GIBBSCAM 2025 CAM for Production Machining

Version 2025 : September 2024

What's New in GibbsCAM 2025

GIBBSCAM

Contents

Highlights

Some of the most important enhancements in GibbsCAM 2025 are as follows:

- **OptiThreading** — This Sandvik Coromant technology is tuned for cutting complex thread profiles or in high-strength alloys or exotic materials. It enables controlled chip breaking, no vibration marks, and higher surface finish in thread turning applications, and it can extend tool life. For details, see "OptiThreading".

- ▦**GibbsCAM Copilot** — An AI-powered chat tool to help users solve problems interactively. For details, see "GibbsCAM Copilot: AI-powered chat tool".

- **Op Tile List** — For MTM models, new Op tile list buttons ⦚ ⊞ ⊟ let you set the mode for displaying flows. For details, see "New modes for Op Tile List: Columns for flows".

This "Highlights" section is just a small subset of the enhancements in GibbsCAM 2025. The next section (Overview of What's New) provides a complete list.

Overview of What's New

Enhancements to **Milling** include:

- Mill > Holes > Mill Feature: With Top Surface Z choice Automatic, you can now specify an incremental offset that will be applied to all selected features.

- Mill > Roughing: For **Pocket** toolpath, you can now optionally machine pockets in the order they were selected.

- Mill > Lead-In / Lead-Out: New options for lead-in and lead-out let you combat tool deflection by cutting past the edge of the material.

- A new plug-in, up2parts autoCAM, lets you import up2parts bundles that contain proposals and recommendations for machining steps, operations, and tools. Importing the bundle generates processes, operations, and toolpath that you then refine using GibbsCAM.

For details, see "Milling".

Enhancements to **Turning** include:

- **OptiThreading**: This Sandvik Coromant technology enables controlled chip breaking, no vibration marks, and higher surface finish in thread turning applications.

- **Roughing**: A new choice for Rough Type, Rib Cut, reduces tool deflection and improves tool life.

- Swiss-style machines now support sliding headstocks and Z axis limits.

For details, see [“Turning”](#).

GibbsCAM 2025 has many enhancements to **Visualization and User Interface**:

- **GibbsCAM Copilot**: An AI-powered chat tool helps you solve problems interactively.

- Op Sim and Machine Sim now provide graphics to indicate spindle direction for both tool and part.

- A new **Edit** > Selection menu option, **Filter Selections**, can help you to refine your geometry selections, opening a dialog that lets you include/exclude various kinds of points, lines, circles, curves, text, PMI, etc., for overlap mode or mouse drag.

- In Machine Sim, **Collision Checking** and **Program Error Check** are now enabled by default for newly created parts.

- In Op Sim, **Analyze Cut Part** now lets you supply a value for Tolerance.

- **Op Tile List** — For MTM models, new Op tile list buttons let you set the mode for displaying flows. You can redraw toolpath in one easy action: Simply Ctrl+click or Shift+click an operation tile. In Op Sim and Machine Sim, you can set a new Stop type: Stop Before Every Operation. When Simulation is running, the **Sync Control** dialog provides a graphical indicator (a moving red bar) that shows the progress of the simulation as it executes.

For details, see [“Visualization and User Interface”](#).

Improvements to **Tooling**:

- Additional tool cutting parameters like Ae (Cut Depth) and Ap (Cut Width) can be stored with tool cut data.

- A link in the **Tool** dialog, Order tool, appears when the current tool is known to be available from a supported tool library manufacturer, so that the tool can be ordered immediately.

For details, see [“Tooling”](#). Also see [“OptiThreading”](#).

Improvements to **Posting and Output**:

- TMS now supports full WFO system functionality.

- Postprocessors can now query and output programmed machining tolerance for operations on solids.

For details, see "Posting and Output".

Enhancements to **Solids** include:

- Ability to edit facet bodies: Boolean, offset/shell, History, heal, refacet, tessellate, and more.

- Collision avoidance of workspace fixtures and Intermediate Fixtures.

For details, see "Solids".

Miscellaneous improvements and changes:

- Copying/pasting of solids now preserves colors and other attributes.

- NLO now allows License borrowing with checkout/return capabilities.

- Op Manager and Process Manager support VoluMill data fields.

- Op Manager and Process Manager support many 5-Axis data fields.

- MDDs can now set maximum spindle RPM for parts and tools.

- VoluMill 9.6 is now supported

For details, see "Miscellaneous".

Improvements to **5-Axis**:

- Integration of ModuleWorks 2024.04, 2023.12, and 2023.08.

For details, see "5-Axis".

Milling

GibbsCAM 2025 includes several important Milling enhancements.

Mill Feature > Top Surface Z=Auto: Auto+ value for incremental offset

Where to find it: Mill > ... > Mill Feature tab; for Top Surface Z, choose Automatic: **Auto+** textbox.

In Milling processes that have Mill Feature tab, when Top Surface Z is Automatic, you can enter a value in the Auto+ textbox that provides a global incremental value above the detected Top Surface Z.
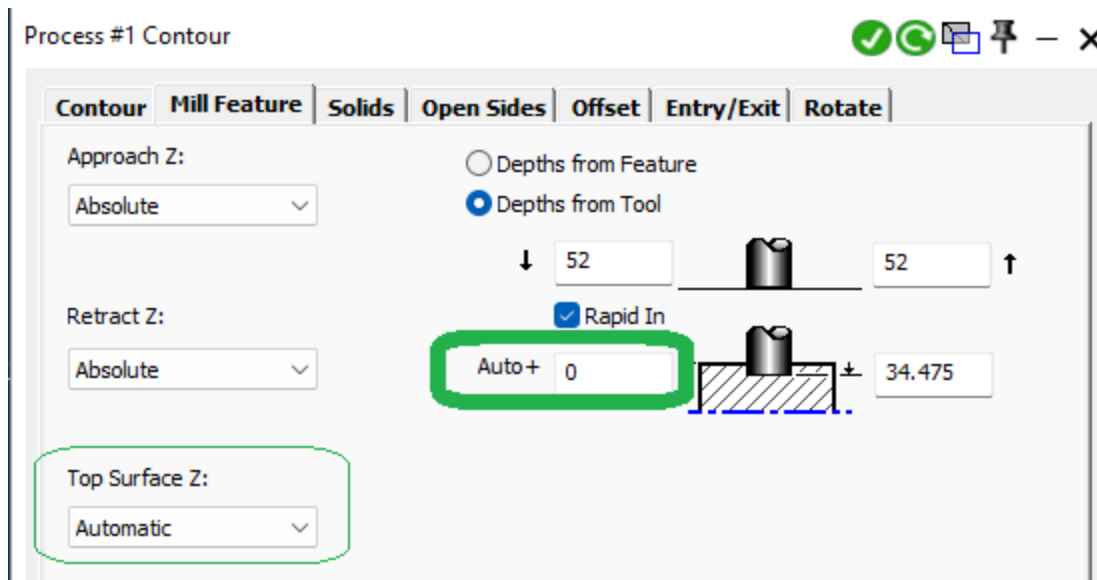
This replaces the previous interface, where the value in the Automatic textbox was automatically 0.

Pocket / Contour: Ability to machine pockets in selection order

Where to find it: Mill Roughing/Contouring, left tab: Mach. Order pulldown option From Selection.

In Milling processes Roughing (**Pocket** tab) and Contour (**Contour** tab), you can now instruct the system to create toolpath in the order of the user's selections. The previous option (Automatic) instructs GibbsCAM to determine the order.
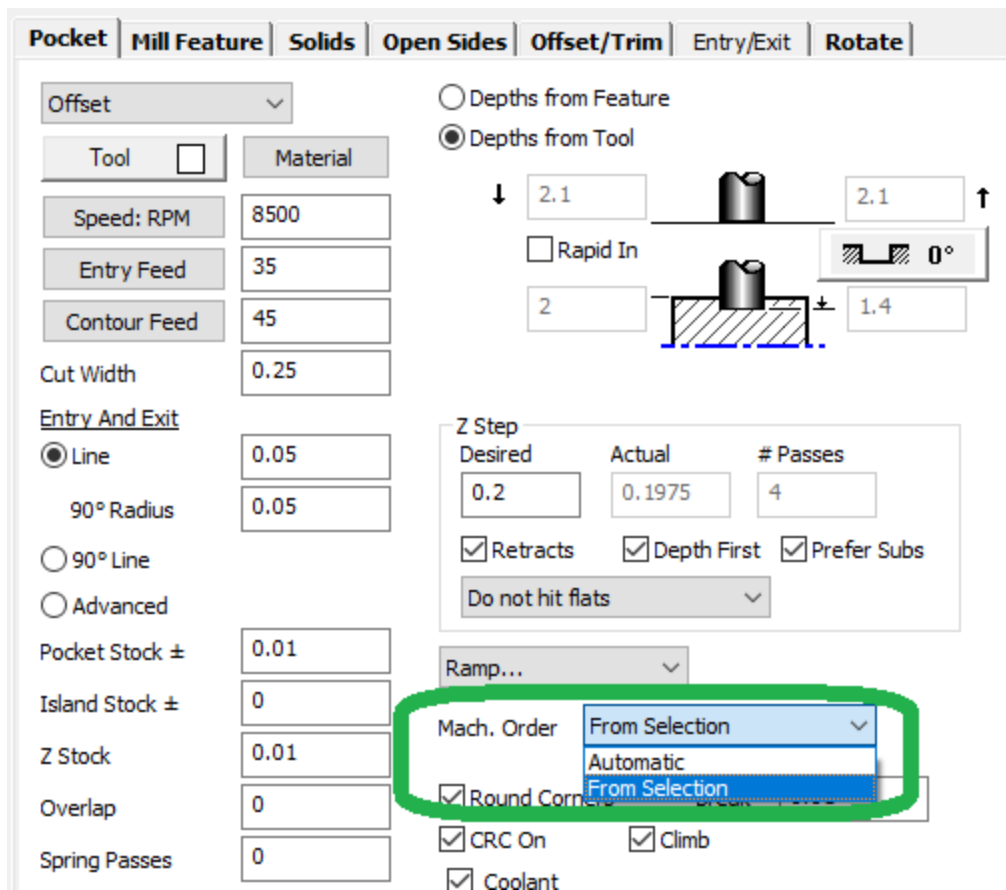
Lead In / Lead Out: Open Sides > Overlap Finish Walls

Where to find it: In Mill process dialogs, with Material Only: Open Sides textbox Overlap Finish Walls.

In all Milling processes, a new parameter when Material Only is enabled lets you cut past the edge of the material, recutting a short section of the finish wall. (The value 0, meaning "no overlap", is the previous and default behavior). You can use this to combat tool deflection and increase tool life.

Plug-In: up2parts AutoCAM

The up2parts AutoCAM plug-in lets you import up2parts *.zip files and use the results to generate GibbsCAM operations, toolpath, simulation, and G-Code.

Workflow: On the up2parts Website

The following steps can be done by anyone familiar with up2parts, not necessarily a GibbsCAM user.

1. The user supplies credentials to the up2parts website and then uses up2parts to:

    a. Specify a solid model in STEP format. Note that AP242 includes PMI and tolerance information, but AP203 and AP214 do not.

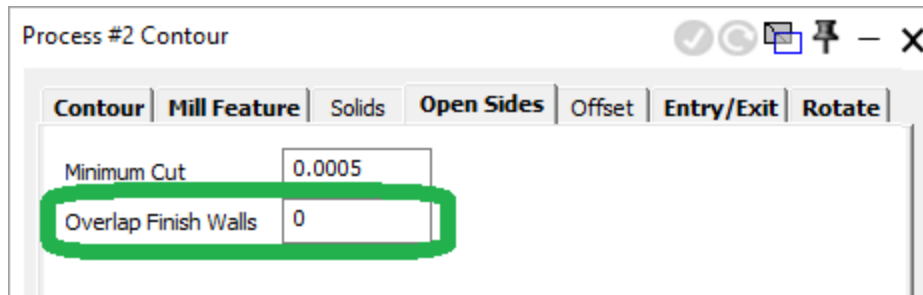   b. Start the up2parts geometry analysis and AI-based feature recognition.

    c. Get a technical description that includes recognized features like pockets, holes, countersinks, and surfaces.

4. Then, up2parts calculation is used to generate a work plan by defining the machine, material specification, and general tolerance (ISO-2768).

5. Next, up2parts proposes a CAM project for six-sided machining that includes "clamping" (approach from top/bottom/left/right/front/back), tools, machining steps, and operations. The up2parts interface allows adjustments and refinements to this proposed project.

6. After the project is confirmed, up2parts bundles its proposals and recommendations into a *.zip file readable by GibbsCAM. This .zip file includes: the solid model in STEP format with complete feature tree; stock information; tooling; features with

operations; and meta information for clamping. Associated PMI (product manufacturing information), if any, is preserved.

Workflow: Within GibbsCAM

Before you begin: Set the units in your DCD to match the units of the up2parts bundle (usually mm).

1. On the **Plug-Ins** menu, under **Main Tools**, click up2parts autoCAM.

2. In the **up2parts program arguments** dialog:
   (for Selected file): Supply the *.zip filename of the bundle prepared by up2toparts.
   (for Select clamping): Select a folder to specify approach: top/bottom/left/right/front/back.
   Click Import.

3. Notice that progress is shown by status bar at the bottom of the GibbsCAM window. When the import process is complete, GibbsCAM displays the model in the workspace. Then, watch as features are processed and operations are generated. Finally, in response to the message "The part is loaded and processes/operations have been created", click OK.

4. You can inspect and optionally modify any aspect of the GibbsCAM model, including tooling, processes, parameter values, and operations. You can then use any of the native GibbsCAM Simulation options to visualize the machining operations and to find/correct any toolpath issues.

5. Use the appropriate GibbsCAM postprocessor to generate G-Code for your specific machine.

Turning

OptiThreading

GibbsCAM Turning is significantly enhanced in Version 2025 by the addition of OptiThreading. This Sandvik Coromant technology enables controlled chip breaking, no vibration marks, and higher surface finish in thread turning applications.

Where to find it: The **Threading** process dialog, under Definition, now offers a third choice (below From Parameters and From Shape): **From Cutting Data**. This is active when the process uses a Threading tool that has been imported from CPTL (CoroPlus Tool Library) with recommended cutting data. If you do not yet have such a tool, see Using the GibbsCAM CPTL Plug-In below.

Process #1 Thread
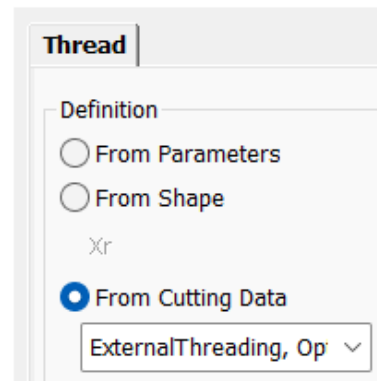
If the tool has OptiThreading data, From Cutting Data turns off some of the default interface and lets you specify OptiThreading-specific parameters like Oscillation Frequency (Coarse/Normal/Fine), a value for Oscillation Extra Lift Distance, and a checkbox for Extra Oscillation Final Rough Pass.
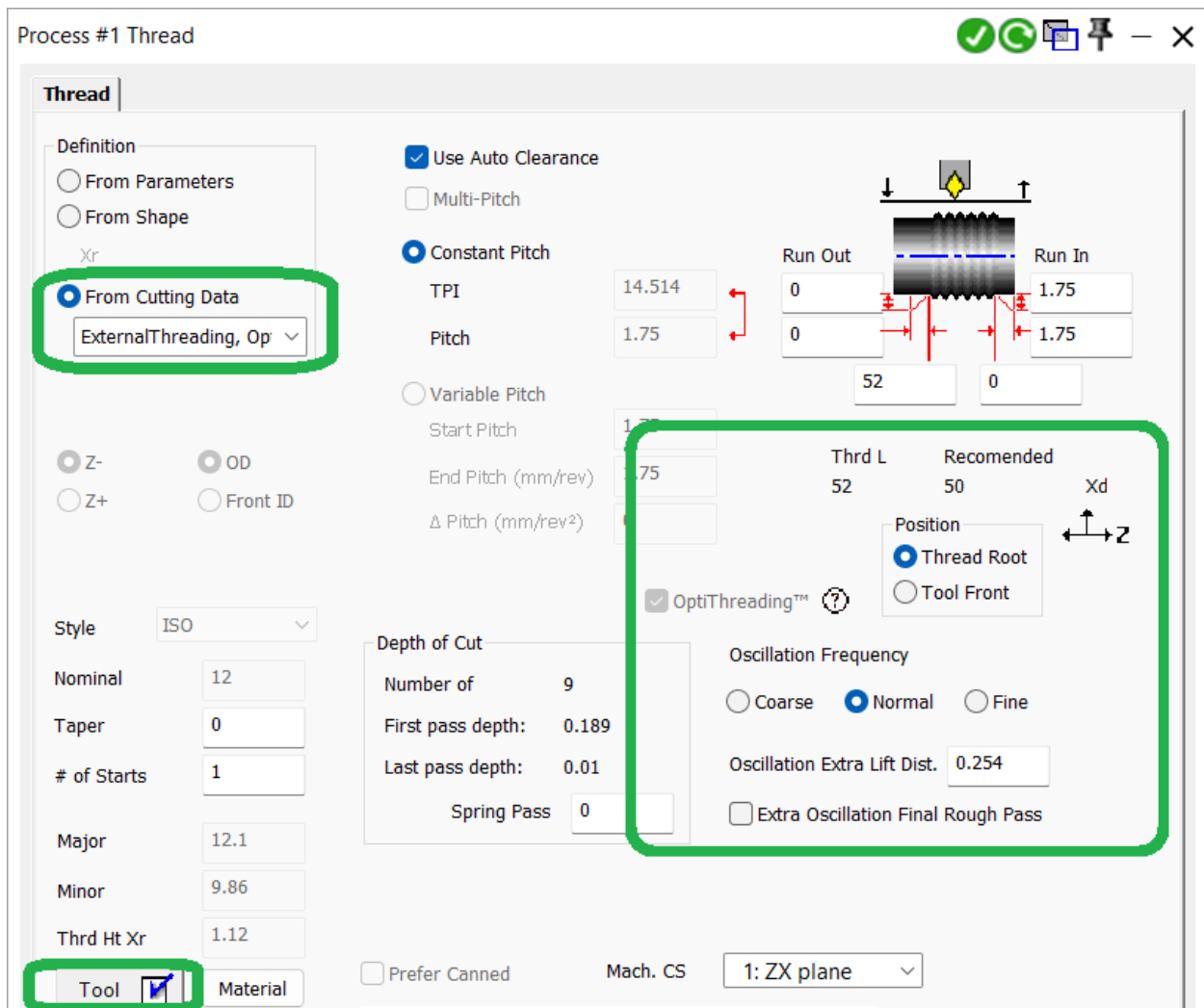
Using the GibbsCAM CPTL Plug-In

As in GibbsCAM 2024, you import tools from the CoroPlus Tool Library (CPTL) by starting the CPTL plug-in and using the CoroPlus Tool Library Importer dialog.

Please Note: To use the GibbsCAM **CoroPlus Tool Library Importer**, you must have a valid and active CPTL account and tool data (tools, holders, cutting data, etc.)

Starting CPTL Using the Plug-Ins **Menu**

Where to find it: On the **Plug-Ins** menu, under Main Tools, click CoroPlus Tool Library.
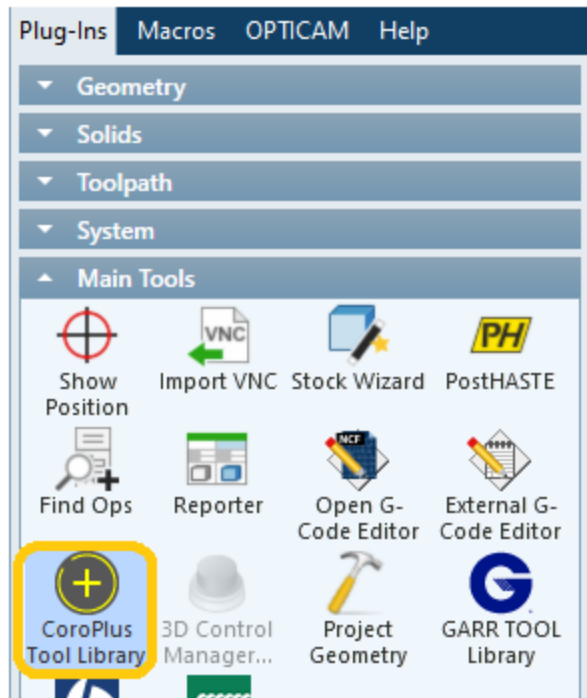
Starting CPTL Using the Process Flyout

Open the Select Tool Type flyout, either by double-clicking a blank process tile and selecting a process or by right-clicking an occupied process tile and choosing context menu item Change Tool. Then, in the Select Tool Type flyout, click CoroPlus Tool Importer.
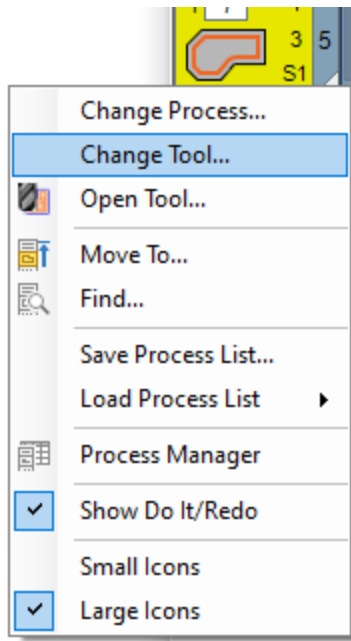
Context menu from right-clicking an occupied process tile.

Select Tool Type flyout

If you are not already logged in to your Coromant account, the Login dialog prompts you to enter your credentials.

Result: After the connections are initialized, the **CoroPlus Tool Importer** dialog appears.
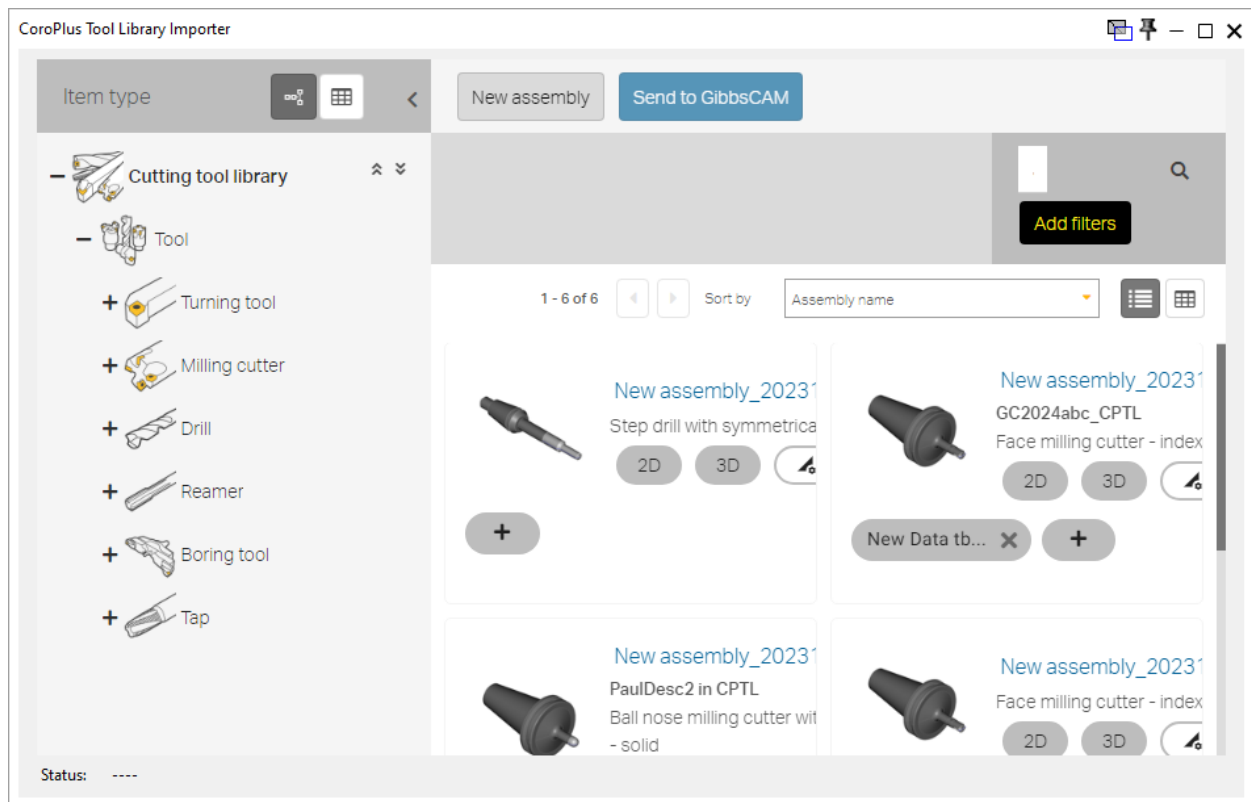
Within CPTL, Create and Export an OptiThreading Assembly

To create, modify, and export an OptiThreading tool assembly, follow these steps.

1. Create a new tool assembly: **By application** > **Symmetrical rotating**.

Create a new tool assembly

By tool type ⓘ

Turning tool	Milling cutter	Drill	Reamer	Tap	Cartridge

By application ⓘ

Symmetrical rotating	Non rotating
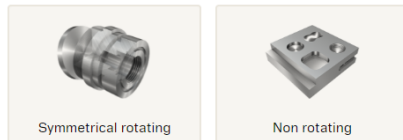
Recent assemblies   My assemblies (2)

New assembly_202...	New assembly_202...
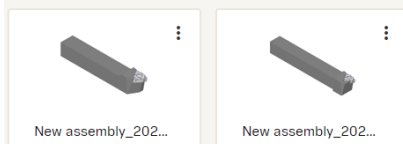
2. Under **Task**: Select the workpiece material (for example: Low alloy steel).

3. In response to the prompt, click **Workpiece parameters** and then choose workpiece geometry by clicking the labels attached to the sample picture. For example:

4. In the ensuing boxes and drop-down menus, specify parameters for the workpiece, such as TDZ=Thread diameter size; THL=Threading length; THDH=Thread hand; TCTR=Thread 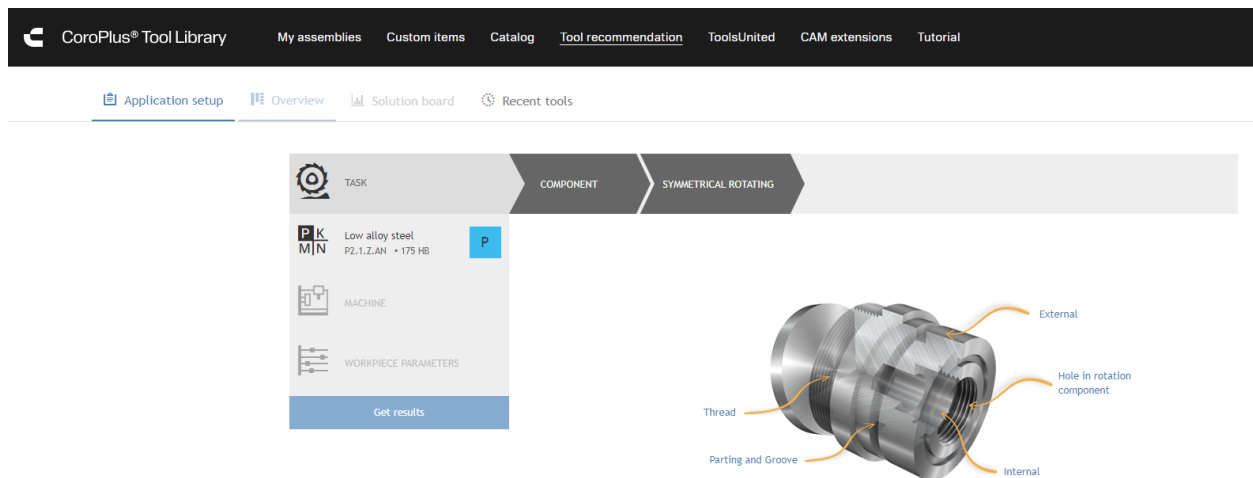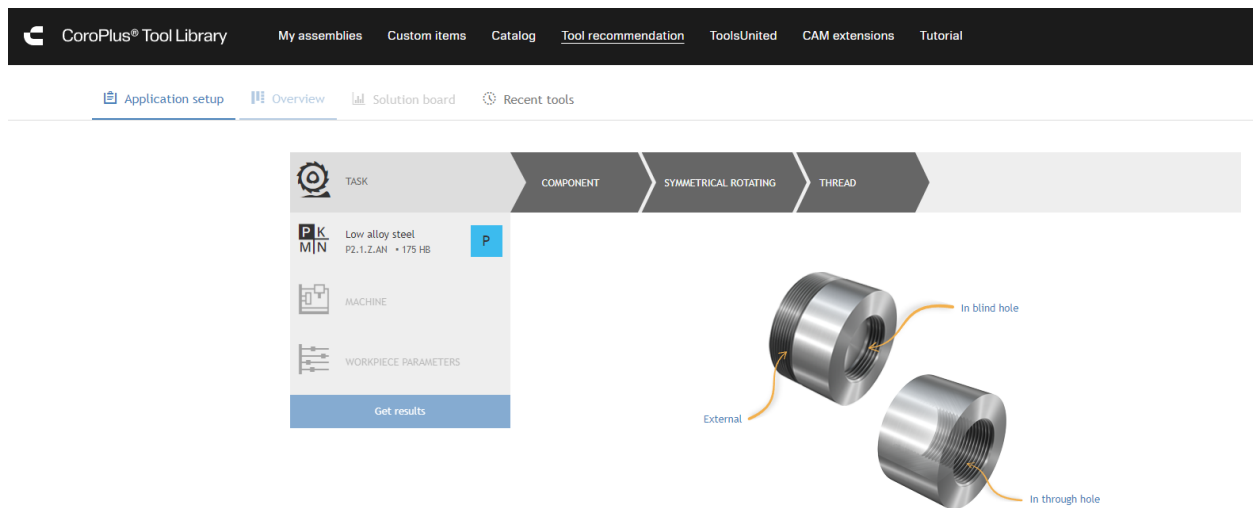tolerance class). Then, below the thread specifications, under the OptiThreading pulldown, toggle OptiThreading enabled from gray (OFF) to **ON** (light blue).

> **Please Note:** The **OptiThreading** toggle must be **ON** to enable the OptiThreading technology.

5. After completing your selections, click the **Get results** button at the bottom of the first column.

6. Click the **Build tool assembly** button near the bottom of the third ("Cutting Data") column.
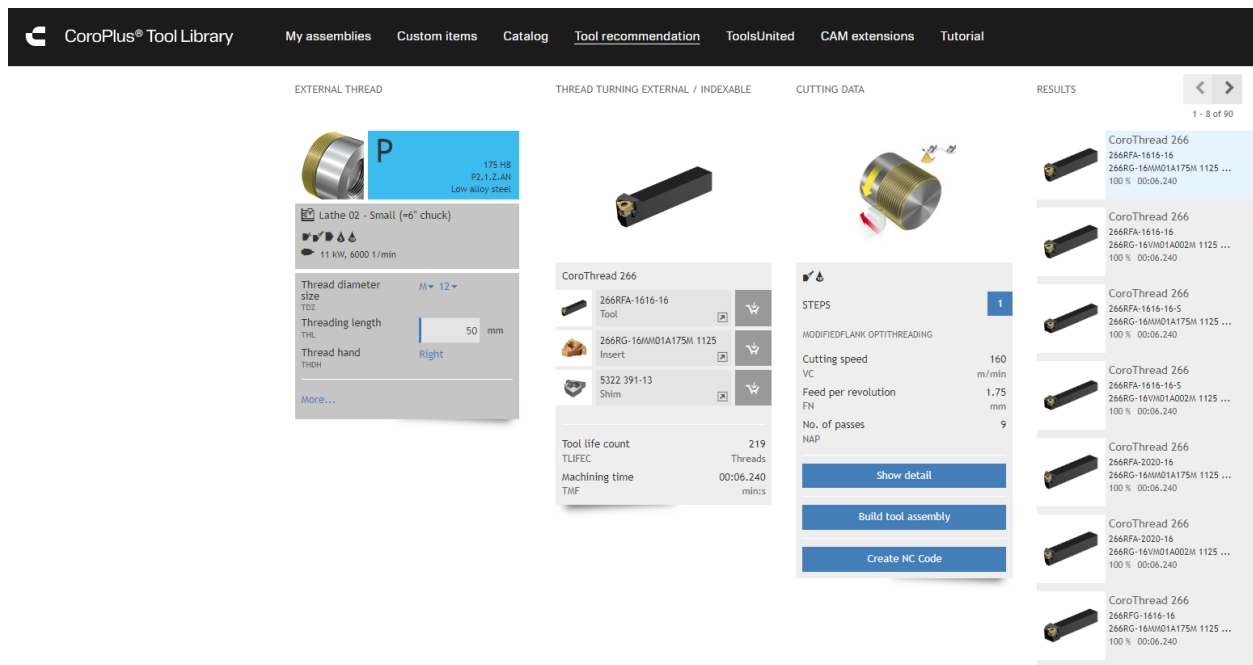
7. In the left column of the tool assembly screen, enter values for Name, Description, and Classification.

8. Optionally, use controls in the top center of the tool assembly screen:

The buttons rotate/reset the view rotation.

The button confirms/resets the view settings, as illustrated here:
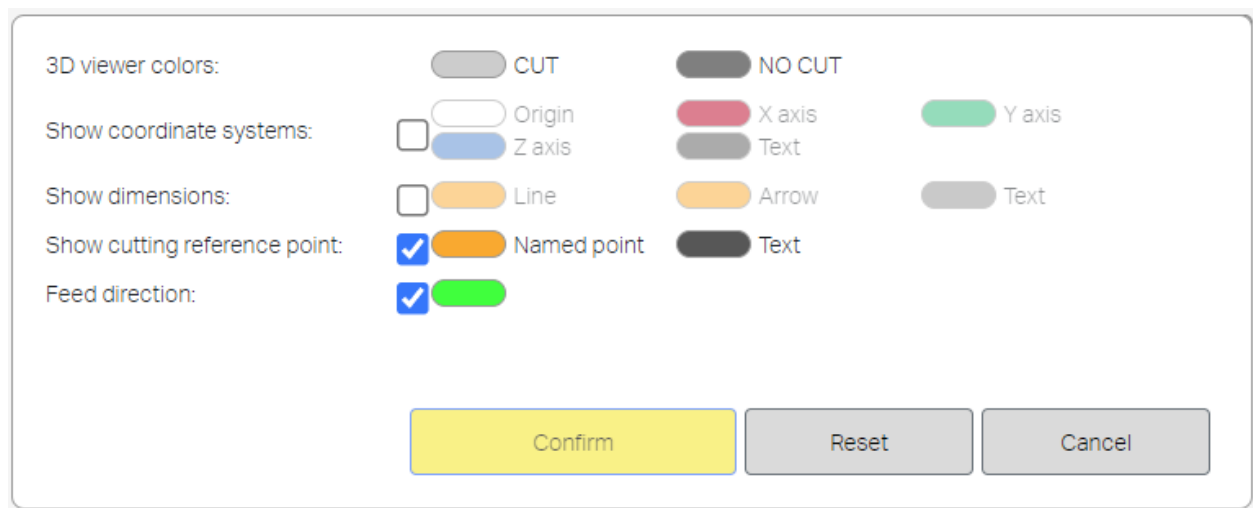
The ⬢▾ pulldown menu sets the style, from: 3D model, 2D drawing, or 2D family drawing.

You can also view the assembly's Properties (▤), Parameters (⇅), and Connections (⚇).

9. In the left column of the tool assembly screen, click the **Save** button (or **Save As**).

10. After the assembly is saved, you can optionally click the Cutting Data (◢) button and use options like Get recommendations or Specify own cutting data. You can override (✴), edit(✏), copy(⬄), or delete(🗑) the data displayed in this screen.

11. When you are satisfied with the tool assembly and its data, click Export GTC package.

Result: The OptiThreading tool assembly is exported and available for selection in the GibbsCAM CPTL plug-in(⊕). In the plug-in, after a GibbsCAM user selects this assembly, clicks the Send to GibbsCAM button, and sets the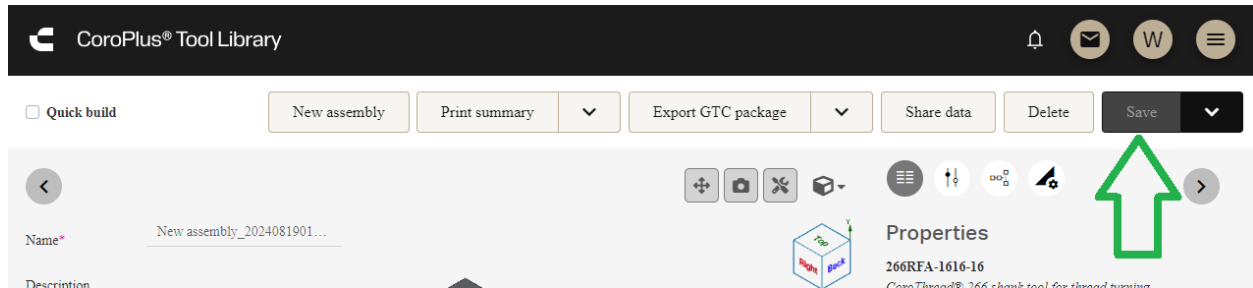 tool orientation, the corresponding tool is brought into GibbsCAM, where it is associated with cutting data (as shown by the ☑ mark in the Cutting Data (Cutting Data ☑) button in the **Tools** dialog).

Within the GibbsCAM Turning **Thread** process dialog, this data can be accessed by clicking the Tool ☑ (Tool ☑) button. More importantly, the dialog's **Definition** section offers the option From Cutting Data, which provides access to the options and parameters specific to OptiThreading.

When to Use the OptiThreading Process

High-Precision Applications: OptiThreading is ideal for applications that require high precision and accuracy in threading operations, as in aerospace, automotive, medical device manufacturing, and any industry where precise thread profiles are critical.

Complex Thread Profiles: It is particularly beneficial for cutting complex thread profiles that are difficult to achieve with traditional threading methods. This includes multi-start threads, tapered threads, and nonstandard thread geometries.

Material-Sensitive Operations: When working with materials that are challenging to thread, such as high-strength alloys and exotic materials, OptiThreading optimizes cutting conditions to minimize tool wear and avoid workpiece damage.

Tool Life Optimization: To extend tool life and reduce tool costs, optimization of cutting parameters reduces the wear on threading tools, leading to fewer tool changes and lower overall costs.

Surface Finish Requirements: In scenarios where an excellent surface finish is required, such as in the production of critical components like hydraulic fittings and sealing surfaces, OptiThreading ensures superior surface quality by minimizing deflection and vibration during the threading process.

Long chips: Long chips tangling around workpiece and/or tool can cause unwanted stops in production. OptiThreading will help in these scenarios due to the oscillating movement in and out of the workpiece.

Why OptiThreading Is Used

Enhanced Efficiency: OptiThreading significantly improves the efficiency of threading operations by optimizing cutting parameters such as speed, feedrate, and depth of cut. This leads to faster production cycles and increased throughput.

Improved Thread Quality: By carefully controlling the cutting process, OptiThreading produces higher-quality threads with better dimensional accuracy and surface finish. This is crucial for components that must meet strict tolerances and quality standards.

Reduced Tool Wear: OptiThreading reduces tool wear by optimizing cutting conditions, which leads to fewer tool changes and lower tooling costs. This is especially important when working with or ductile abrasive materials that can quickly degrade cutting tools.

Minimized Material Stress: The optimized cutting strategy minimizes stress and deformation in the workpiece material, reducing the risk of defects such as thread tearing or surface cracking.

Versatility: OptiThreading can be adapted to a wide range of threading operations and materials, making it a versatile solution for various manufacturing environments. It supports different threading types and can be used on both CNC lathes.
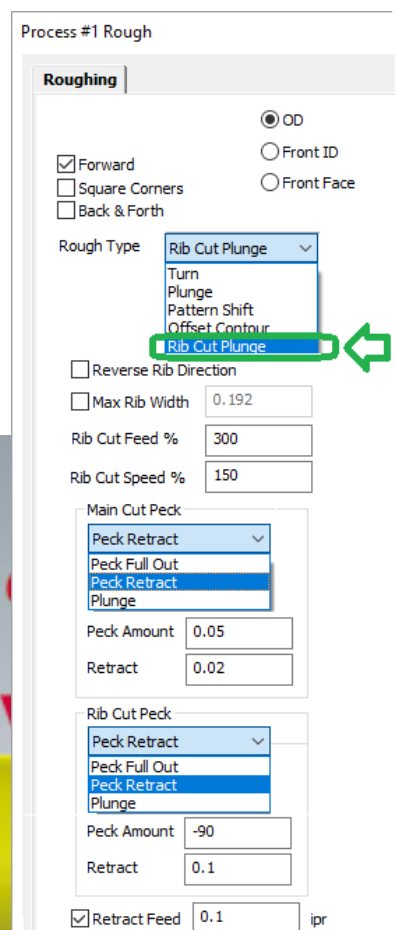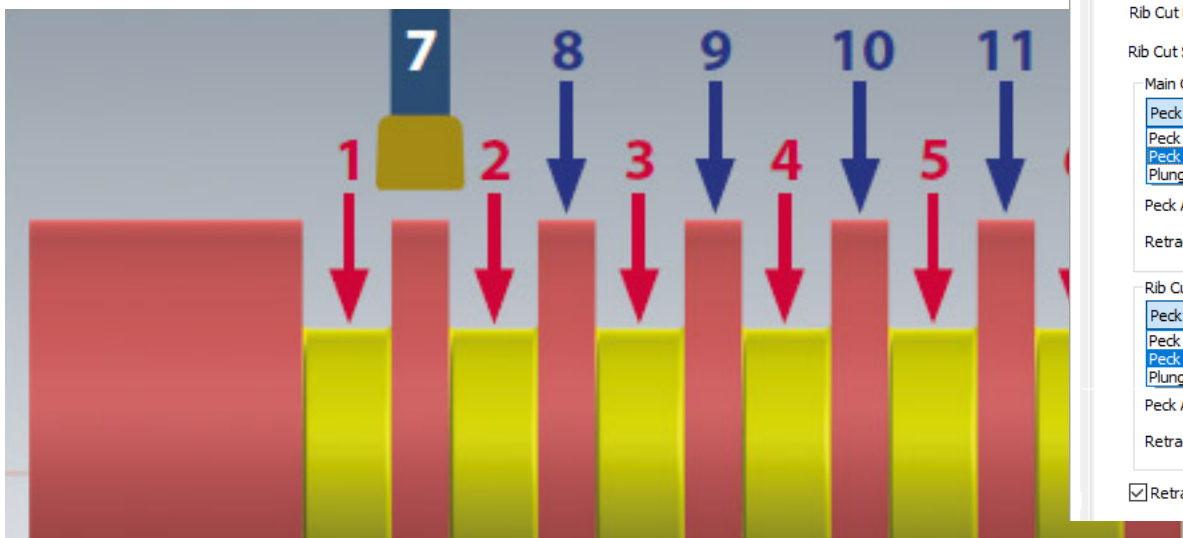
 Competitive Advantage: Manufacturers implementing OptiThreading can achieve higher productivity, better thread quality, and reduced operational costs, providing a competitive advantage in the market. This technology lets companies meet stringent customer requirements and deliver superior products.

In summary: OptiThreading is used in situations where precision, efficiency, tool life, and surface finish are critical. Its ability to optimize the threading process makes it an invaluable tool in modern manufacturing, particularly in industries demanding high-quality threaded components.

Roughing > Rib Cut

Where to find it: Turning > Rough; for Rough Type, choose Rib Cut Plunge.

This new Turning strategy consists first of a preliminary pass where the tool repeatedly plunges into the part at full engagement to create a series of cuts with each cut more than a tool-width away from the previous one. Then the next pass cuts the remaining "ribs". Rib cutting avoids tool deflection in both passes, and it offers excellent chip control. The rib cuts can be made safely at high speed to decrease the total amount of cycle time.

Process #1 Rough

Roughing

- ☑ Forward
- ☐ Square Corners
- ☐ Back & Forth

◉ OD
○ Front ID
○ Front Face

Rough Type   Rib Cut Plunge ⌄

| Turn |
| Plunge |
| Pattern Shift |
| Offset Contour |
| **Rib Cut Plunge** |

- ☐ Reverse Rib Direction
- ☐ Max Rib Width   0.192

Rib Cut Feed %   300

Rib Cut Speed %   150

Main Cut Peck

Peck Retract ⌄

| Peck Full Out |
| Peck Retract |
| Plunge |

Peck Amount   0.05

Retract   0.02

Rib Cut Peck

Peck Retract ⌄

| Peck Full Out |
| Peck Retract |
| Plunge |

Peck Amount   -90

Retract   0.1

☑ Retract Feed   0.1   ipr

Swiss-style machines: Sliding headstocks and Z-axis limits

Where to find it: In Machine Manager, for node type ⊩🖵 P (lathe spindle): ☑ Headstock Origin.
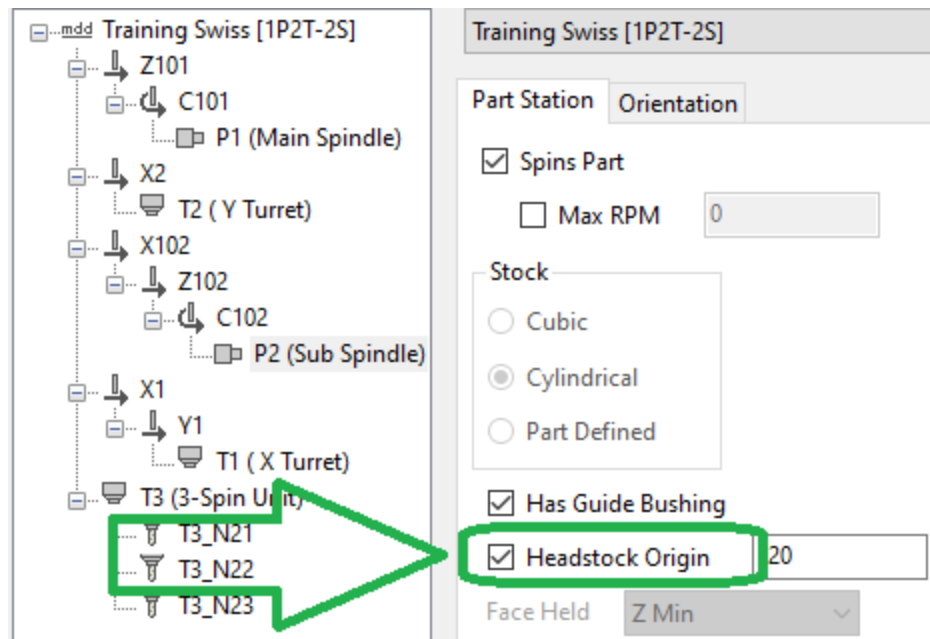
The new Headstock Origin checkbox lets you provide a value for the distance between the front of the guide bushing and your headstock's o position. This lets you correctly position Simulation bodies on the headstock axis, and it provides a more intuitive interface in the DCD for Swiss-style machines.

Visualization and User Interface

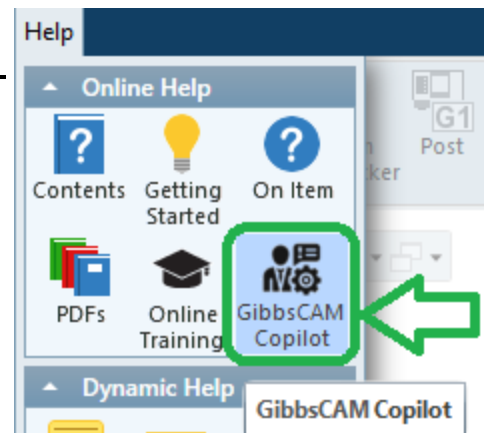## GibbsCAM Copilot: AI-powered chat tool

Where to find it: On the **Help menu**, click GibbsCAM Copilot.

GibbsCAM Copilot is an AI-powered chat tool — a ChatGPT LLM (large language model) that has been trained on a wide variety of source material relating to GibbsCAM. It lets you access information without having to search through documentation or call Technical Support. It can respond in any of the languages supported by the GibbsCAM user interface. A "thumbs-down" feedback mechanism allows you to register dissatisfaction with any reply. GibbsCAM Copilot works best when responding to specific queries that are narrowly worded.

Please note that, because some countries' laws relating to AI are not yet fully analyzed, the GibbsCAM Copilot chatbot is disabled in some countries.

GibbsCAM Copilot

Welcome to GibbsCAM Copilot, how can I help you today?

How can I access VoluMill chip control?

To access the VoluMill chip control in GibbsCAM, follow these steps:

1. **Create or Open a Process**: Either create a new VoluMill process or open an existing one by double-clicking its tile.

2. **Find the Chip Control Button**: In the VoluMill page of the process dialog, locate the "Chip Control" button.

3. **Open Active Chip Thickness Control (ACTC)**: Click the button to access the Active Chip Thickness Control dialog.

4. **Calculate Parameters**: Use this dialog to calculate cutting parameters like Speed, Feed, Cut Width, and Chip Thickness by selecting options from a dropdown menu and entering necessary values.

5. **Apply Changes**: After calculating your parameters, click "Apply Parameters" to update them in your VoluMill process dialog.

This will help you optimize your machining conditions effectively with VoluMill's capabilities.

Connected.

The provided response is AI-generated and must be reviewed for accuracy, correctness, and relevance before use. By using this Copilot, you acknowledge that you have read the Transparency Note and agree to the Terms and Conditions associated with GibbsCAM Copilot.

# Rotary Hints: Op Sim / Machine Sim indication of spindle direction

Where to find it: On the context menu for the **Rendering** dialog's title bar, enable Show Rotary Hints.
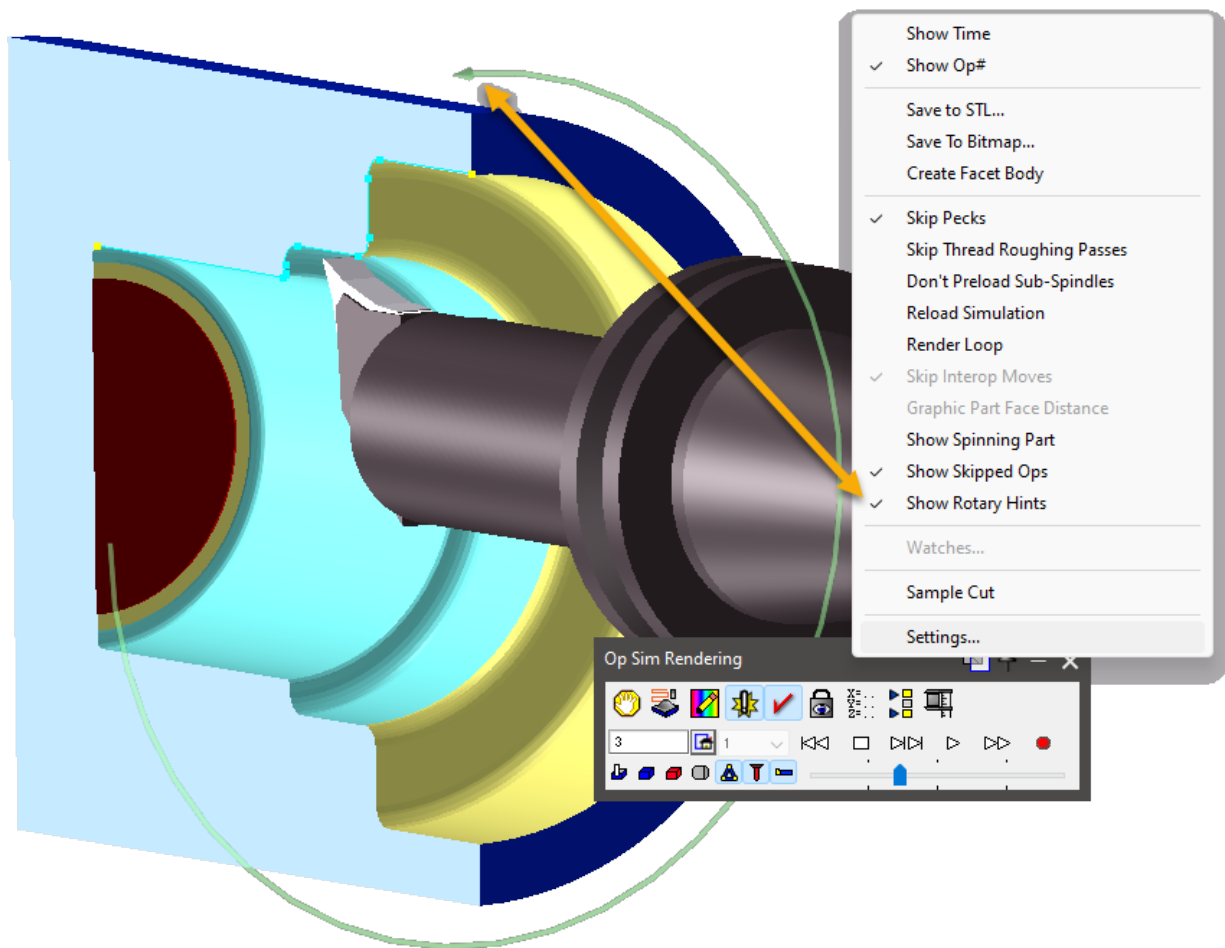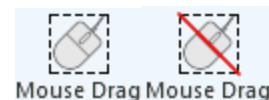
Quick and Powerful Improvements to Selection

Edit > (Selection >) ▦ Filter Selections

Where to find it: On the **Edit** menu, under **Selection**, click ▦ (Filter Selections) to open the **Selection Filter** dialog shown below.
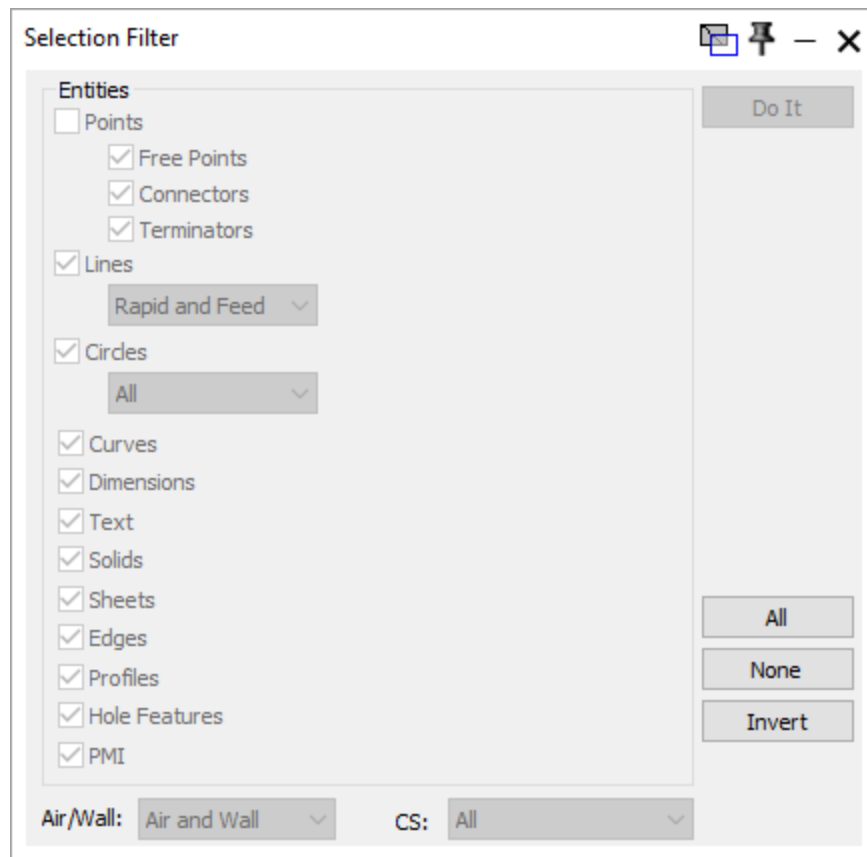
Items selected/deselected via the **Selection Filter** dialog can be stacked.

Deselect by Mouse Drag

Complementing the existing command "Select by Mouse Drag" is a new command, "Deselect by Mouse Drag". Each command has its own default keyboard shortcut, such as `Shift+drag` to select and `Ctrl+Shift+drag` to deselect. These actions can be stacked.

Containment and Overlap Mouse Drag: Left-to-right and right-to-left

In previous releases, left-to-right mouse drag selection has been inclusive: It selects any entity that is even partly within the drag region. A new feature is mouse drag selection that is exclusive: The system selects only entities that are entirely contained within the drag region.

File > **Preferences** offers a new toggle (left-to-right inclusive containment as in previous releases; or, instead, right-to-left inclusive containment coupled with left-to-right exclusive containment).

Simulation default: Collision Checking, Program Error Checking

Where to find it: In Sim dialogs, for new parts, notice that **Collision Checking** and **Program Error Checking** are now enabled by default.

Sync Manager during Sim: Graphical indicator of Simulation progress

Where to find it: In the **Sync Control** dialog (except in Uniform view) while Simulation is actively running, a red bar indicates the timeline of the simulation as it progresses through the flows.
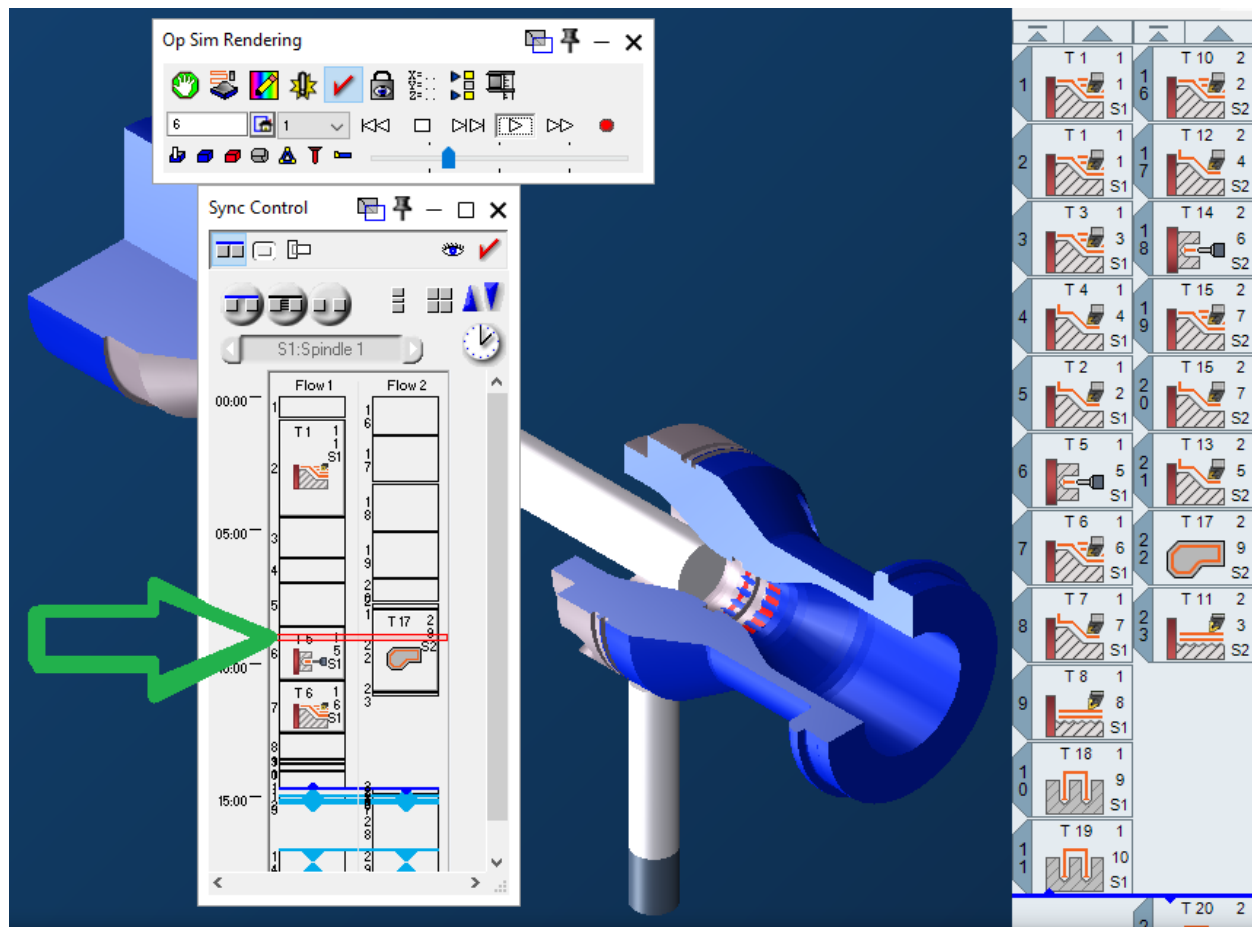
Op Sim Analyze Cut Part can use user-set tolerances

Where to find it: With a solid selected, run Op Sim and then, after completion, click to open the **Analyze Cut Part** dialog and supply a value for Tolerance:
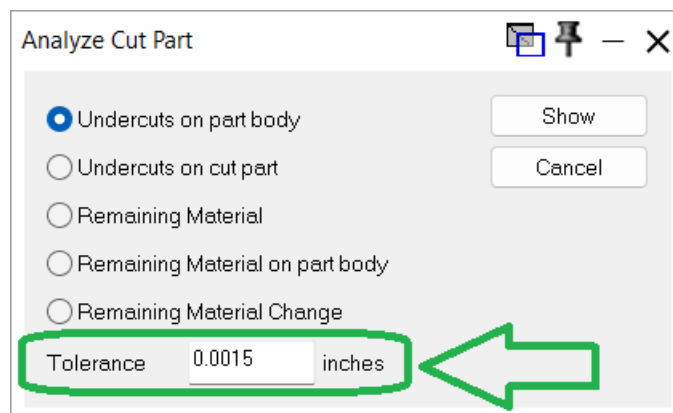
New Stop type in Simulation: Stop Before Every Operation

Where to find it: In any **Sim Rendering** dialog, click to open the **Stops** dialog. If you want to review every operation individually, check the Stop Before Every Operation checkbox before running Sim.
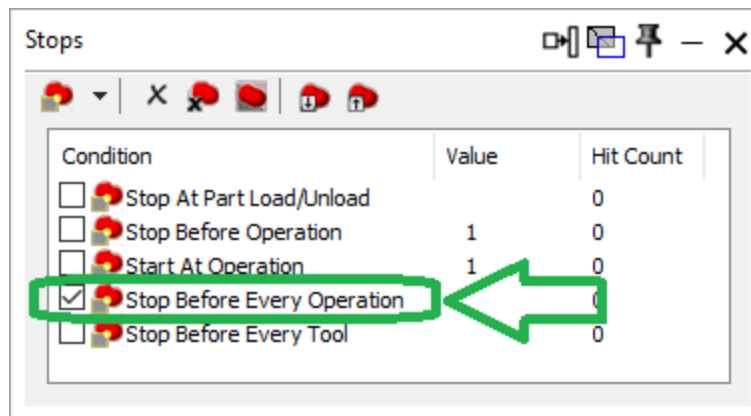
New modes for Op Tile List: Columns for flows

Where to find it: Near the top of the Op Tile list, a new set of buttons — — lets you set the mode for displaying flows:

- Show All Flows

- Show Each Flow

- Sync Flow Mode
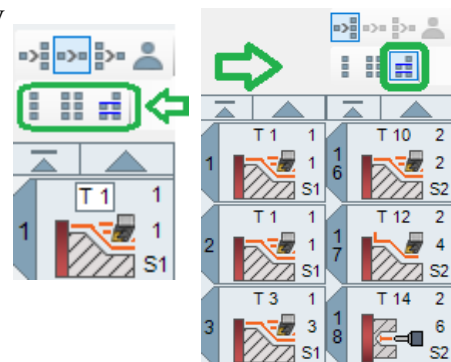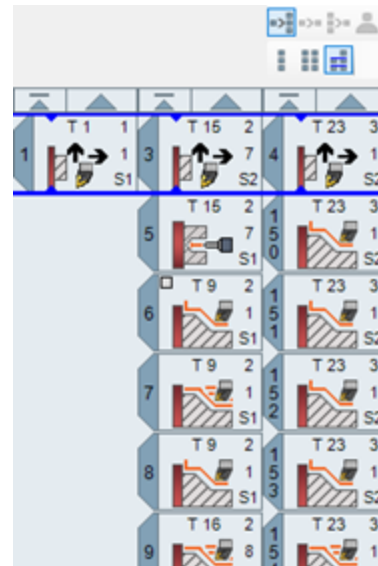
| Show Each Flow | Sync Flow Mode |

Quick-and-easy toolpath redraw

Where to find it: In **File** > **Preferences** > Interface, in the upper right of the pane, option buttons let you specify the behavior of single-clicking an Op tile-tab or single-clicking an Op tile.

You can set a preference to redraw toolpath in one easy action either when you click an operation tile tab number or when you click an op tile itself.
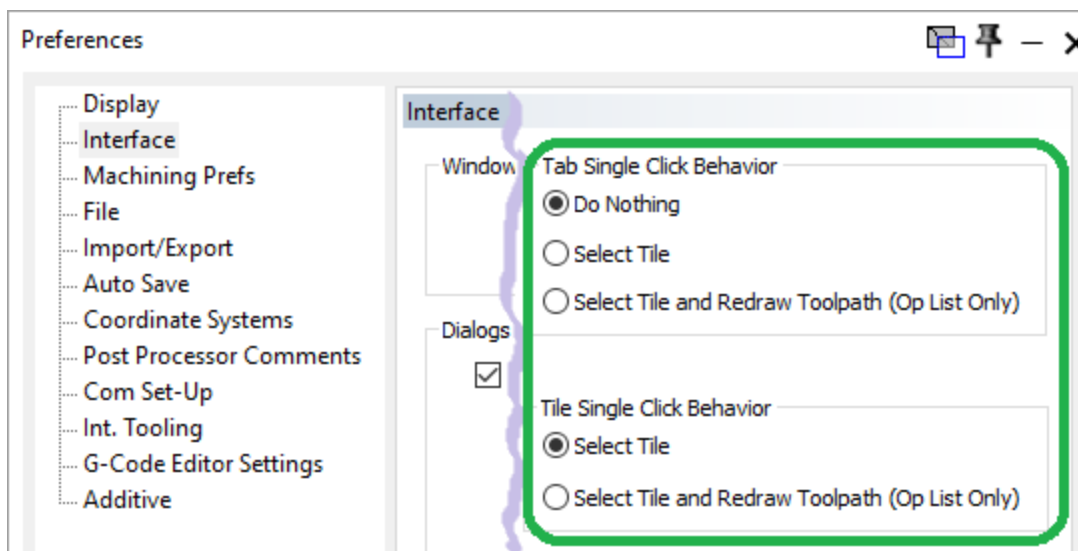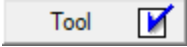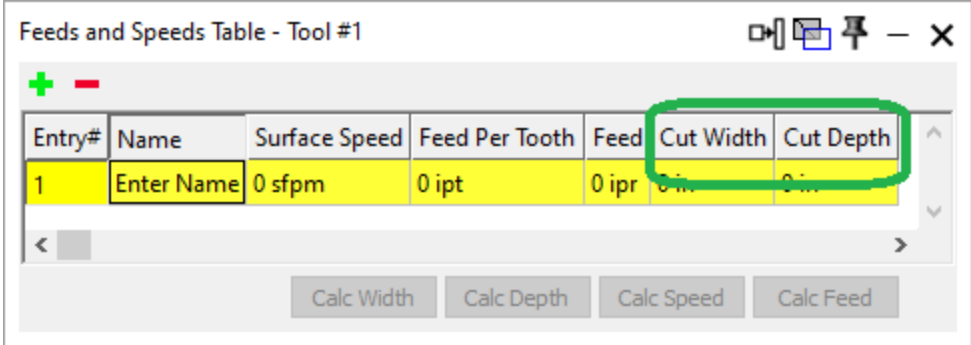
Tooling

GibbsCAM 2025 brings further improvements to tools and tooling.

More tool parameters stored with tool data, such as Ae and Ap

Additional tool cutting parameters like Ae (Cut Depth) and Ap (Cut Width) can be stored with tool data. For tools that have such parameters imported from systems like CPTL = CoroPlus Tool Library, these additional parameters are automatically available.

Where to find it: The **Tool** dialog and most process dialogs have a button that signals the presence of tool cutting parameters stored with the tool. Clicking this button opens a dialog resembling the following:

Also see additional parameters for tools imported by CPTL (CoroPlus Tool Manager) in "OptiThreading".

Order Tool

When cutting tools are imported via catalogs that GibbsCAM supports (such as CoroPlus Tool Library, Harvey Tool, Helical Solutions, and Garr Tool), the GibbsCAM **Tool** dialog presents a link: Order tool
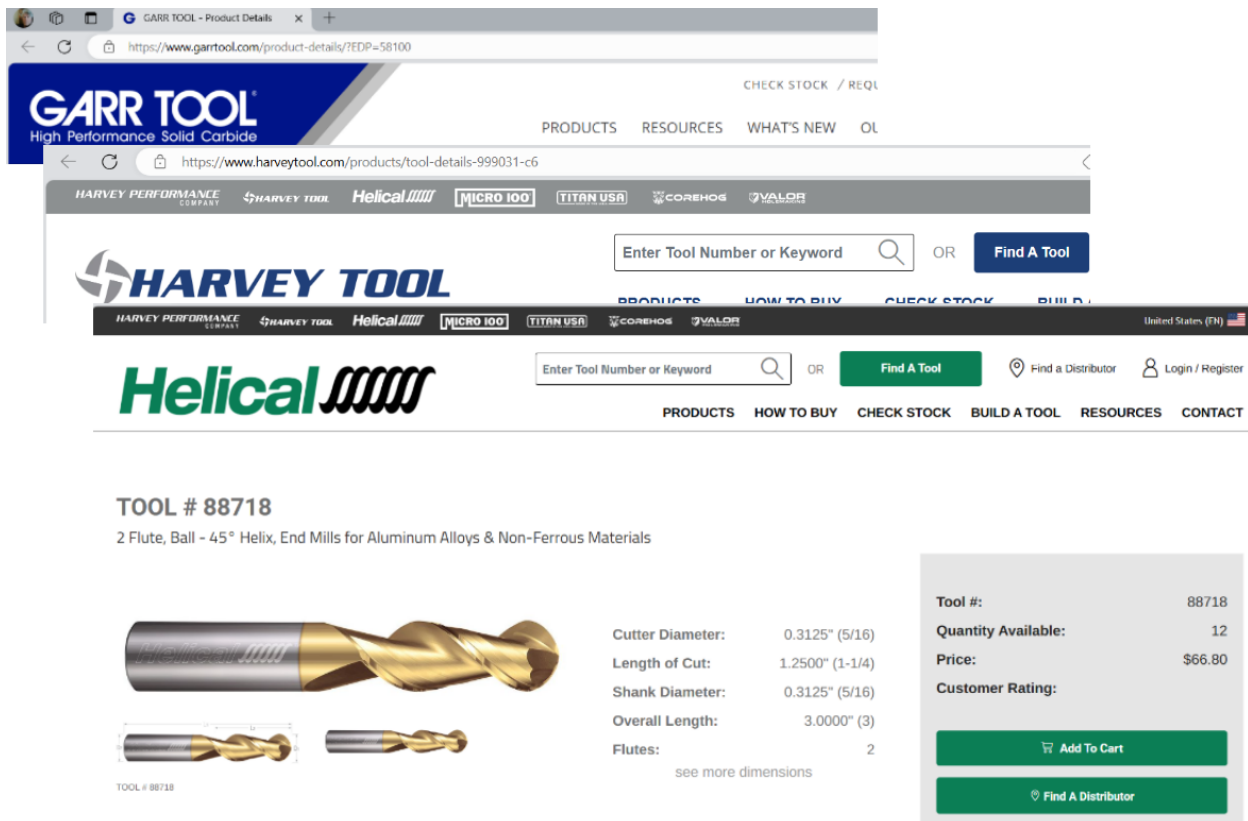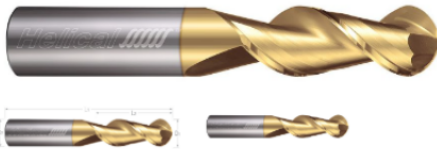
Clicking the **☰ Order tool** link directs you to the catalog of the specific tool library.

Advantages: Clicking the link takes you to the manufacturer's catalog so that you can view/inspect/compare the current tool and possibly import it from the manufacturer's ordering page for that tool. If you import from a catalog, GibbsCAM keeps track of the tool's item/catalog number and manufacturer.

Posting and Output

GibbsCAM 2025 includes improvements and enhancements to output capabilities.

TMS now supports full WFO (Work Fixture Offset) functionality

Where to find it: In the **Post Processor** dialog, choose the **Workfixtures** tab.
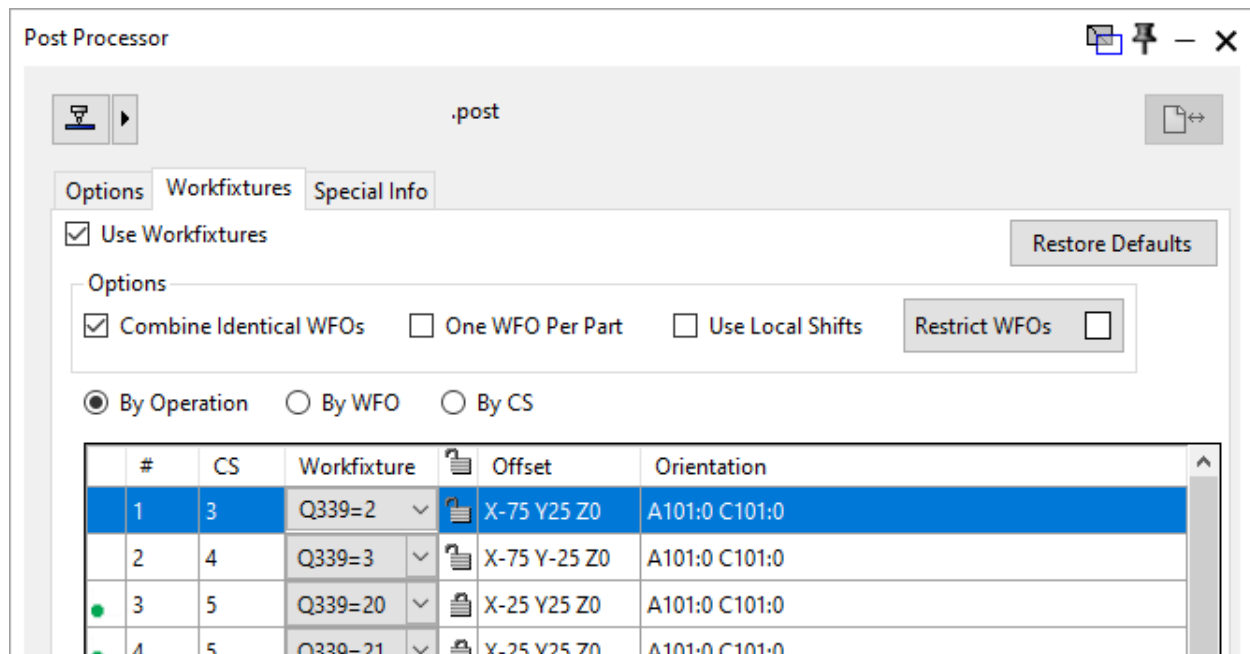
Postprocessor support for machining tolerance in solid ops

There are new commands for querying/outputting the operations machining tolerance, so that you can learn (for example) whether you can turn on a particular kind of look-ahead.

> **Please Note:** This enhancement requires a post upgrade.

Solids

GibbsCAM 2025 brings further improvements to solids and surfaces.

Edit Facet Bodies

Where to find it: The main tool palette has a new command:
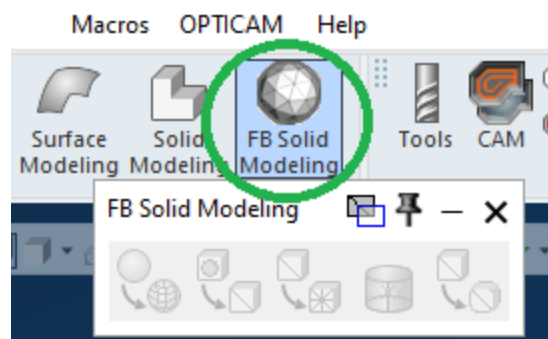
Clicking this ⬡ command opens the **FB Solid Modeling** palette, which lets you edit facet bodies using most of the tools you have for solid geometry, such as: Boolean operations (union, subtraction, intersection); Slice; Offset/Shell; Separate; Unstitch Body; Shrinkage; History; Check Body Validity; and calculations of surface area and volume. Just as with solid bodies, each modification to a facet body is preserved in its History Tree. When a facet body is combined with a regular (B-Rep) solid body via Boolean commands, the result is a facet body with a history.

The following new functions are specific to facet bodies: Heal Facet Body; Smooth Region; Reduce Complexity; Refacet; and Tesselate.

Collision Avoidance of Fixtures

Where to find it: In many process dialogs, the **Solids** tab now offers a new checkbox: Avoid IT Fixtures. For Advanced 3D processes, this checkbox is in the **Boundary** tab.
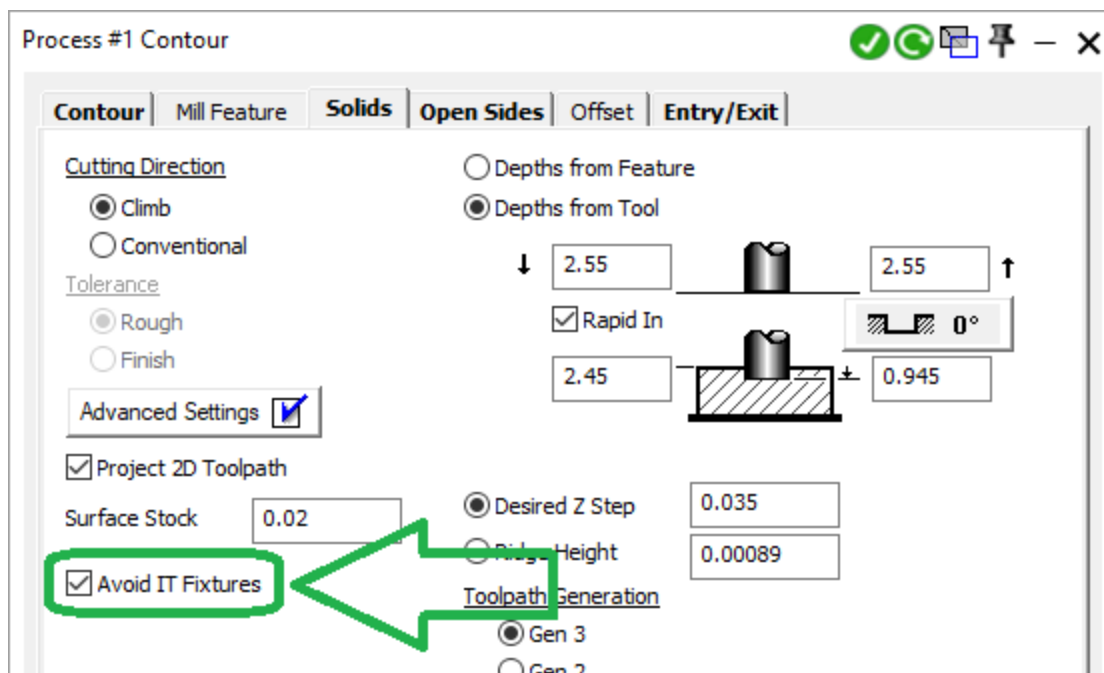
When this checkbox is enabled, collision avoidance is aware of both Intermediate Tooling fixtures and workspace fixtures, allowing it to avoid gouging clamps, jigs, etc.

Also: In parts whose MDDs define Clearance Volume, clearance transition moves now automatically avoid collisions with all types of fixturing if possible. For Advanced 3D processes, you can fine-tune this by entering a value in the new Fixture Clearance text field in the **Boundary** tab.

Miscellaneous

System requirements are unchanged from GibbsCAM 2024:

- *OS:* Windows 10, 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

VoluMill 9.6 is now supported.

Because GRANITE does not support importing the latest PTC Creo files, the legacy GRANITE import options have been removed; the Spatial options are retained.

Note: Add-ins for Cimatron 15 and 16 are still supported, but they no longer appear in the InstallShield Wizard, because they are now installed with Cimatron rather than GibbsCAM.

Copying/pasting of solids preserves colors and other attributes

When you copy-and-paste a solid, all attributes on the solid, faces, and edges are now preserved, including not just Color attributes but also attributes of types Feature, Real, Integer, and Text.

How to use it: Open two sessions of GibbsCAM. In the first session, open a GibbsCAM part that has attributes (such as color or PLM). Copy part or all of it and then paste into the second session.

NLO License borrowing with checkout/return/expiry parameters

Users can now check out an NLO license for a specified time period, then disconnect from the network. This allows users to borrow licenses, disconnect, move their laptop, and keep using the GibbsCAM products even when the license server is stopped. Borrowed licenses can be returned manually, or they can be set to expire after a set time.

Where users can find this option: Under **File** > **Preferences** > Interface, click License Options; then, in the **License Manager** dialog, click Start Roaming.
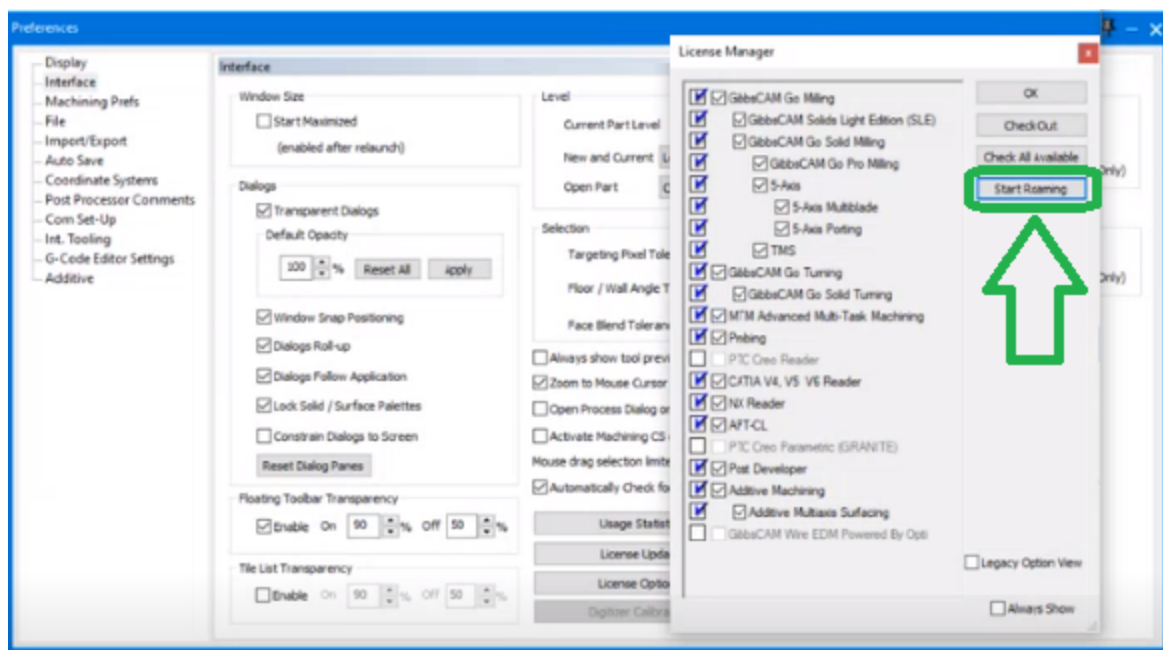
How administrators can support this option: Reprise License Server Administration shows the count of roaming licenses in its License Pool Status:

Op Manager, Process Manager support for VoluMill data fields

Where to find it: Open a VoluMill model and access Operation Manager or Process Manager, as illustrated below.

Op Manager, Process Manager support for many 5-Axis data fields

Where to find it: Open a 5-Axis model and access Operation Manager or Process Manager.

MDD limits on maximum spindle RPM

MDDs in GibbsCAM 2025 can set a maximum spindle RPM. Exceeding the maximum RPM will result in a program error.

Where to find it: Machine Manager > Tool Station tab (☑ Live Tool): **Max RPM**	
Where to find it: Machine Manager > Part Station tab (☑ Spins Part): **Max RPM**	

5-Axis

This chapter discusses the following enhancements to 5-Axis in GibbsCAM 2025.

Note: For illustrations and full descriptions of 5-Axis functions, see the *5-Axis* guide.

Multiblade

Edge Rolling for Leading and Trailing Edges - With this enhancement, leading and trailing edges can now have their own edge rolling settings. This gives users more flexibility and allows them to define how the leading and trailing edges should be individually machined.
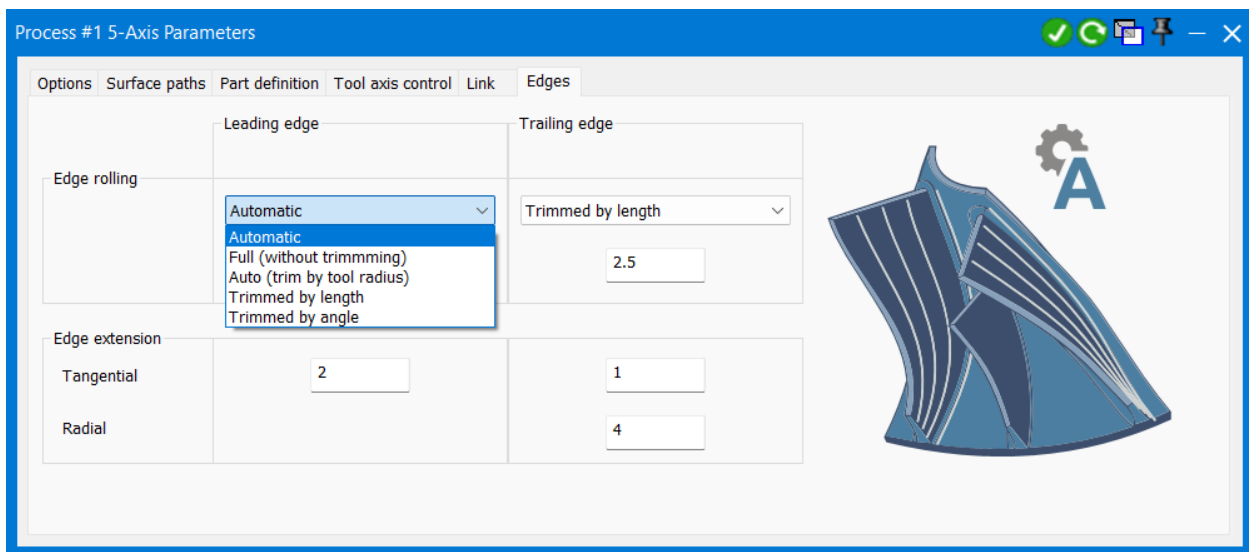
Feedrate for Leading/Trailing Edge - The toolpath can now have different feedrates at the leading and trailing edges, providing greater control. This results in improved surface quality, more consistent cutting, and smoother machine tool movements.
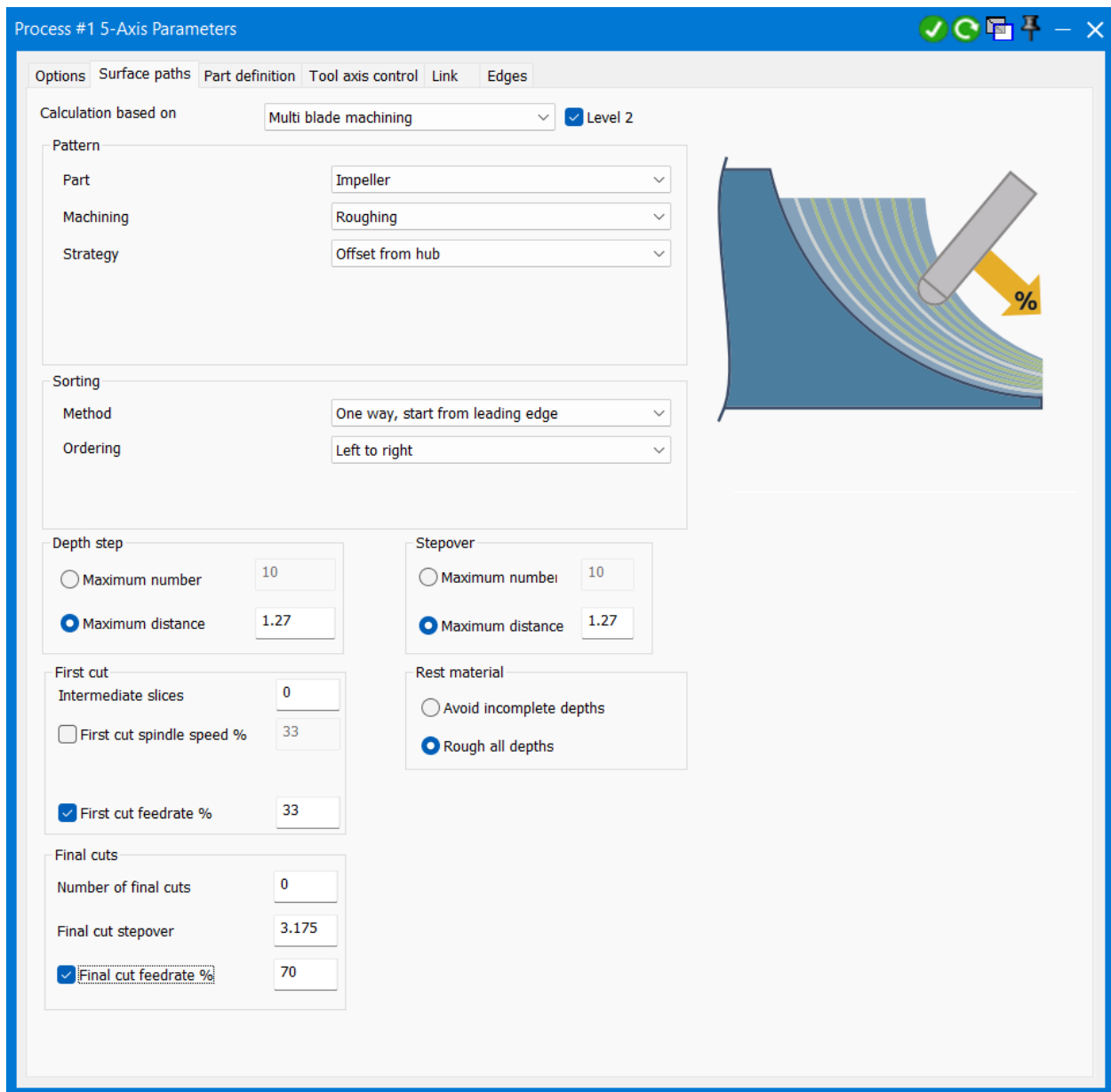
Core Technologies / Linking

Fade Lead In/Out - This new feature adds a new lead type called "Fade" that moves slowly away from and towards the machining surfaces.

Automatic as Default - This new feature changes the default settings for the linker. Setting the default for clearance to "automatic" results in more successful toolpath generation without any user interaction for clearance inputs. This reduces the programming time.

3-Axis Machining / 3-Axis

Machine Flatlands after Each Slice - The new option 'Machine Flatlands after Each Slice' machines flatlands on top surfaces after each slice. This option, along with DHC, enables the tool to reach deeper pockets, which would otherwise not be possible to reach.

3D Contact Point Boundary for Finishing - The new implementation of the 3D contact point boundary accurately limits the finishing toolpath to the 3D containment on the part, improving the surface quality. Users only need to define the cut area on the actual part, and the calculation automatically limits the toolpath to ensure the tool contacts the surface exactly at all the points. Users no longer need to recreate the containment or manually add the offsets.

Optimal XY Angle for Flatland Milling - An option to optimize the XY angle has been added to the flatland toolpath. It adjusts the angle for separate regions independently to optimize material removal and surface quality. This is achieved by aligning the parallel passes with the longest length of the relevant area.

Ordering by Region - The parallel cuts toolpath can now be generated using new region-based ordering. This provides better surface quality as there are no engage or retract moves in a region. Each region is milled completely and then the tool moves to the next region. This option is offered in addition to the shortest path ordering option.

Store Contact Point Normals - This new enhancement for 3-axis finishing toolpaths stores the contact-point normals along with the toolpath in the structure. These contact-point normals can be used subsequently for 3D cutter compensation or tilting in the 3-axis finishing strategy.

Outside-In Ordering for Flatland Milling - This enhancement enables flatland toolpath to order the passes from the outside to the inside of the island or region. This helps to improve surface quality and greatly increases the tool life, because the tool no longer engages directly on the surface from the top.

3D Machining / 3-Axis

Improved Stock Aware Linking - The stock awareness for linking has been improved. In addition to checking against the part, in-process-stock is now also checked during linking motions. This minimizes the number of link and ramp motions while guaranteeing a collision-free toolpath.

Optimized Link Height - The improved in-process-stock awareness optimizes link heights and keeps them to a minimum. This reduces linking motions by up to 30% and therefore reduces machining time while maintaining machining safety.

Where to find it: tab "Link" > dialog "Retracts" (lower left) > checkbox "Optimize lead and link distances"

Optimized Ramp Height - The improved in-process-stock awareness optimizes ramp heights and ensures the ramps start exactly at the height of the in-process-stock. This reduces air cutting and machining time while ensuring optimized entry.

Where to find it: tab "Link" > dialog "Retracts" (lower left) > checkbox "Optimize lead and link distances"

Smoothing Control for Offset - Offset toolpaths for complex and detailed features are smoothed out while preserving the original last contour. This reduces unnecessary jerking motions caused by changes in acceleration and deceleration during the machining process and ensures constant, high-speed roughing.

Where to find it: tab "Roughing" > dialog "Advanced" > checkbox "Smooth contours" / textbox "Smooth deviation (stepover %)"

Orthogonal Extension for Leads - In addition to the tangential extension for lead motions, a new orthogonal extension has been added for profile passes. This is especially useful when an extension is necessary for switching to CRC mode on the CNC machine, because it minimizes the space needed to perform the switch. This is particularly convenient when machining small pockets.

Where to find it: tab "Surface Paths" > dialog "Profile Pass" > textbox "Orthogonal line length"

Corner Detection Threshold - The corner detection threshold defines the maximum adjacent angle between adjoining faces for a pencil toolpath calculation. The toolpath is not calculated for corners with angles larger than this threshold. By adjusting the threshold, the user can optimize the toolpath and avoid unnecessary or undesired machining.

Where to find it: tab "Surface Paths" (area"Surface Quality) > checkbox "Corner detection threshold"

Optimal XY Angle - An option to optimize the XY angle has been added to the parallel cuts pattern. It adjusts the angle for separate regions independently to optimize material removal and surface quality. This is achieved by aligning the parallel passes with the longest length of the relevant area.

Where to find it: tab "Surface Paths" (area"Pattern") > checkbox "Optimal machining angle in X,Y"

Advanced Filtering for Finishing - A new Inscribed circle option has been added to the filtering options for finishing cycles. This provides additional user-control for filtering unnecessary toolpath segments.

Where to find it: tab "Surface Paths" (area"Area") > dialog "Filtering" > (Filter by: Regions) > "Type" pulldown choice "Inscribed circle"

Spiral Machining for Undercuts - Constant Z undercut machining now offers an additional spiral ordering option for generating a spiral toolpath for undercut machining. This new option ensures a continuous, seamless machine motion and thus improves machining performance as well as surface quality.

Where to find it: tab "Surface Paths" (area"Sorting") > "Cutting Method" pulldown choice "Spiral"

Report Minimum Shaft Length - A new Report minimum shaft length feature is now available for finishing cycles. It calculates the minimum shaft length required to reach all the areas to be processed within the specified machining height limits. The tool can be adjusted to ensure collision-free machining.

Where to find it: tab "Gouge Check" > dialog "Advanced parameters" > checkbox "Report minimum shaft length"

Improved Stock Aware Linking - The stock awareness for linking has been improved. In addition to checking against the part, in-process-stock is now also checked during linking motions. This minimizes the number of link and ramp motions while guaranteeing a collision-free toolpath.

Where to find it: tab "Surface Paths" (area"Pattern") > DHC=dialog "Dynamic Holder Collision" > checkbox "Check with in-process stock"

Improved Peg Loops - The corner peg loop strategy has been improved. It now merges the peg loop segments into offset contours to avoid changes in the machining direction. This improves machining performance and increases tool life.

Where to find it: tab "Roughing" > dialog "Advanced" > checkbox "Smooth Links"/textbox "Smooth link gap size (stepover %)"

Adaptive Roughing

Zig-Zag Threshold - The new Zig-Zag Threshold ensures that corner regions smaller than the given threshold are connected via the one-way trochoidal method, while zig-zag linking is maintained for longer toolpath contours. This improves the machining conditions while keeping the toolpath length short.

Where to find it: tab "Surface Paths" (area "Sorting") > checkbox "Zigzag threshold" / textbox "× Tool diameter"

Start Point Control –

Core Technologies / Tilting Core

Strict Preference for Automatic Tilting - A new feature is now available for the Automatic tilting gouge check strategy. This feature restricts the degree of freedom for collision avoidance to one. This allows you, for example, to maintain a constant table rotation from the input data and use only the tilt axis to avoid collisions. Previously, the rotation also contributed to collision avoidance, which meant the table rotation sometimes showed reversals.

Where to find it: tab "Gouge Check" ("Tilt Tool"+"Automatic") > dialog "Parameters" (area "Behavior") > "Preference" pulldown choice "Rotary tilt (strict)"

2D Machining / 2-Axis

External Sharp Corners for Open Ends –

3D Machining / 3-Axis

Flatland Ignore Gaps –

Report Minimum Shaft Length –

Independent Filter for Closed and Open Contours –

Corner Peg Height Control for Ball and Bull Mills –

Parallel Cuts Ordering Enhancement –

Multi-Axis Roughing / Rotary Machining Roughing

Leads for Roughing - This enhancement adds lead-in/out options for rotary machining roughing toolpaths. These additional motions ensure the smooth entry and exit of the tool into and out of the material. This is especially useful for open pockets and also for rest machining operations. It generates smoother transitions and ensures optimum start points on open contours. Tool wear and the risk of tool damage are also reduced.

Multi-Axis Roughing / Area Roughing

Minimize Links - This enhancement reduces the number of ramps in the multi-axis roughing strategy by optimizing the start points for closed contours. This means that some ramps are removed and the tool, in these cases, engages from previously cut passes. This reduces the total cycle time of the operation while increasing tool life.

Avoid Small Profile Ramp - This improvement enables the minimum size of profile ramps to be controlled by a parameter. This helps to avoid small profile ramps, making it possible to use tools with wider non-cutting areas. It also improves tool engagement efficiency, and larger ramps means less ramping time.

Trim Line Ramps - With this improvement, line ramps are now trimmed to the stock. This ensures that the ramp length is appropriately defined according to the available stock, which in turn reduces the overall engagement time.

Multi-Axis Roughing / Geodesic Machining

Improved Corner Smoothing - This option allows smoothing corners in toolpaths. This, in turn, leads to better surface quality and also to shorter machining time, because the tool does not need to decelerate and accelerate abruptly.

Improved Medial-Axis Cuts - This feature makes medial axis cuts more powerful. With this option, users can add additional medial axis cuts that are either merged with the main cuts or added as finishing cuts. This greatly improves surface quality and reduces the manual effort required to clean up the cusps afterwards.

Multi-Axis Roughing / Wall, Floor, and Rest Finishing

Stepover by Cusp Height - This enhancement adds an option for the user to define the stepover based on the cusp height for advanced tools, such as barrel mills. With a cusp-based stepover, users can easily setup operations and avoid mistakes in manual calculations.

Tool Segment Selection for Tilting - This feature enables users to define tool tilting using a tool segment when working with advanced tools such as barrel tools. This gives users greater control over the tilting range as well as the tool contact point for operations.

Multi-Axis Roughing / Surfaces

Slot Mill Enhancements - This enhancement introduces a new option for slot mill tools. Users can now add chamfered edges to the tool. This ensures full coverage of available slot mill types.

Multi-Axis Roughing / Swarf Machining

Gouge Checking against Individual Tool Parts - With this update, it is now possible to select individual parts of the tool for collision checking. This feature gives the user more flexibility and control over which sections of the tool are included in collision checking.

Multi-Axis Roughing / Turn Milling

Radial Cuts - This new option adds a completely new cutting approach to turn milling. The 'standard' turn milling cuts are spirals that are created in the axial direction and increment (depth step) in the radial direction. The radial cuts option does the opposite: the cuts propagate slice-wise in the radial direction to the final cutting depth while a sidestep is applied to reach the next slice.

Multi-Axis Edge Finishing / Deburring

Corner Arc Support for Chamfer Tools - This new enhancement creates a continuous toolpath around inner and outer corners when using a chamfer mill without any intermediate links for 5-axis deburring. This reduces the linking motions and hence the overall cycle time for chamfering.

Multi-Axis / Automatic 3+2-Axis Roughing

Marker in 3+2 Roughing Simultaneous - Linking for the start and end of 5-axis transitions. Clearance blend spline feedrate for 3+2 roughing

Multi-Axis / Deburring

Allow Asymmetric Edge Shape –

Multi-Axis / Geodesic Machining

Containment Cuts Order –

Multi-Axis / Multi-Axis Machining

3D Containment Curve Offset –

Sort by Regions/Lanes – Allows the user to choose a sorting method for the finishing cycles.

Multi-Axis / Multi-Axis Profiling

Advanced "Run tool at contact point" –

Multi-Axis / Surfaces

Leads – Tool Orientation Normal To Lead is a new orientation type for leads that keeps the tool normal to the toolpath at all positions.

Leads – Along Tool Axis is a new lead type that follows the tool axis direction.

AT 2.0 – For Automatic Arc Leads With Automatic Tool Orientation and Barrel Tools allows AutoTilt 2.0 (AT 2.0 to work on automatic arcs for barrel tools.