ability to clone toolgroups
- Machine Manager Clearance Volume option: Minimize Moves
- AP242 in menus for File>Import and File>Open

For details, see "Miscellaneous" on page 22.

5-Axis integrates ModuleWorks 2024.08, 2024.12, and 2025.04, enhancing the following areas:

For details, see "5-Axis" starting on page 24.

GibbsCAM Viewer

Customers have long requested a product for reviewing GibbsCAM parts and programming. Such a product can allow operators to better understand the parts they will be cutting, can help with shop floor communication, and can reduce scrap rates and the amount of effort needed for reports.

GibbsCAM 2026 provides this new product: **GibbsCAM Viewer**. Its users have the ability to directly access all part information without allowing toolpath generation, posting, or saving/exporting, etc. GibbsCAM Viewer and GibbsCAM Industrial share the same installer (product level is determined by license) and user interface: Existing GibbsCAM users can use Viewer with no learning curve or training.

Users of GibbsCAM Viewer can do all the following.

- Open parts (`*.vnc`) and packages (`*.gcpkg`)
- **Rendering**, including:
  - Toolpath (including color modes)
  - Wireframe geometry, Solids, Surfaces, and Facet Bodies
  - CS grid, stock, and origin
  - Dimensioning and PMI
  - **Show Clearance Planes**/**Volumes**
  - **Section View**
  - All **Simulation** modes
- All **View** and **Viewport** controls
- Measurement: Dimension creation (other than Text), **View** > **Measure**, etc.
- Solid and part query tools and plug-ins: **Show Taper Angle**/**Curvature**, **Get Draft Angle**, **Show Surface Normals**, **Solid Inquiry**, **Show Position**, **Find Ops**, etc.
- **Reporter** output
- Op, Tool, and Process tile lists (including tile list view modes)
- **Op Manager**, **Tool Manager**, and **Process Manager** (but without **Edit** capabilities)
- **Op Data dialog**
- **Op To Geometry** (to permit dimensioning)
- Utility markers (but not marker placement)
- Tool and Process dialogs (but with **Do It** and **Redo** unavailable)
- **Activate Ops** / **Deactivate Ops**
- **Feature Manager**, **Attribute Manager**, and **Hole Manager** (with restrictions)
- **CS List** / **WG List** (with restrictions)
- **Program Error Checker**
- Body Bag
- Printing
- Update Notification
- Documentation and Help: PDFs, Online Help, Balloons, Check for Updates, and Copilot

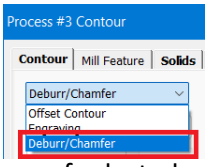
For technical details, such as running Industrial and Viewer on the same workstation, PowerOpts, and NLO considerations, see the ***Installation*** guide.

Milling

GibbsCAM 2026 provides an important Milling enhancement.

Mill Contour: Deburr/Chamfer of solid part edges

GibbsCAM 2026 adds a new capability to Mill **Contour**: Automated **Deburr/Chamfer**. (Requires license for 2.5D Solids or SolidSurfacer.) This replaces and supersedes the **Deburring** plug-in.



Where to find it: In the **Contour** dialog box, **Contour** tab, in pulldown menu, choose **Deburr/Chamfer**.

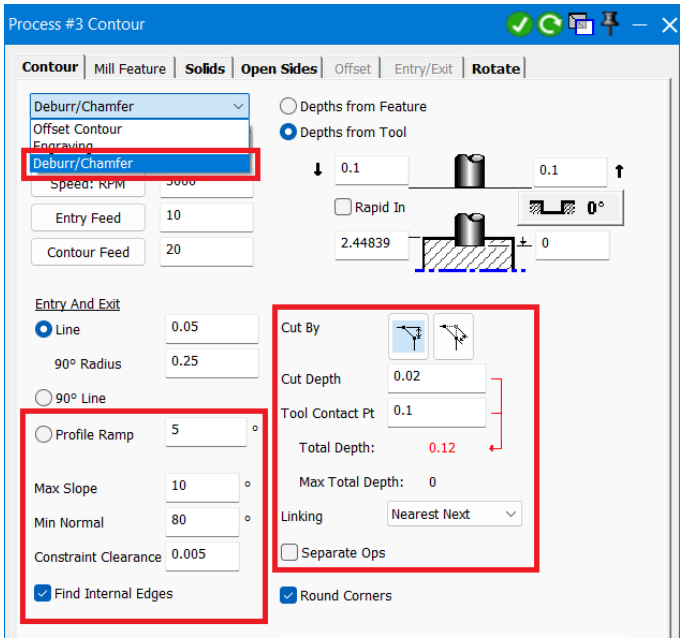
What it does: You can now deburr or chamfer edges that you manually select and/or edges of selected faces; or you can automatically deburr or chamfer all applicable edges on a selected solid body.

Note 1: This function operates on solids (it ignores sheets and facet bodies), and thus requires a license for 2.5D Solids or SolidSurfacer.

Note 2: This function expects appropriate tools:
(a) spherical tips (ball end mills, lollipop mills);
(b) concave round tips (roundover tools); or
(c) tools capable of chamfering with a tapered edge (drills, spot drills, and chamfer mills).
It does not support tools of arbitrary shape or complexity.

Note 3: When **Deburr/Chamfer** is active, the **Offset** tab is not needed and is thus intentionally disabled, and inapplicable functions in the **Solids** tab are suppressed.

For information on controls specific to Deburr/Chamfer, such as the edge-selection rules used by the checkbox **Find Internal Edges**, see the ***Mill*** guide.



Turning and MTM

Version 2026 enhances Turning and MTM in several important ways.

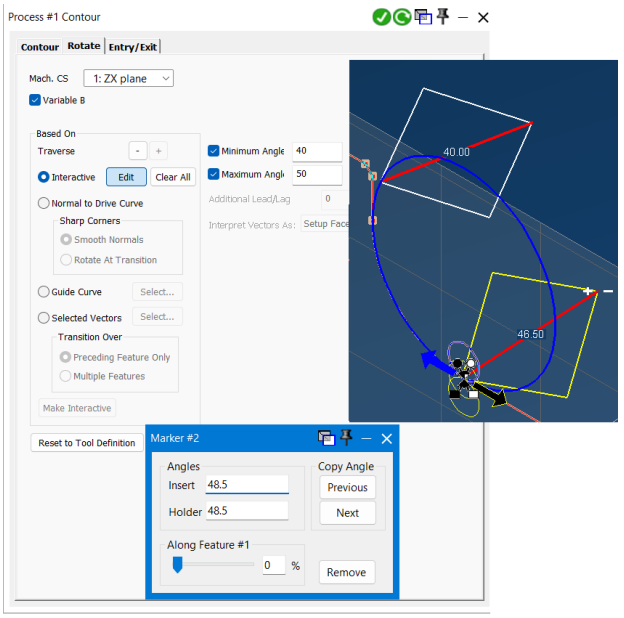
Variable-B Turning: Improved UI

New features make Variable B turning easier and more interactive.

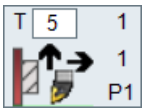
Where to find it: Turning **Contour** > **Rotate** tab > **Variable B** > (Based On) **Interactive**.

What it does: You can now directly control the tool angle at arbitrary points along the cut path by specifying control points along the path and using **Edit** and dragging the red line (+ or –) to set the tool angle of the tool at those points. The angle is automatically interpolated between control points.

For more information, see the ***Turning*** guide.

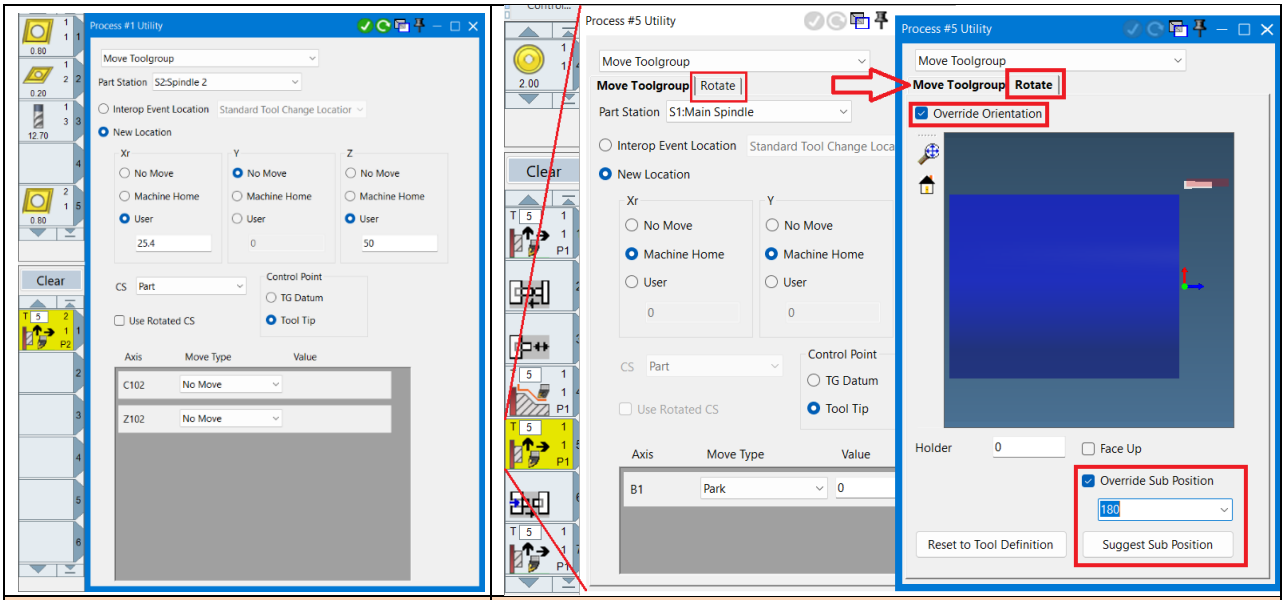


MTM/Utility Ops: Extended Move Toolgroup for tool subposition



Previous releases let you park tools using the Move Toolgroup (MTG) utility operation, but you could not select tool subposition. If parameters in a Lathe Contour operation's **Rotate** tab overrode tool subposition, an immediate MTG operation would default to the standard subposition, triggering an interop event because of the unintended change in tool subposition.

GibbsCAM 2026 remedies this with an enhancement to the extended Move Toolgroup utility process. In its new **Rotate** tab, tick the **Override Orientation** checkbox.



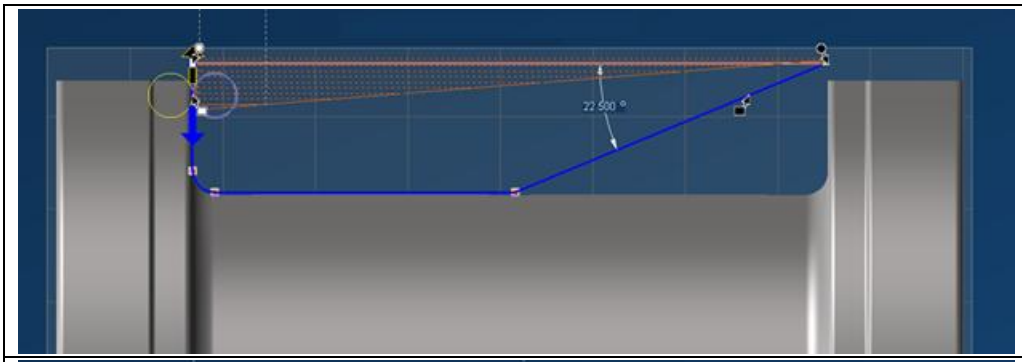
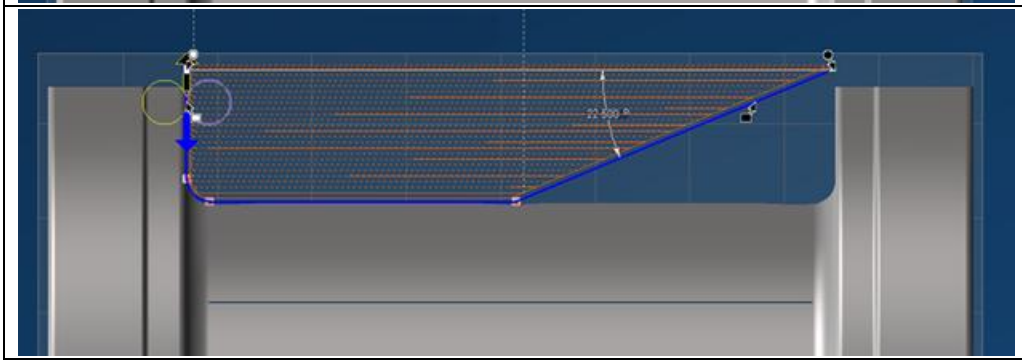
Before GC2026 | GC2026 MTG utility process dialog: **Rotate** tab

Improved toolpath for multi-sided Turning tools

In GibbsCAM 2026, toolpath generation is enhanced to provide material calculation to account for multi-sided Turning tools (MST), such as CoroTurn Prime inserts.

Before: In previous releases, with nonconvex tools, GibbsCAM was overly cautious in avoiding gouges and collisions, placing safety over complete material removal but sometimes leaving unmachined areas:

Now: GibbsCAM 2026 preserves safety while using more of the insert shape for Prime A and B tools.

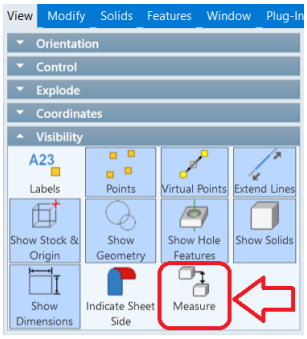
	Toolpath in previous releases
	Toolpath in GibbsCAM 2026

Also related to Turning

- VoluTurn support for non-round inserts ..... see page 19

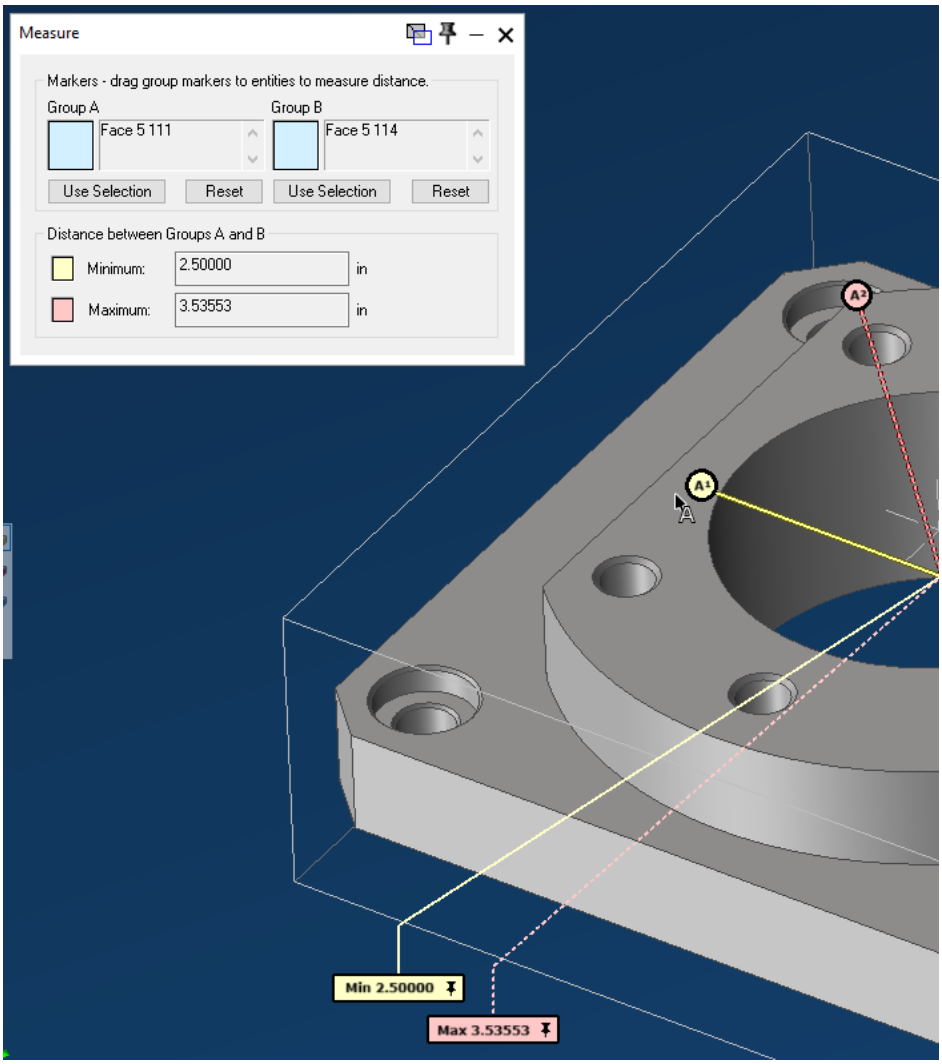
Visualization and User Interface

View > Measure minimum/maximum distance

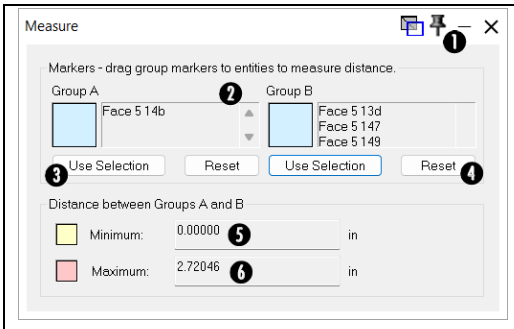


You can quickly measure minimum and maximum distance between any elements in your part, including 2D geometry (point, lines, circles, and splines) and the faces and edges of bodies (sheets, solids, and facet bodies).

Where to find it: On the **View** menu, under Visibility, click **Measure**.



Clicking **Measure** opens the **Measure** dialog box. To specify a pair of elements, you can drag and drop markers A and B onto the workspace and then reposition the marker as needed. And/or, you can select an element or a group of elements and click **Use Selection**. The minimum and maximum distances between A and B are displayed in the dialog box.



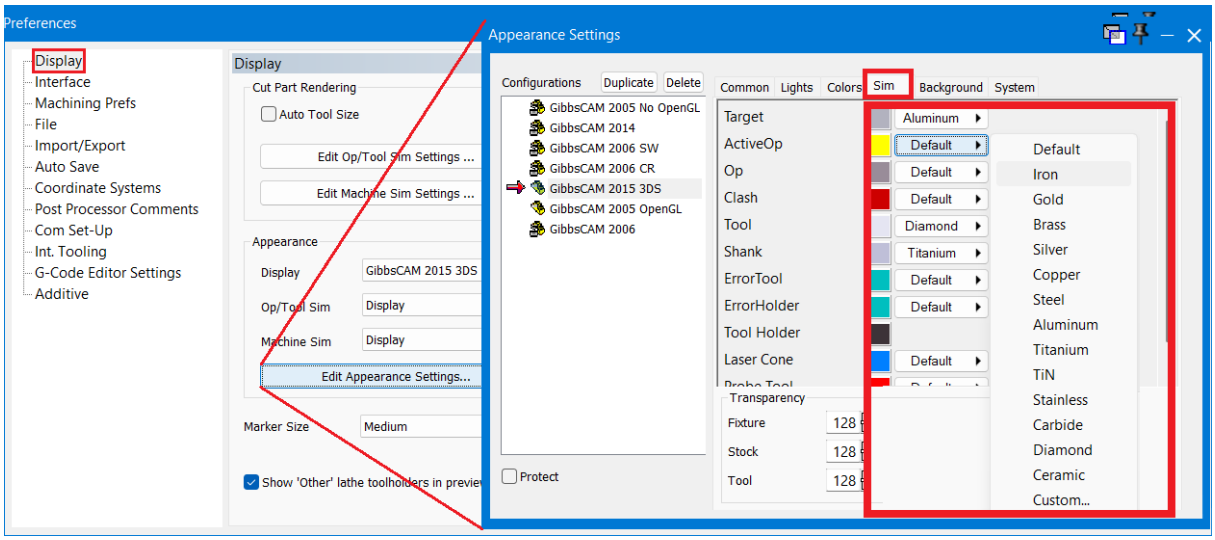
1. Pin() button
2. **Group A**, **Group B**:
   Each group lists the ID(s) of elements to be measured.
3. **Use Selection** —
   Replace current group items with selected element(s).
4. **Reset** — Remove all elements from this group.
   To reset both groups: Right-click, then choose **Reset**.
5. **Minimum** — Minimum distance between A and B.
6. **Maximum** —Maximum distance between A and B.

The minimum (yellow) and maximum (pink) distances are also displayed in the workspace as textboxes that can be repositioned. To lock the position of a textbox, click its pin button ().

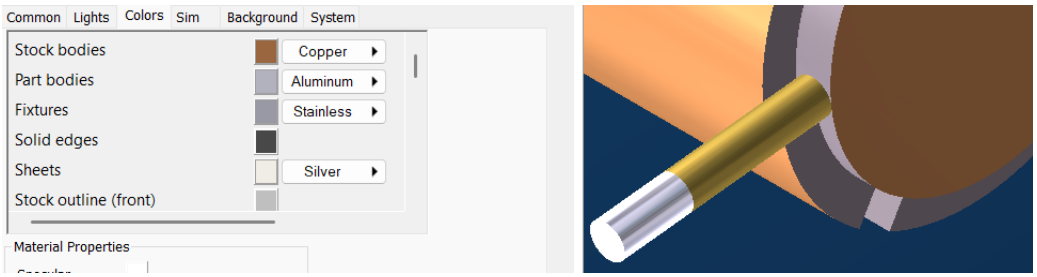
Preferences > Appearance Setting > Color|Sim > Metallic Rendering

You can now specify not just a color but optionally also a metallic appearance for Simulation items.

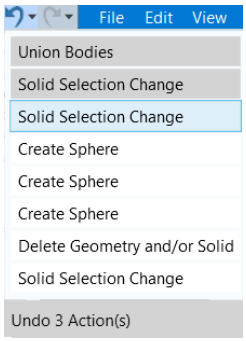
Where to find it: On the **File** menu, click **Preferences**; in the **Preferences** dialog, under Display, click **Edit Appearance Settings**; in the **Appearance Settings** dialog, click either the **Color** tab or the **Sim** tab.



Use the **Colors** tab to assign metallic rendering such items like workpieces, cutting tools/shanks/holders, intermediate tooling, fixtures, etc.; use the **Sim** tab for such items as Target, ActiveOp, Op, Clash, etc.



Undo/Redo pulldown menus: Navigate Undo / Redo history



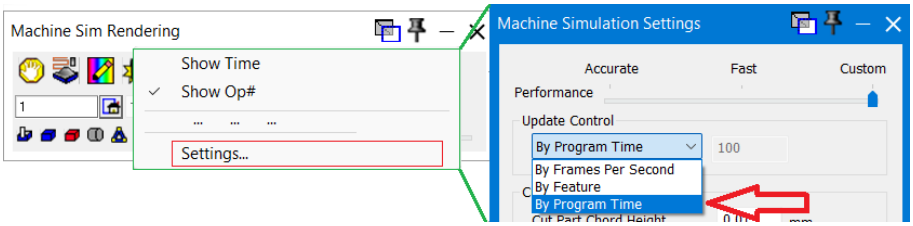
In the Quick Access toolbar (, just left of the File/Edit/View main menu), the **Undo** and **Redo** commands are now pulldown menus. Each pulldown provides a list of all the items that can be undone or redone. Dragging the cursor down the list provides a count of the number of items that will be undone or redone if a selection is made.

This new function greatly speeds and facilitates navigating to a particular action instead of having to pressing `Ctrl+Z` or `Ctrl+Y` repeatedly to undo or redo many actions.

Sim Replay Proportional to Machining Time

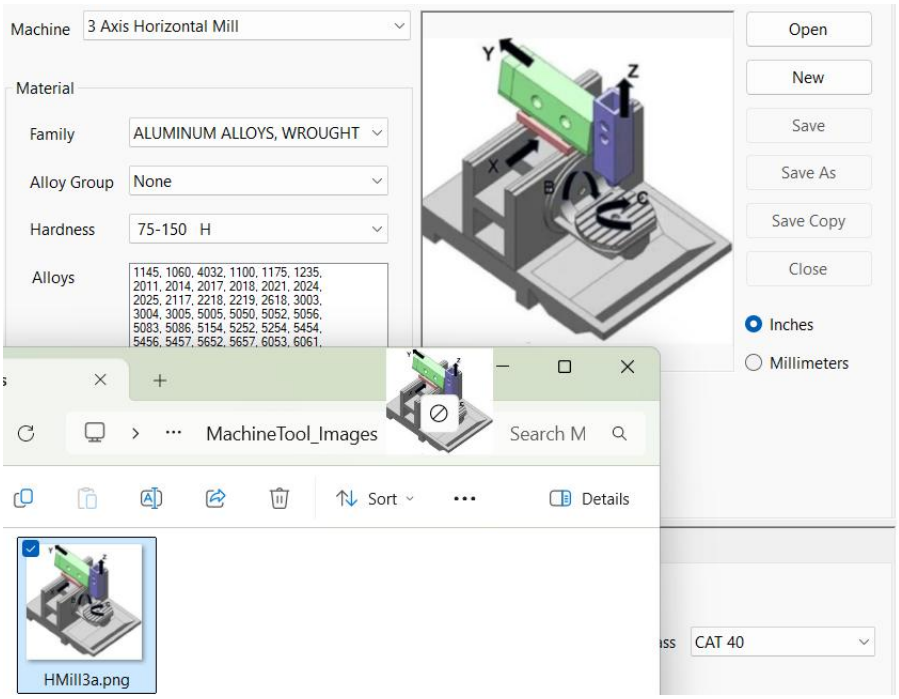
In the **Simulation Settings** dialog, a new option under Update Control lets you set the Sim replay to approximately match the actual machining time.

Where to find it: Right-click the titlebar of a **Sim Rendering** dialog; on the context menu, click **Settings**. In the **Simulation Settings** dialog, on the Update Control pulldown menu, choose **By Program Time**.



DCD: Custom image for each MDD (machinetool)

In GibbsCAM 2026, you can assign a custom image for the DCD for each MDD. Simply drag-and-drop an image file (*.png or *.jpg) onto the area of DCD, to the right of the machine description parameters and to the left of the **Open** / **New** / **Save** / … buttons.

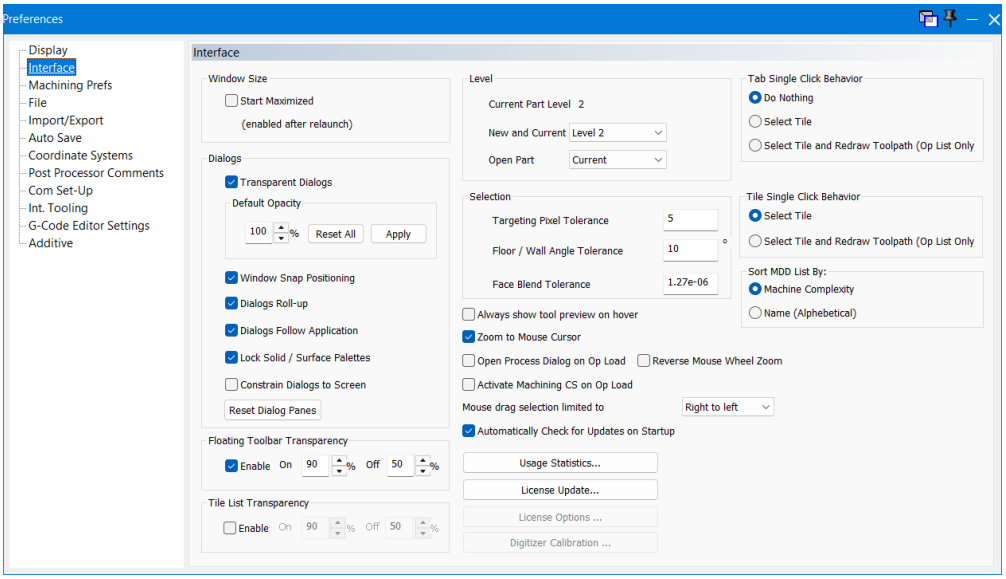


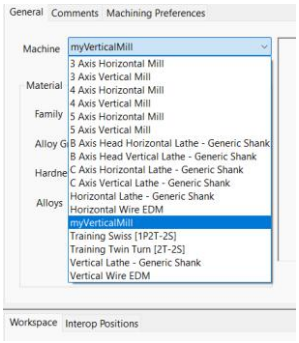
Alternatively, you can place an image file named the same as the MDD's menu inside the MDD\ folder.

Preference option to order MDD list alphabetically (by name)

You can now order the MDD list by name (or, as in previous releases, by machine complexity).

Where to find it: **File>Preferences** dialog, **Interface** tab, far right: **Sort MDD List By**.



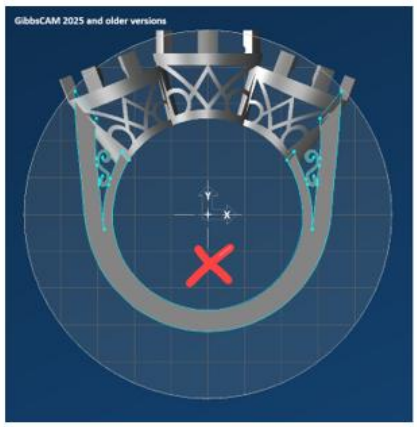


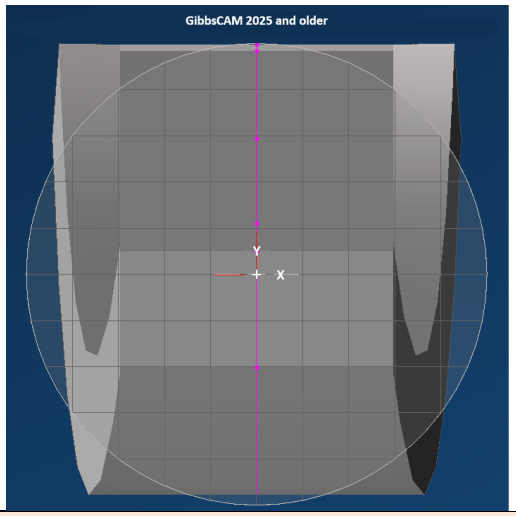
What it does: The list of MDDs (e.g., **DCD** > **General** tab, **Machine** pulldown) is ordered as specified by this **Preferences** setting.

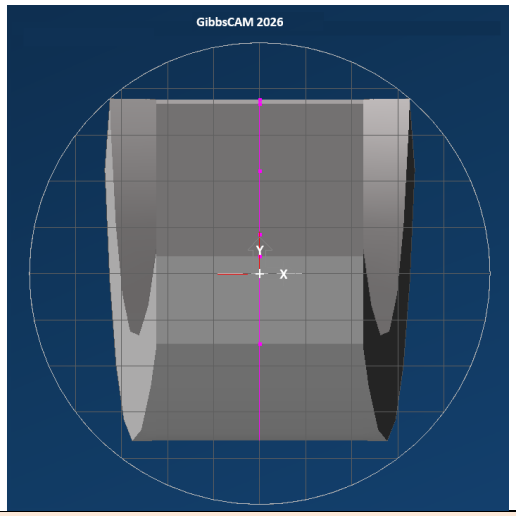
Shrink Wrap with cylindrical stock

When a part uses cylindrical stock, Shrink Wrap in GibbsCAM 2026 adjusts cylindrical stock to fit tightly around the part, instead of fitting the cylinder around the box containing the part.

Where to find it: On the **Modify** menu, under Wrap, click **Shrink Wrap** or **Shrink Wrap Visible**.





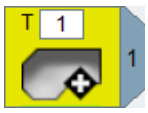


Before GibbsCAM 2026	GibbsCAM 2026

Also related to visualization and user interface

- "Variable-B Turning: Improved UI" on page 8
- "Tool Usage Reporter" on page 18
- "Tool List: Spreadsheet-like list view" on page 18

Solids



New engine powering Advanced 3D Machining

GibbsCAM 2026 introduces a completely re-engineered core for Advanced 3D Machining, now powered by the industry-leading ModuleWorks 3-axis engine, delivering improved performance, greater robustness, enhanced surface finish, and more efficient toolpaths across a wide range of 3D strategies.

The user interface remains familiar, but the engine underneath is entirely new — designed to support not just today's needs in an efficient and productive way, but the future of intelligent CAM automation.

Note: Saving toolpath from Advanced 3D Machining or 5-Axis back to older versions of GibbsCAM is not supported. If a part file with Advanced 3D Machining is opened in an older version of GibbsCAM, you can view and simulate toolpath and post its NC code, but you cannot re-generate or edit its toolpath.

While the new engine has no known issues and is fully integrated and production-ready, this is a major architectural change. **Redo** of Advanced 3D operations from previous GibbsCAM releases will often be more efficient while preserving the original machining intent, but toolpath generated by the new engine may differ slightly or significantly from previous GibbsCAM versions, even with the same parameters.

Careful testing of Advanced 3D toolpath is required. All Advanced 3D operations should be validated by simulation, visual comparison, or other means before being released to production, to ensure the newly generated toolpath meets your machining intent. Despite some inconvenience in the short term, this change represents the responsible adoption of a next-generation engine. Please treat this as a transition to a significantly more advanced solution. Check the Readme file for useful information on Advanced 3D.

History of SolidSurfacer and Advanced 3D

- First-generation SolidSurfacer (1996) included toolpath and non-toolpath components.
- Second-generation SolidSurfacer (2009) included Advanced 3D for the first time, with a then-modern third-party solution, especially suited for mold and die creation.
- Third-generation SolidSurfacer (2025) includes "**Advanced 3D 2.0**" – a full engine replacement from ModuleWorks. In GC2026 (phase 1), this new toolpath engine delivers improvements in performance, reliability, and surface finish, as well as greater toolpath efficiency. In future, the new engine provides an architecture for advanced capabilities and features.

Why re-engine Advanced 3D?

- Slow toolpath generation was inadequate for many general manufacturing companies.
- Results were very sensitive to variations in parameters, making it difficult to use well.
- Often, multiple iterations were required to obtain a toolpath of good quality.
- The previous engine went long periods without significant enhancements/fixes/improvements.

What are the goals of re-engining?

- Improve performance of toolpath generation
- Reduce or eliminate the need for multiple iterations.
- Retain existing user interface as much as possible — minimal retraining for existing users.
- Equivalent or better surface quality *and* equivalent or better runtime.
- Broader tool compatibility.
- Closer integration with 5-Axis.

Minimal changes to the user experience

The engine is a core-level improvement, and so the user interface for Advanced 3D is mostly unchanged, ensuring a familiar experience while improving productivity. You may notice the following:

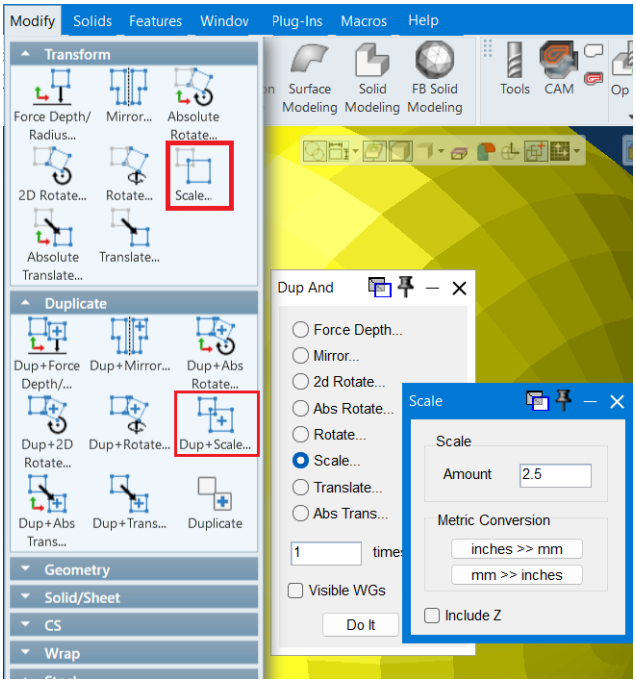
- *Improvements to reduce confusion:*
  - No more **Remove spikes**, **Optimize Z Level**, etc.: these are no longer necessary.
  - **Hit flats** behavior is more consistent.
  - Boundaries are no longer strictly required, and the Boundary Type pulldown has been trimmed to remove **Shallow Areas** and **Cutter Contact Areas**. Legacy part files that use either of these boundary types are converted, issuing a message saying the boundary style has been changed to Part Bounding Box.
  - **Rest Material** is replaced by GibbsCAM-standard "Material Only" functionality.
  - **Pocketing** and **Pocketing With Core Detection** strategies are combined.
- *Replaced functionality based on user intent:*
  - **Adaptive Stepdown** is mostly replaced by a more sophisticated **Wall Cleanup** function.
  - **Min/Max Stepover** in Pocketing is replaced by constant stepover with corner cleanup.
  - Simplified linking: Has fewer options and yields more efficient results.

For complete details, see the ***SolidSurfacer*** guide, chapter "Advanced 3D".

Modify > Scale for solids and sheets

In GibbsCAM 2026, you can scale not just geometry, but also solid bodies, facet bodies, and sheets. This makes resizing solids and sheets simple, easy, and safe within defined limits.

Where to find it: Under the **Modify** menu, Transform > **Scale** (also Duplicate > **Dup+Scale**). The change is that these functions can now be applied to elements other than 2D geometry.



Bodies and sheets can be scaled between 0.02× and 50× their original size. (Attempting to scale outside this range will trigger a warning and may result in invalid geometry.) All scaled bodies and sheets retain their original attributes. Scaled facet bodies retain the same number of facets.

This function is particularly useful for fixing import issues caused by unit mismatches (inches vs mm).

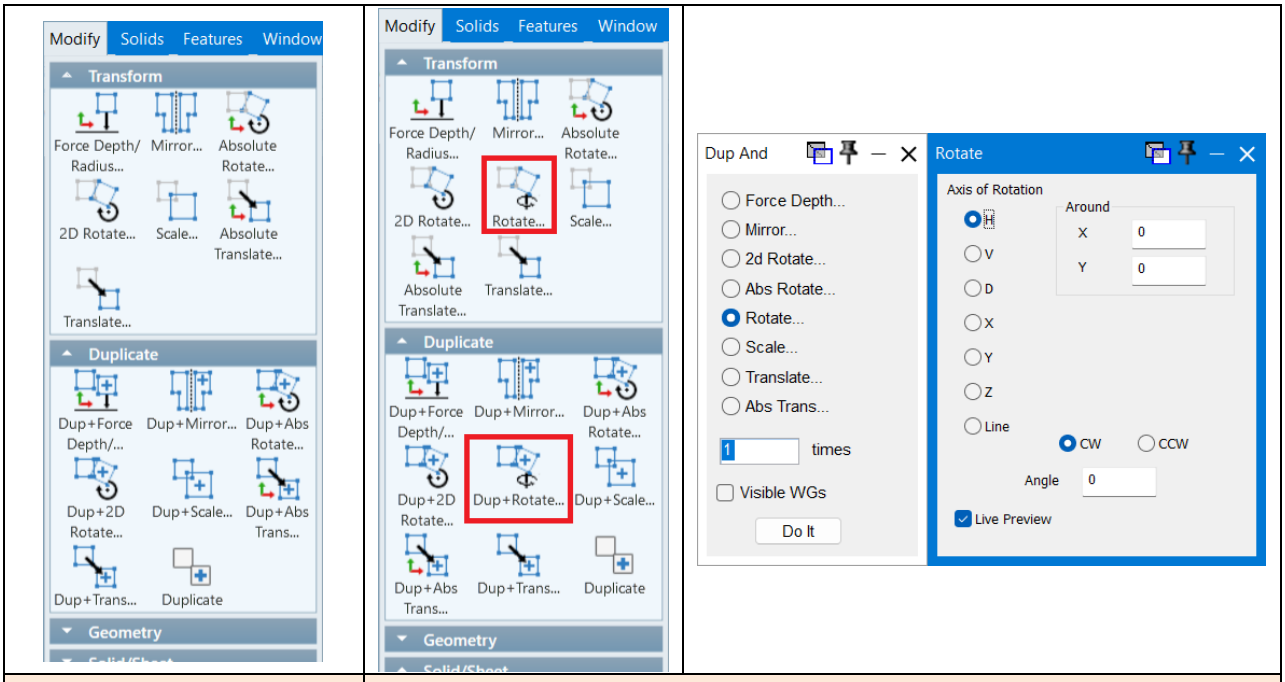
For complete details, see the ***Common Reference*** guide, chapter "Main Menu" section "Modify".

Modify > Rotate for nonplanar rotations

Where to find it: Under the **Modify** menu, Transform > **Rotate** (also Duplicate > **Dup+Rotate**).

GC2026 adds two new commands under the **Modify** menu for nonplanar (out-of-plane) rotations, reducing the need to create construction CSes. New **Rotate** and **Dup+Rotate** commands let you rotate geometry and features around **any axis**, not just within the current coordinate system plane.

This new feature makes it much easier to rotate parts, features, or fixtures that are not aligned to standard planes, something that was often difficult or time-consuming in previous releases.



Before GC2026	GibbsCAM 2026

- *Axis of Rotation:* You can choose how you want to define the rotation axis. For example, **Line** lets you select a line, hole, or straight edge from geometry or solids. Each method lets you set the center point for rotation. Point snapping / selection is supported.
- **Live Preview***:* A real-time preview of the rotation direction and axis is displayed, allowing you to make changes before applying the results. A clear visual (arrow around a bar) appears in the workspace. This helps you avoid mistakes with direction or axis choice.

For complete details, see the ***Common Reference*** guide, chapter "Main Menu" section "Modify".

Also related to Solids

- Mill > **Contour**: **Deburr/chamfer** of edges on 2.5D or solid parts ..... see page 6
- **Simulation**: Optional metallic rendering appearance ..... see page 11
- **View** > **Measure** is particularly useful for solids and sheets ..... see page 10
- **Plug-Ins > Auto CS Create** offers new options for cylinders and cones ..... see page 20

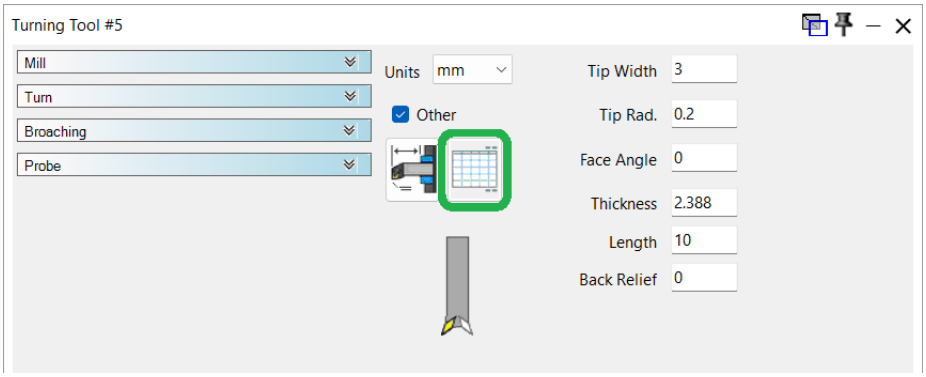
Tooling

Improvements to tools and tooling have occurred throughout GibbsCAM 2026.

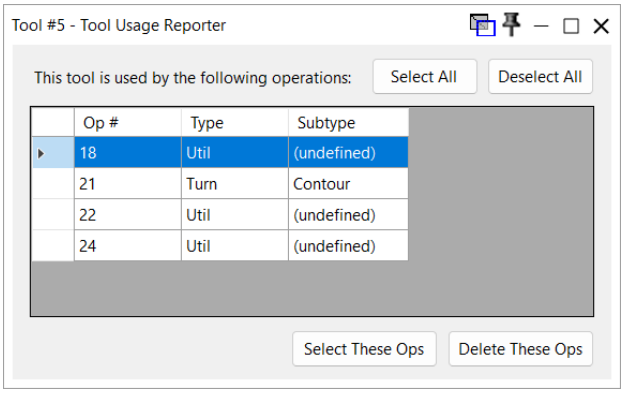
Tool Usage Reporter

A new manager-type dialog lets you see and manage tools that are currently in use, by operation.

Where to find it: Double-click a tool tile to open the **Tool** dialog. Then click to open the dialog.



Result: The **Tool Usage Reporter** dialog opens, showing all operations that use the selected tool.



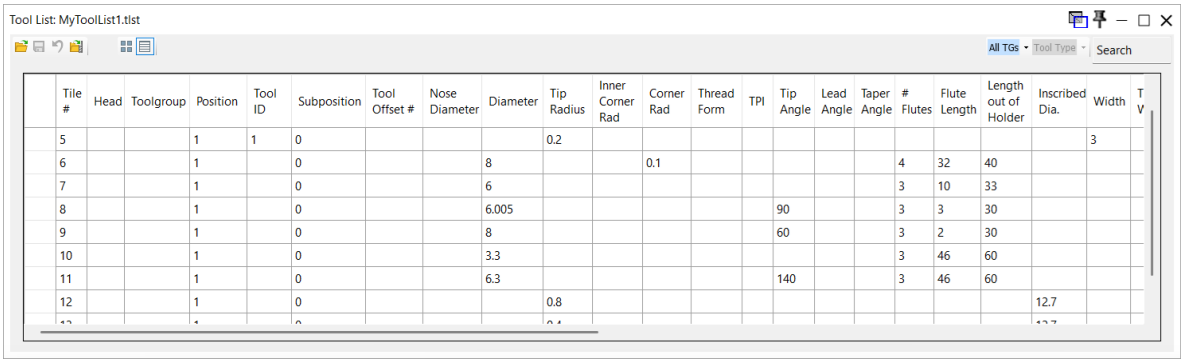
In the dialog, you can …

- Click a row to select it, or double-click a row to also select the operation and the affected workspace elements.
- Use the upper buttons in the dialog to select/deselect rows.
- Use the lower buttons to select or delete the associated operation.

Tool List: Spreadsheet-like list view

You now have the ability to display tool lists in a spreadsheet-like List View with useful column labels.

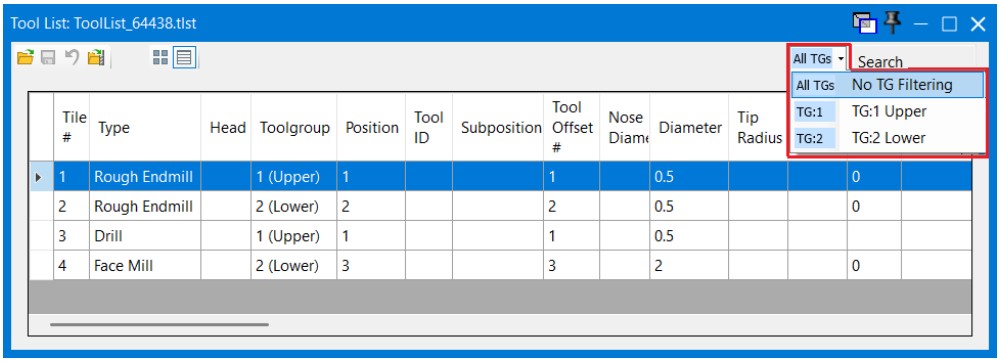
Where to find it: Right-click a tool tile and, on the context menu, choose **View/Edit Tool List**. Navigate to a tool list and open it. Then, in the **Tool List** dialog, click to display the manager-like list view.



Tool List: Filter by toolgroup

Both views of the Tool List now allow you to filter tools by toolgroups.

Where to find it: Right-click a tool tile and, on the context menu, choose **View/Edit Tool List**. Navigate to a tool list (`*.tlst` file) and open it. Then, in the upper right of **Tool List** dialog, click the pulldown menu to change from **All TGs** to the name of toolgroup whose tools you want to see.



VoluTurn support of non-round inserts

In GibbsCAM 2026, VoluTurn introduces support for non-round inserts.

Also related to Tooling

- **Plug-Ins**: **TMiQ** AI-based tool management system ..... see page 20
- **Plug-Ins** > **Tool Libraries:** Tool libraries for ToolsUnited and MachiningCloud..... see page 21

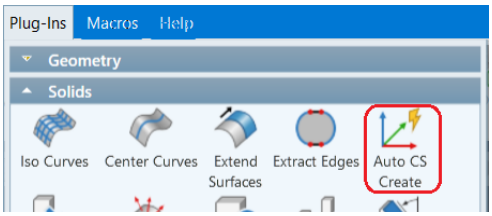
Plug-Ins

The Deburring plug-in is no longer offered, because its functionality has been folded into the Mill Contour process; see For technical details, such as running Industrial and Viewer on the same workstation, PowerOpts, and NLO considerations, see the ***Installation*** guide.

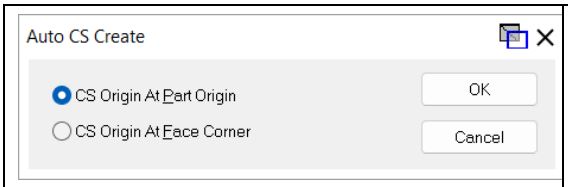
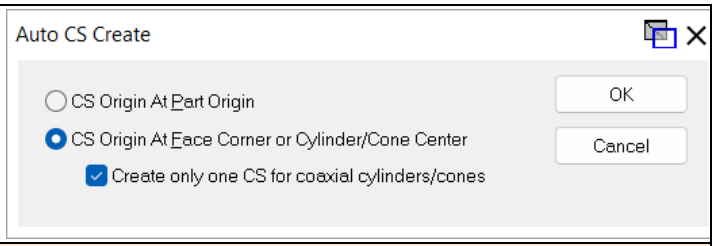
Milling on page 6 or the ***Mill*** guide, "Processes" chapter, "Contour" section.

Plug-Ins > Solids > Auto CS Create: Enhanced functionality

The **Auto CS Create** plug-in is now "modeless." It lets you select UI items and workspace elements while the dialog remains open. In addition to its previous functionality, you can now instruct the system to automatically create CSes at the tops of the axes-of-rotation of cylinders/cones, including holes. A checkbox lets you specify whether to create only one CS for cylinders/cones/holes that share the same axis.

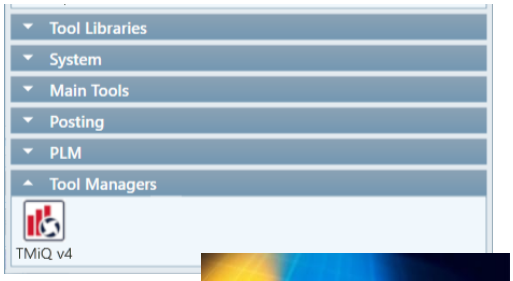


Where to find it: **Plug-Ins** >Solids> **Auto CS Create**

Before GibbsCAM 2026	GibbsCAM 2026
	

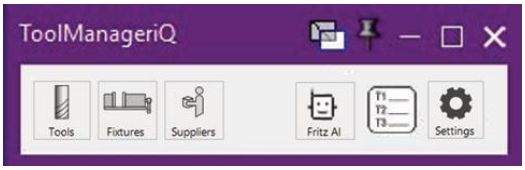
New Plug-In: TMiQ

ToolManager*iQ* (TMiQ), developed by CAM Solutions, is an AI-based tool management system for organizing cutting tools, holders, assemblies, fixtures, and suppliers, accessed via the **Plug-Ins** menu. A standalone version is also available.



- Initially, TMiQ is not installed, and **Plug-Ins** > **Tool Managers** > **TMiQ** plays an embedded video. In the right lower corner of the video is a green INSTALL NOW button that lets you install a fully functional version of ToolManager*iQ* for 30 days. When the free 30-day trial expires, you will be prompted to purchase a license through the dedicated TMiQ site.

- After ToolManageriQ has been installed as an integrated GibbsCAM plug-in, you can access its functionality via **Plug-Ins** > **Tool Managers** > **TMiQ**:



- For installers and documentation, visit this website: https://toolmanageriq.com/downloads
- For video tutorials, visit this website: https://www.youtube.com/@toolmanageriq1062/videos

Tier 1 support for TMiQ is provided by the GibbsCAM team. Advanced or in-depth support is handled directly by CAM Solutions.

New Plug-In Tool Libraries: ToolsUnited, MachiningCloud

GibbsCAM 2026 supplies two new third-party tool library importers: ToolsUnited and MachiningCloud, similar in workflow to CPTL (CoroPlus Tool Library), accessed via the **Plug-Ins** menu.

Plug-Ins > Main Tools > up2parts autoCAM

The GibbsCAM up2parts autoCAM plug-in is now directly connected to the up2parts cloud after login credentials are supplied. Instead of transferring a *.zip file, you now select the desired CAM project in the redesigned user interface and open it directly in GibbsCAM. A new up2parts autoCAM feature is support for Through Pockets, including operations for Level Roughing and Wall Finishing.

	This symbol indicates that an internet connection is required.

Miscellaneous

Microsoft desupport of Windows 10

Microsoft has officially announced that Windows 10 will reach end-of-support in October 2025. In line with this, GibbsCAM 2026 will not support Windows 10.

What does this mean for you?

Microsoft will stop providing technical support for Windows 10 and will stop updates, such as:

- Security patches to protect against viruses and malware
- Bug fixes and improvements
- Compatibility updates for new hardware and software

Windows 11 has been stable and widely adopted since its release in 2021, and we are fully aligned with it going forward. If you are on Windows 10, upgrade to Windows 11 to ensure:

- Full compatibility with GibbsCAM 2026 and later GibbsCAM versions
- Better performance and reliability
- Enhanced security and ongoing updates from Microsoft

Change to default GibbsCAM paths

In GibbsCAM 2026, unlike previous releases, default installation and data paths are now:

- *Installation:* `C:\Program Files\Gibbs\GibbsCAM\<release>_<locale>\`
- *Global data (**All Users**):* `C:\ProgramData\Gibbs\GibbsCAM\<release>\`
- *User data:* `C:\Users\<username>\AppData\Roaming\Gibbs\GibbsCAM\<release>\`

The transition is seamless for most users, because the change is fully supported by the GibbsCAM Migration Tool. However, sites that do not use the Migration Tool and people who manually access folders for copying or referencing files should know to change (from ~\CAMBRIO\GibbsCAM\~) to ~\Gibbs\GibbsCAM\~.

System requirements

OS: ..... Windows 11 or Windows Server 2022.

CPU: ..... Intel Core i9, i7, or i5 with four or more cores; or
AMD Ryzen or Threadripper

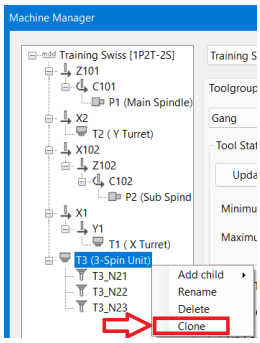
RAM: ..... 16+ GB

Video card:..... NVIDIA video card with 4+ GB of video memory

Third-party library support

- ModuleWorks 2024.08, 2024.12, and 2025.04
- VoluMill 10.5; VoluTurn 10
- Parasolid 37, ACIS 2025.1, SolidWorks 2025, Solid Edge 2025, Catia v5 R2025, Inventor 2026, etc.; see the ***Data Exchange*** guide for the complete list

Machine Manager: Ability to clone toolgroups



Machine Manager now allows you to clone a toolgroup ( ) node with all child nodes it may contain.

Where to find it: In Machine Manager, right-click a toolgroup node and, on the context menu, choose **Clone**.

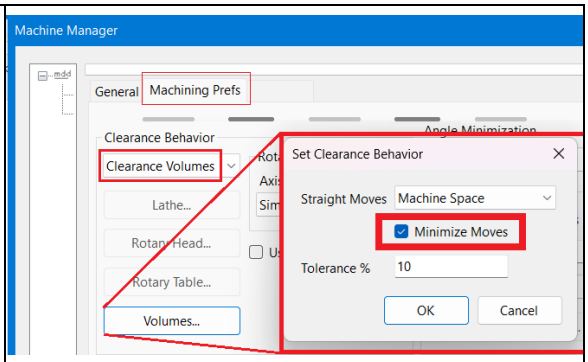
Machine Manager: Clearance Volumes > Minimize Moves

To minimize tool retracts, Clearance Volumes can be defined in the MDD — see illustration — causing the **Clearance (Δ)** field to appear in the DCD's **Workspace** tab.

In GibbsCAM 2026, Machine Manager's **Set Clearance Behavior** dialog now provides a **Minimize Moves** checkbox that further increases efficiency between operations where the tool does not rotate to a new orientation, and where the tool is aligned with a linear axis.

For same-tool interops that stay on the part, when **Minimize Moves** is in effect, the tool retracts to the highest position needed to prevent its traversal from intersecting the clearance volume as the tool rotates and moves above the start point of the next operation.

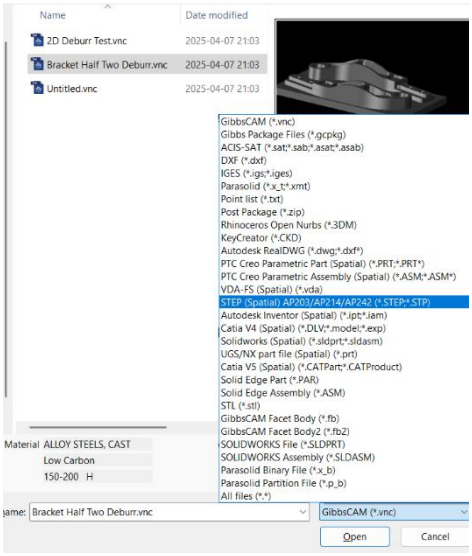
Where to find it: In Machine Manager, **Machining Prefs** tab, under **Clearance Behavior**: In the pulldown, choose **Clearance Volumes**. Then, click the **Volumes** button to open the **Set Clearance Behavior** dialog, which now offers a **Minimize Moves** checkbox.



AP242 in pulldown menus for File>Import and File>Open

The **File>Import** and **File>Open** dialogs now list AP242 along with other STEP (Spatial) `*.stp/*.step` data types.

Where to find it: In the **File>Open** or **File>Import** dialogs, click the pulldown menu in the lower right (just above the **Open** button).



5-Axis

Doc Note: As of the start of Open Beta, this chapter of *What's New* is still under construction. It might contain errors, duplications, or omissions.

This chapter discusses the following enhancements to 5-Axis in GibbsCAM 2026.

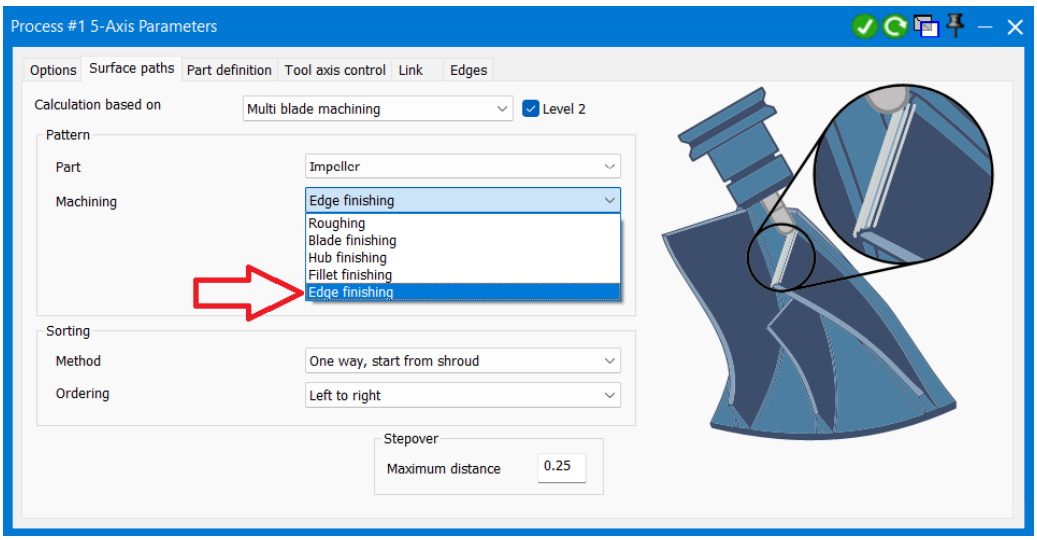
Calculation based on Multi-blade Machining

~\WN_MW2025.04\sources\1543735070.html Toolpath: Special Parts > Multiblade

Multiblade Edge Finishing

Where to find it: (Multi-blade machining) **Surface paths** tab > **Machining** > **Edge Finishing**

This new edge finishing strategy guides the tool longitudinally along the edge to remove any excess material left from the previous operation without causing damage to the thin leading edge. This allows the part to be finished with the required surface quality and material integrity. It ensures that the thin leading edge is not damaged, maintaining the integrity and quality of the finished part.



~\WN_MW2024.12\sources\1477050564.html Toolpath: Special Parts > Multiblade

Feedrate for Links in Multiblade

Where to find it: (Multi-blade machining) **Link** tab > **Link Feedrate (%)**

Previously, it was difficult to optimize the non-cutting time because it was not possible to adjust the feedrate on the links. But it is often desirable during a non-cutting move to increase the feedrate to reduce machining time. With this release, you now have the ability to customize the feedrate for links. This gives you greater control over the machining process while reducing the overall cycle time.

Feedrate Override for Leads in Multi-blade

This feature allows lets you set the feedrate override for leads.

Where to find it: (Multi-blade machining) **Link** tab > **Lead-in Feedrate(%)** / **Lead-out Feedrate(%)**

In previous releases, you were limited to using feedrate overrides only for links, extensions, and the actual toolpath. Now, there is an option to set the feedrate override for lead-in and lead-out moves as a percentage of the actual toolpath feedrate. This feature ensures the tool enters and exits the workpiece smoothly, resulting in a better surface finish and improved tool life.

~\WN_MW2024.08\sources\1385529588.html Toolpath: Special Parts > Multiblade

Expanded tool support for Blade Finishing

Where to find it: **Surface paths** tab > **Machining**/blade finishing > **Leading/Trailing edge** > **Feedrate(%)**

In previous releases, Multiblade Finishing required ball end mills for machining impellers and blisks. This prevented the use of standard tools such as flat end mills and bullnose mills.

Now, you can use bullnose mills, end mills, and all types of taper tools for blade finishing. This provides more machining flexibility to use standard tools such as bullnose mills for multiblade finishing.

Control Feedrate on Edge Extension

You can now control the feedrate for leading and trailing edge extensions.

Where to find it: (Multi-blade machining) **Tool** tab > **Tool type**

When applying extensions to the leading or trailing edge, the aim is to leave offset material to finish later. Therefore, since the extension is not going to actively cut the material, it makes sense to increase the feedrate to save machining time. The **Feedrate(%)** parameter, which previously modified only the feed when the tool touches the blade, has been expanded and now also applies to the extensions. It is now possible to use a faster feedrate for leading and trailing edge extensions, reducing the overall machining cycle time.

Calculation based on SWARF Machining

~\WN_MW2024.12\sources\1481670953.html Toolpath: Multi-Axis Surface Finishing > SWARF Machining

The 4-axis SWARF mode has been enhanced to achieve optimal geometric accuracy. The surfaces of some parts, such as cylindrical cams, can be geometrically machined using flank milling on a relatively inexpensive 4-axis machine. However, when machining with a tool smaller than the slot, it is difficult to optimally align the tool with the surfaces.

This enhancement improves the accuracy of parts, reduces machining costs, and offers greater flexibility in selecting tool diameters.

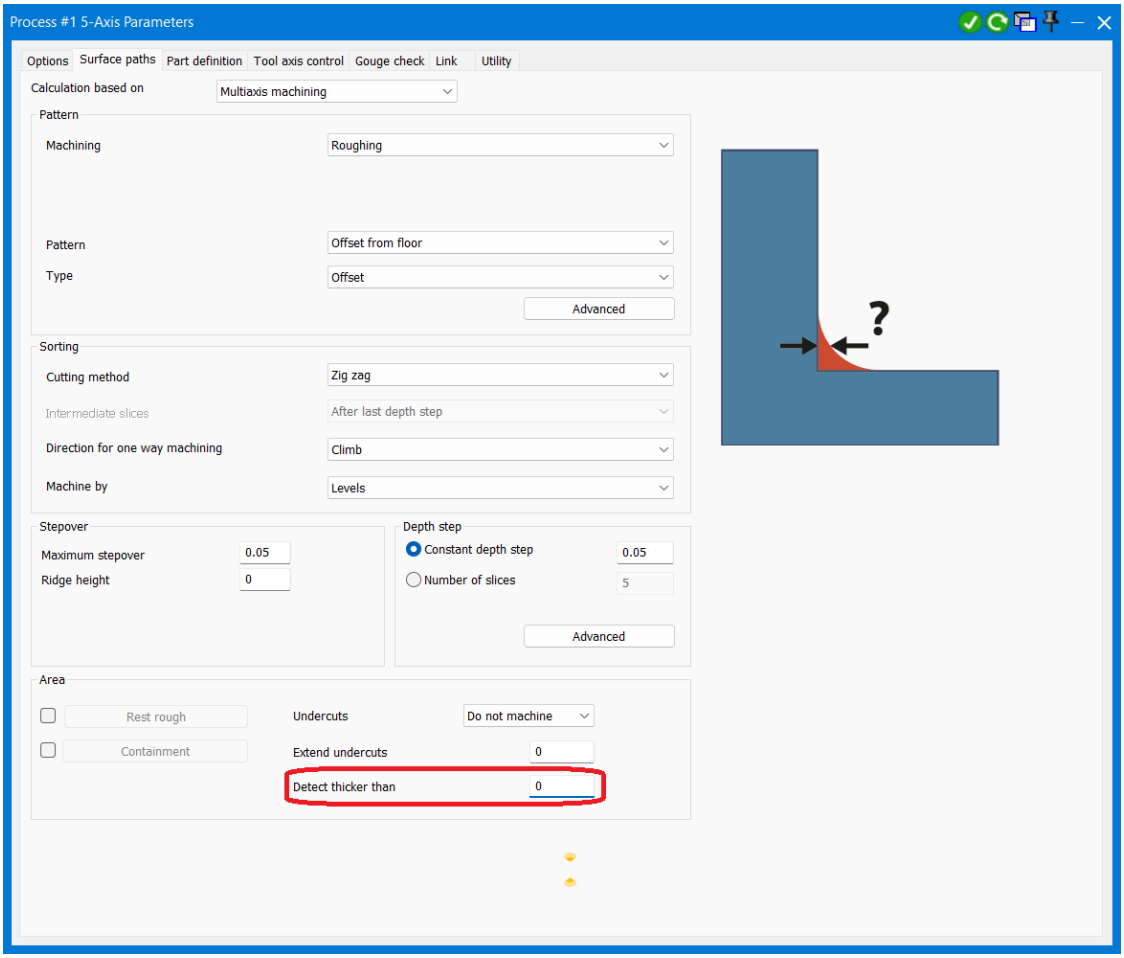
Calculation based on Multiaxis Machining

~\WN_MW2024.08\sources\1394770483.html Toolpath: Area Roughing

Detect Material Thicker Than

Where to find it: (Multiaxis machining) **Surface Paths** page > **Area** > **Detect thicker than**

With this feature, thickness of rest material gets detected considering the preset value. This improves the quality of stock-based roughing operations, as the tool does not go into the unnecessary areas.



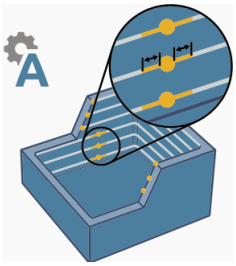
~\WN_MW2024.12\sources\1447952990.html Toolpath: Multi-Axis Surface Finishing > Wall/Floor/Rest Finishing

Advanced Barrel Tool Support for Multi-Axis Surface Finishing

Advanced barrel mills allow you to precisely define the geometry of barrel tools.

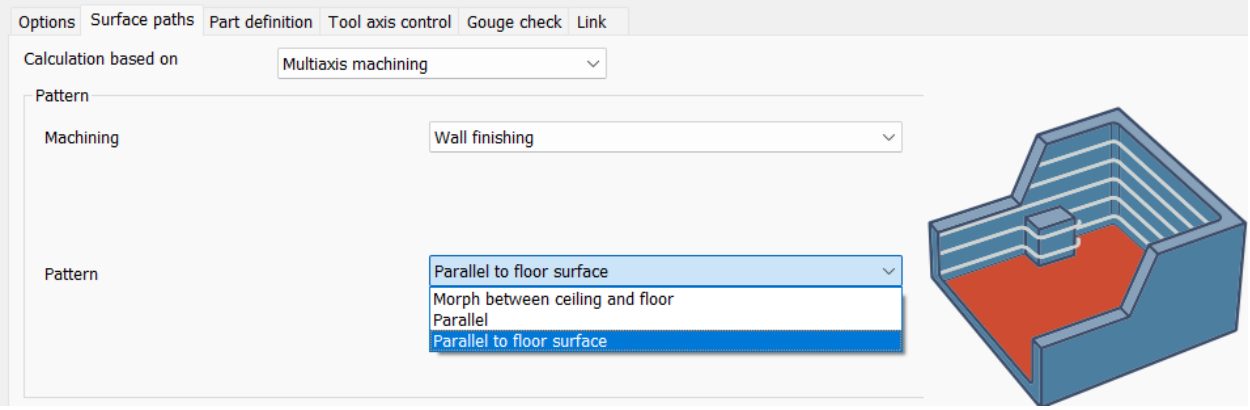
~\WN_MW2024.08\sources\1398341663.html Toolpath: Multi-Axis Surface Finishing

For **Machining** choices **Wall Finishing**, **Floor Finishing**, and **Rest Finishing**, under **Area**: New controls for **Extend Toolpaths**; and (for **Wall Finishing**) new **Parallel to floor** pattern.



Toolpath Extensions: This feature allows the tool to extend its motion beyond the start and end point of each closed contour. This reduces the size of the cusps at the start and end points, particularly with advanced tools such as barrel tools, and produces clean surfaces.

Parallel to floor pattern: The new **Parallel to floor surface** cut pattern for wall finishing generates slices parallel to a planar floor to ensure that features on the wall do not disrupt the pattern generation. This introduces a whole new set of geometries that can be machined with less tilting and better surface quality.



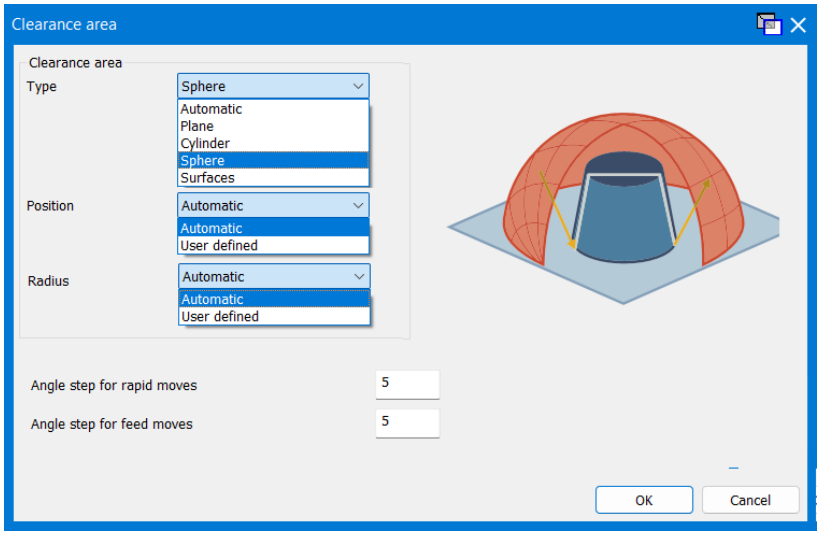
~\WN_MW2024.12\sources\1477051587.html Toolpath: Multi-Axis Roughing > Area Roughing

Primitives Definition as Clearance Area

Where to find it: **Link** tab > **Clearance area** > **Clearance area** dialog

You can now define the clearance area as **Plane**, **Sphere**, **Cylinder**, or **Surface** to better handle retracts.

- **Plane**: The planar area is raised above the workpiece at the specified height and in the desired direction. The direction is determined by the direction of the plane normal vector.
- **Cylinder**: The cylindrical area around the workpiece is specified by the radius and desired direction. This can be useful for 4-axis and 5-axis machining operations.
- **Sphere**: The spherical area around the workpiece with a specified radius and desired position. The position can be defined as XYZ coordinates or by an imported point.



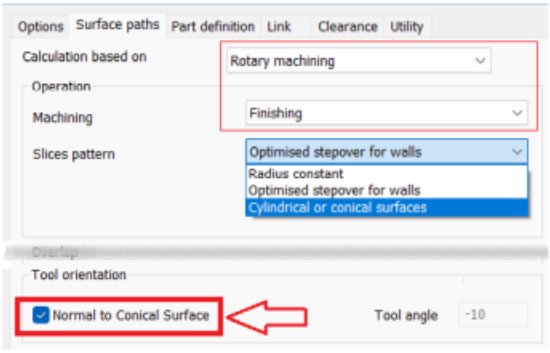
Calculation based on Rotary Machining

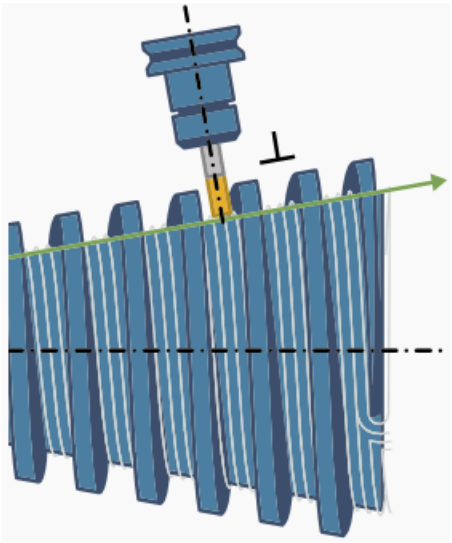
~\WN_MW2025.04\sources\1560740134.html Toolpath: Multi-Axis Surface Finishing > Rotary Machining Finishing

Cone Angle Normal to Conical Surface

Where to find it: (Rotary Machining > Finishing) **Surface paths** tab, under **Tool orientation**

With this enhancement, it is now possible to define the angle for floor detection while keeping the milling head stationary or tilting it to a completely different angle. This enables more versatile and efficient machining of angled floors. The system detects and applies toolpath on conical floors without tilting the milling head, providing greater flexibility and precision during rotary finishing.





~\WN_MW2024.12\sources\1447953226.html Toolpath: Multi-Axis Surface Finishing > Rotary Machining Finishing

Support for Fixtures/Chucks for Rotary Machining Finishing

Where to find it: Part definition tab > Machining surfaces > Fixture

Fixtures and chucks can now be selected as a mesh/model. The mesh/model is then spun and the resulting profile is used to avoid collisions.

With this enhancement, you can select Fixtures/chucks that will now be included in the collision avoidance calculations for Rotary Roughing toolpaths.

You can now select fixtures/chucks for inclusion in the collision avoidance calculations for rotary finishing toolpaths. The selected mesh/model is spun and the resulting profile is used to avoid collisions. You can specify an offset distance to provide more control and safety, resulting in more confidence during toolpath generation. This also simplifies the implementation for CAM partners.

Leads for Floor Finishing

Where to find it: Part definition > Link tab > Contour Leads

You can now define lead-in/lead-out for the finishing pattern for rotary machining of cylindrical or conical surfaces. This ensures smooth entry into the material and fewer tool marks, improving the surface finish in some cases.

~\WN_MW2024.12\sources\1447953162.html Toolpath: Multi-Axis Roughing > Rotary Machining Roughing

Support for Fixtures/Chucks for Rotary Machining Roughing

Where to find it: Part definition tab > Machining surfaces > Fixture

You can select fixtures and chucks to include them in collision-avoidance calculations for rotary roughing toolpaths. (The selection is treated as a mesh/model that is spun, and the resulting profile is used to avoid collisions.) An offset distance is also provided to give users more control and safety.

Support for fixtures/chucks provides more safety and confidence to the user during toolpath generation. This also simplifies implementation for CAM partners.

~\WN_MW2024.08\sources\1381171832.html Toolpath: Rotary Machining Finishing

Containment Trim for Rotary Finishing

Where to find it: **Part definition** tab > Containment > Trim

This feature trims the toolpath as the tool center touches the containment region. This helps to avoid unnecessary extensions of toolpath beyond the containment.

The containment is designed to have a toolpath up to its boundaries to ensure that material is removed. However, when finishing, you can add the boundary without adding a toolpath. This lets you tailor the toolpath and benefit from the containment function.

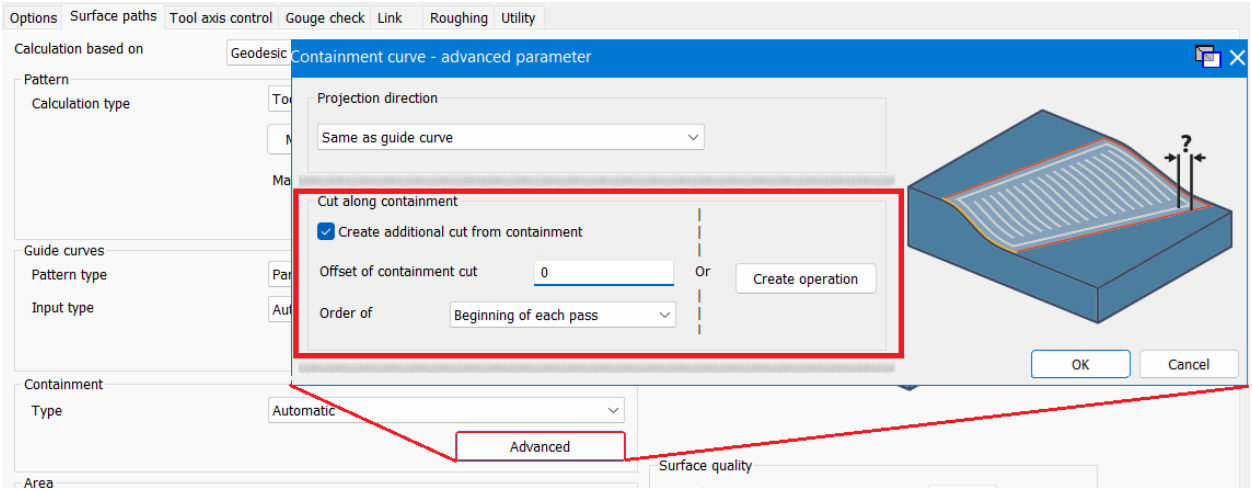
Calculation based on Geodesic Machining

~\WN_MW2024.12\sources\1480196382.html Toolpath: Multi-Axis Surface Finishing > Geodesic Machining

Multiple Boundary Passes

Where to find it: (Geodesic) **Surface paths** tab > Containment > **Advanced** > **Cut along containment**

This feature lets you set up a machining operation for the boundary of an area. A new operation is created that allows full customization of the boundary pass. For example, a different stepover or tilting behavior can be used for the boundary pass. This results in improved surface quality and safer operations.



~\WN_MW2024.08\sources\1356988591.html Toolpath: Multi-Axis Surface Finishing > Geodesic Machining

Robustness against Input Deviation

With this enhancement, small changes to the mesh (new triangulation, different mesh offset or different mesh patching) now have a smaller effect on the toolpath.

Calculation based on Deburring

~\WN_MW2025.04\sources\1556545976.html Portfolio=Toolpath > Multi-Axis Edge Finishing > Deburring

Automatic Machining Direction

Where to find it: (Deburring) **Surface paths** tab > Path parameter > Direction

In previous releases, determining the appropriate machining direction for a 3- and 4-axis toolpath often led to suboptimal ordering. Now, you can use **Automatic Machining Direction** to determine the machining direction automatically based on the contour definition. This makes toolpath deburring more user-friendly, while also reducing the machining cycle time.

User-Defined Edge Sequence

Where to find it: (Deburring) **Surface paths** tab > Geometry input

With automatic sorting, it is difficult to meet the deburring requirements of each part, because the predefined order may not align with specific needs.

This feature provides an alternative to automatic sorting: The input order of the included curves can be preserved and the resulting toolpath is sorted accordingly, but you can select edges for machining in whatever sequence you prefer, providing greater flexibility during the deburring process and improving overall efficiency and precision.

~\WN_MW2024.12\sources\1396081627.html Toolpath: Multi-Axis Edge Finishing > Deburring

Consolidated 3+2 Behavior

The common direction algorithm has been enhanced for deburring to ensure a 3+2 orientation with minimal 5-axis movements. This minimizes undesired 5-axis motions, resulting in safer and more accurate machining.

Tool axis control tab: Tilting

~\WN_MW2024.08\sources\1409351785.html Toolpath: Tilting Core

Autotilt Support for Bullnose Tools

This enhancement enables the autotilt option to be used with bullnose tools.

Tilted into plane

A new tilting strategy,, **Tilted into plane**, lets you tilt the tool axis in the plane defined by the normal to a direction you select. The tool axis is projected on the selected plane on each toolpath point.

If the plane normal is aligned with a rotational axis of the machine, it limits rotational machine movements to that axis.

Tilt Tool in a User Defined Plane

A new tilt option, **Tilt tool in a user defined plane**, has been introduced to eliminate one Degree of Freedom (DOF). This feature lets you restrict the tool axis in a plane that you define. Depending on the orientation of this plane, either rotary or tilt can be avoided completely. If the plane normal is aligned with a rotational axis of the machine, it will limit rotational machine movements to that axis.

Benefits: You can expect fewer machine axis movements for more stable machining.

Automatic Tilting Kernel for Existing Toolpath

The new automatic tilting kernel for existing toolpaths provides more consistent results and acts as an enabler for using the latest features.

Link Tab: Linking

~\WN_MW2024.12\sources\1447952990.html Toolpath: Linking

Tool Axis Clearance Control

Instead of retracting normal to the clearance, this release now allows the option to retract or approach along the tool axis until it connects with the clearance.

Automatic Clearance Improvement

This enhancement optimizes links for cylindrical or spherical surfaces, reducing machining time while maintaining necessary clearance during linking motion. It applies specifically when the clearance area type is set to automatic, or when the user selects a cylinder/sphere clearance area type with position and radius options set to automatic.

User Defined Clearance Areas

Where to find it: Link tab > Retracts dialog

This new feature lets you select user-defined surfaces as clearance areas. With this feature, toolpaths can be limited to shorter retracts, thereby reducing the overall cycle time while providing more flexibility and control to the user.

Virtual Fixture Plane for Safe Linking

Where to find it: Link tab > Retracts dialog

With this feature, you can avoid links that go beyond the defined virtual fixture plane. This prevents the tool from entering dangerous areas: The fixture plane is defined according to a user-specified height so that links are forced to stay above that plane. Defining the fixture plane reduces the risk of collisions, even on complex machining setups.

Note: The **Virtual Fixture Plane** option is available for Sphere Clearance only.