GIBBSCAM 2023 CAM for Production Machining

Version 2023 : October 2022

What's New in GibbsCAM 2023

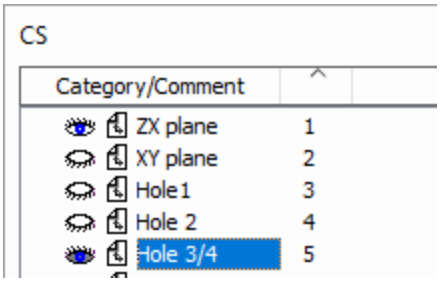
CAMBRIO

Contents

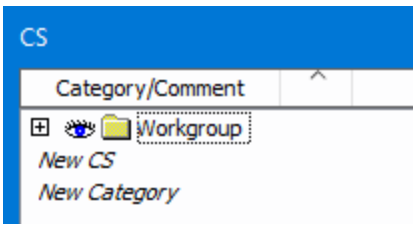
Part Organization and Ease-of-Use

Categories for Coordinate Systems and Workgroups

Coordinate Systems (CSs) and Workgroups (WGs) can now be organized into categories that appear throughout the system. You can create categories from the main toolbar using the Coord Systems or Workgroups dropdowns, or by using these steps in the **CS** or **Workgroup** dialog:



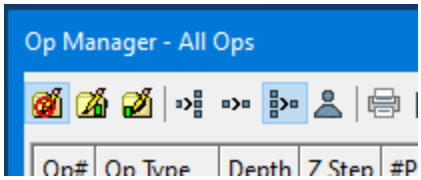
1. In the **CS** or **Workgroup** dialog, drag and drop to add CSs and WGs to categories.
2. To rename a category, click it twice.
3. To make all CSs or WGs in a category visible or invisible, toggle the eyeball for that category.



CS and WG dropdowns throughout the system, such as Machining CS and Patterns, have been updated to show categories.

Operation Groups in Op Manager

Grouped operation tiles now appear in Op Manager. You can use the Op Manager toolbar to change the grouping mode, which is synchronized with the grouping mode in the Op Tile list.



- When collapsed, each group shows only the values that are shared between operations in that group. Editing shared data in the operation or process changes the corresponding value for all operations in that group.
- When expanded, each operation's data is fully populated individually, and the data can be edited for each individual operation.

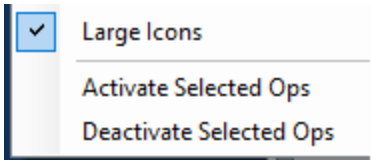
Active and Inactive Operations

In previous versions of GibbsCAM, posting or simulating Selected Ops could change 5-axis solution selection and other program properties unexpectedly. These functions have been replaced with a new op status for inactive operations, which are kept in the part file but do not participate in or influence the NC program. The Inactive Operation

feature can be used to work with parts as multiple programs in sequence, to experiment with alternative programming strategies, or for any other purpose where an op needs to be retained in the file but excluded from the program.

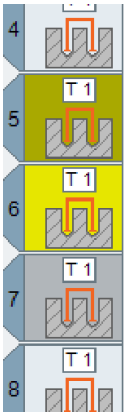
Op active/ Op inactive status persists when op selection is changed and even when the part is saved and reloaded. You can also view or set active/inactive status using Operation Manager.

To activate or deactivate ops, do one of the following:



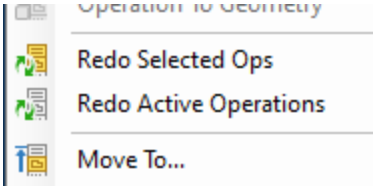
- Right-click an op tile selection. Context-menu choices now include Activate Selected Ops and Deactivate Selected Ops.
- Use Operation Manager to batch-activate/deactivate operations.
- Use Edit > Operation and click Activate All Ops or Deactivate Unselected Ops.
- Use Plug-Ins > Main; the Find Operations dialog box has a Look In checkbox for Inactive Ops.

Colors. Op tiles for inactive ops have a dark gray background when unselected, dark yellow when selected. Toolpath for inactive ops is drawn in gray rather than orange (when using default colors). Note that inter-operation connection moves are not drawn for inactive ops, since they are not participating in the program.



Affected functions. Inactive ops do not participate in any of the following GibbsCAM functions: Post Processing, Simulation, Material State Calculation (for Material Only or Start at Op), Program Error Checking, Sync Control, WFO assignment, Reporter output, and Multipart (TMS and Part Instancing).

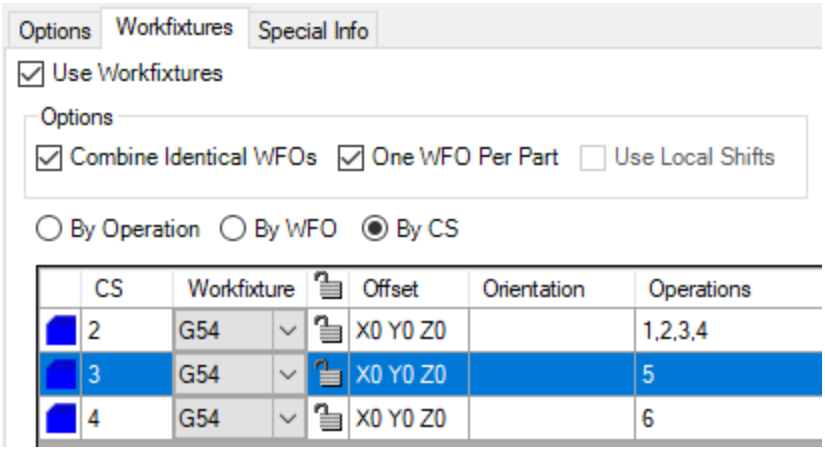
Redo and **Op Modifiers.** If you perform a manual Redo on an inactive op, or if you add an op modifier to it, then its status changes to Activated. However, **batch** Redo actions skip inactive ops by default, and their names have been changed to Redo Active Ops and Redo Selected Active Ops.



Simulation. The Skip Unselected Ops function now preserves material state. This is equivalent to setting multiple Start at Op triggers. Selected ops will play in detail; unselected ops will have material removed immediately, and playback will skip ahead to the next selected op, so selected ops are always played in the correct program and material context. If you want to use the pre-GC2023 functionality, which skipped material removal for unselected ops, simply deactivate the ops instead of deselecting them, and the program will simulate as if they were not present.

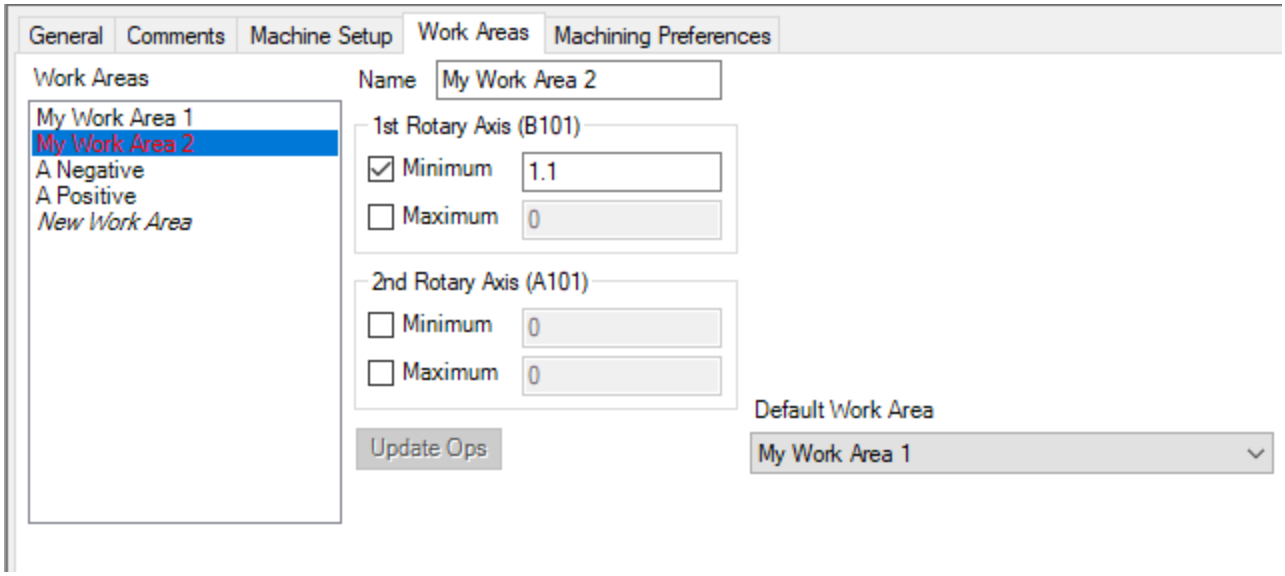
WFO GUI (Post Processor dialog, Workfixtures tab): By CS

The **Workfixtures** tab of the **Post Processor** dialog now provides a third view, By CS, which shows operations and WFOs grouped by CS. If you prefer using one WFO per CS, you may find this helpful to better understand and manipulate your program.



Compared to other views. From the By CS view (unlike the By WFO view) multiple CSs can be assigned to the same WFO. The same status icons used in the By Operation view notify you when the assigned WFO involves an origin or angle shift from the CS. If operations programmed in a single CS are assigned to multiple WFOs, that CS will show "(multiple)" in the WFO column, indicating you should instead use the By Operation view to better understand or change this WFO assignment.

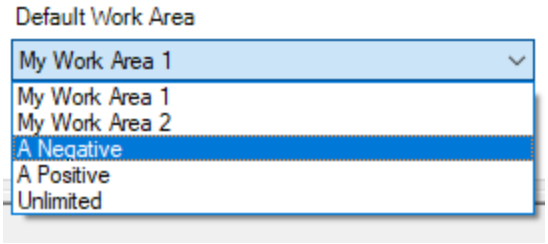
DCD: Work Areas



When a machine has two or more rotary axes, the Document Control Dialog (DCD)'s upper half has a new tab, **Work Areas**. This lets you view work areas defined in your MDD and, for each part, you can create work areas with user-specifiable values for such parameters as minimum and maximum angles for rotary axes. If you specify a default Work Area, it is used for all newly created operations.

Effect on Operations. If you change a part-defined Work Area, the Update Ops button becomes available, allowing you to update any operations that were assigned to that Work Area before your changes. If multiple identical Work Areas are defined, all but the first display red to inform you that these Work Areas cannot be matched.

In parts that have Work Areas defined, the **Op Data** dialog provides a Default Work Area dropdown.



Milling

Cylindrical Milling from Cylindrical Profiles

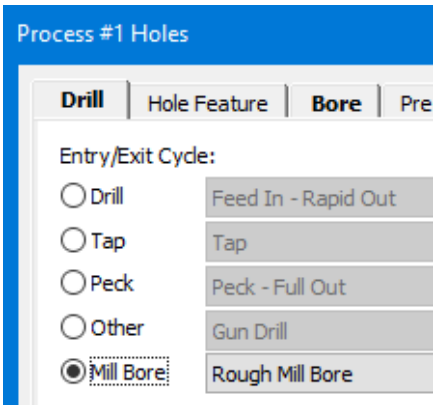
Using the Profiler, you can select profiles in Slice Cylinder mode and use them in Cylindrical Milling. Remember that Slice Cylinder mode aligns the slice cylinder around the CS depth axis, so you will need to activate an appropriate CS in your workspace for that use (such as the XY plane on a C-axis mill/turn), and select a different CS as your machining CS in the process (such as the YZ plane on a C-axis mill/turn). If you work with these operations frequently, consider using **File > Preferences > Interface Preferences** to uncheck Activate Machining CS on Op Load.

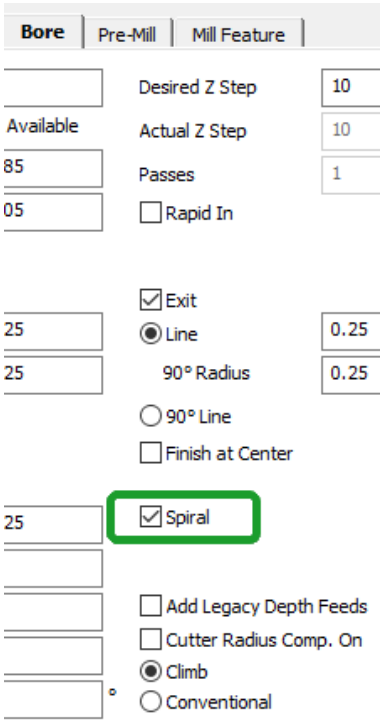
- If you select a closed wrapped profile, it is machined as a cylindrical boss or pocket, with the cutting side determined automatically by the solid that is profiled. This is like extracting the profile as geometry, unwrapping it, and machining the unwrapped geometry.
- If you select two loops that circumscribe the part, you can automatically machine a groove or ring. This is like creating wall geometry with the wrapped length of the groove or ring, with air walls at both ends. Note that some machining strategies, such as Ramp Down Contouring, will automatically extend the unwrapped groove in order to “wind up” and avoid unnecessary retracts.

Toolpath generated from this mode, like all Cylindrical Milling toolpath, is true analytic wrapped toolpath. Because it contains only lines and arcs, it can be output in cylindrical interpolation mode for your NC control. It is not gouge-checked against the solid, and always holds the tool on-center; you must use an appropriately-sized tool to avoid gouges in radial toolpath. If, instead, you prefer close fidelity to the solid over efficient NC code, you might want to use the 5-Axis **Rotary** strategy to hold the tool off-center, at the cost of longer and more complex segmented linear toolpath.

Mill Bore

A new option in the Mill **Holes**: Rough Mill Bore strategy allows Spiral boring. This option cuts a spiral out to full diameter at each Z step, in contrast to Helix Bore which helixes down in Z before stepping out to the diameter. Spiral boring produces a more consistent tool load without full engagement, which lets you substantially increase your cutting feedrate, cut depth, or both, to clear the bore more quickly.

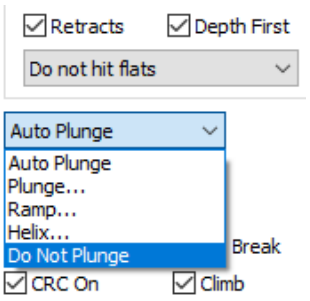




Transitions between depths for Rough Mill Bore and Finish Mill Bore are improved, eliminating and shortening redundant depth moves. This more-efficient toolpath is enabled by default; to disable it, check Add Legacy Depth Feeds on the **Bore** tab, to allow subroutine output on older mill post processors without a post modification.

Mill Roughing: Do Not Plunge

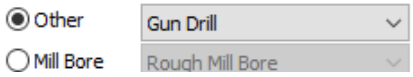
A new option in Mill **Roughing** strategies Offset and Offset With Cleanup lets you omit “cavity” cutting areas. When the entry style is set to Do Not Plunge, the tool enters the part only horizontally from air; enclosed pockets are not machined. This is useful for certain types of high-feed mill tools that are incapable of plunging or ramping. Remaining enclosed areas can be machined with a tool capable of ramping in a followup process with Material Only enabled.

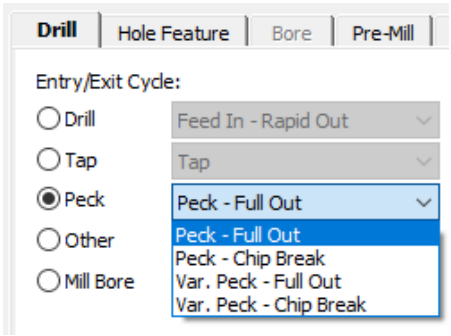


Extended Drill Cycles – Holes

Default drill cycle support is extended to the following cycle types:

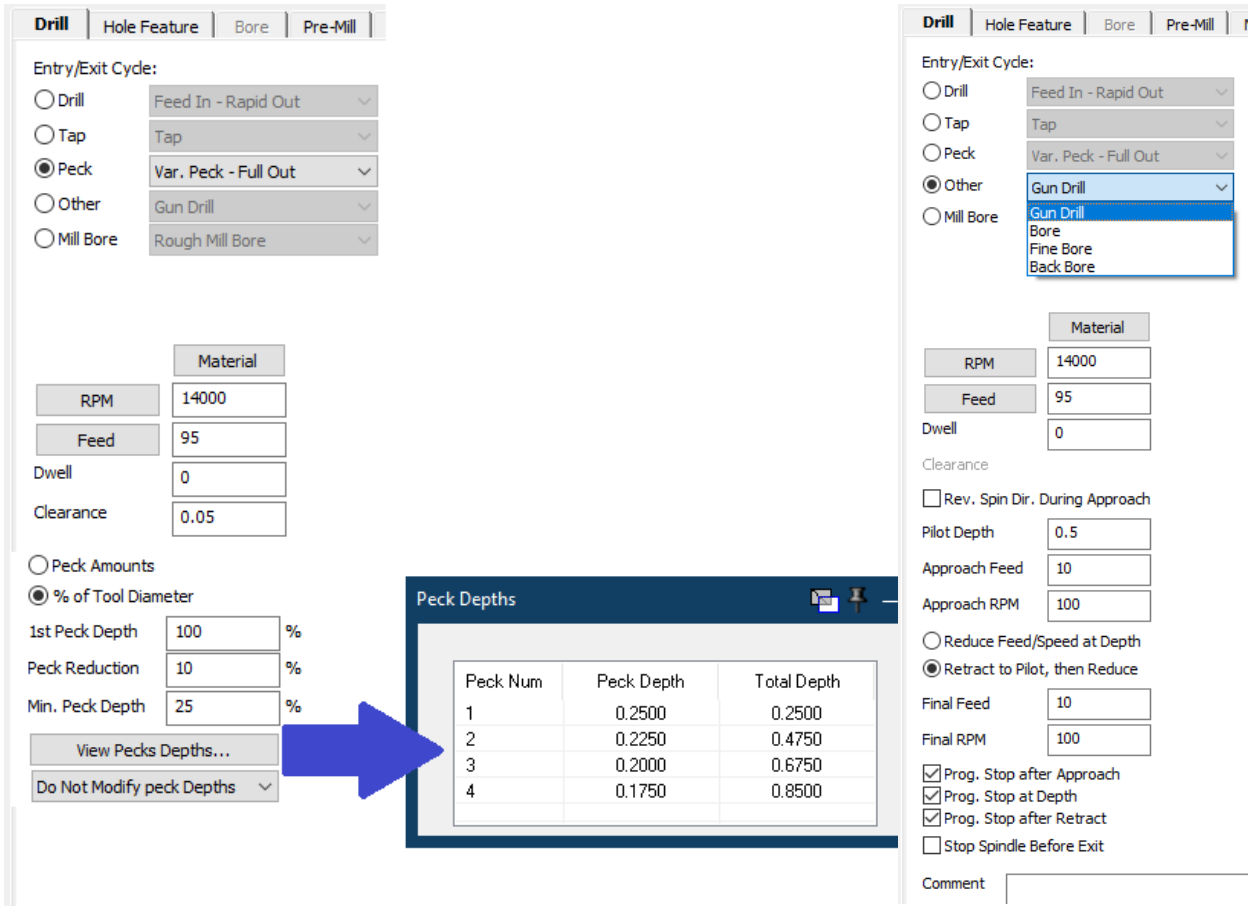
- Peck Tap – Full Out
- Peck Tap – Chip Breaker
- Variable Peck – Full Out
- Variable Peck – Chip Breaker
- Gun Drilling





For Variable Peck cycles, the **Peck Depths** dialog gives you fine control over the toolpath you want.

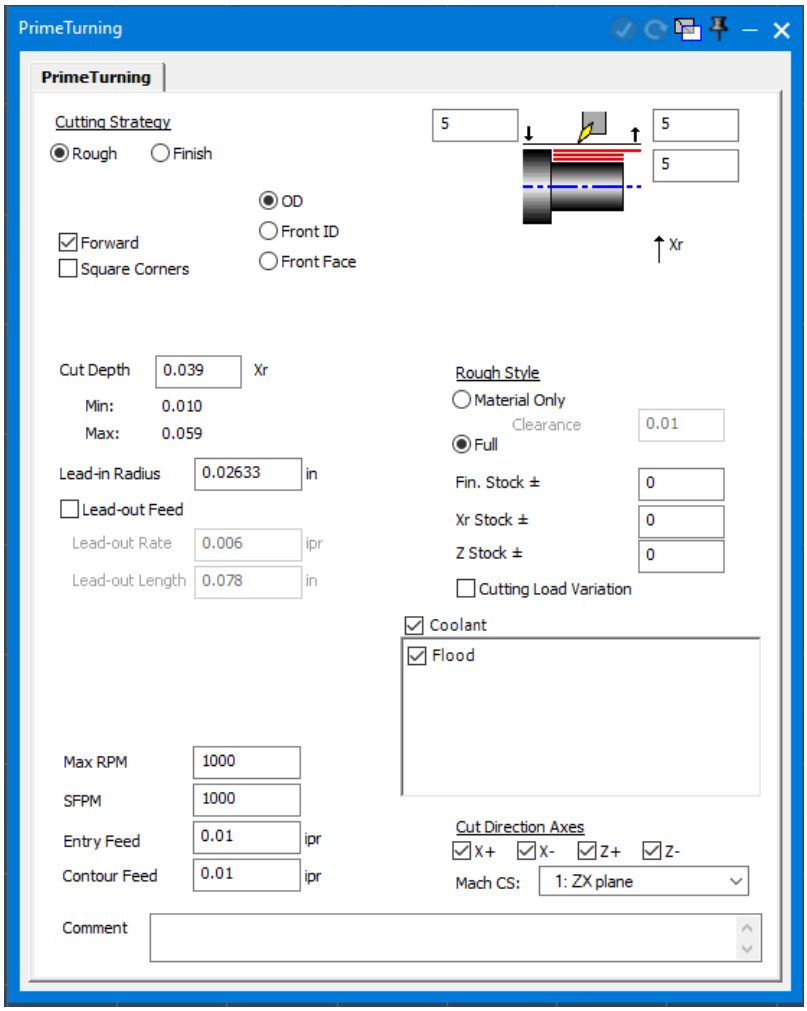
Please Note:
This feature may require a post modification if you require canned cycle output for your NC control. Or, if you are already using macro cycles for any of these types, you can continue to do so without problems.


Drill
Hole Feature
Bore
Pre-Mill
Entry/Exit Cycle:
Drill
Feed In - Rapid Out
Tap
Tap
Peck
Var. Peck - Full Out
Other
Gun Drill
Mill Bore
Rough Mill Bore
Material
RPM
14000
Feed
95
Dwell
0
Clearance
0.05
Peck Amounts
% of Tool Diameter
1st Peck Depth
100
%
Peck Reduction
10
%
Min. Peck Depth
25
%
View Pecks Depths...
Do Not Modify peck Depths
Peck Depths
Peck Num
Peck Depth
Total Depth
1
0.2500
0.2500
2
0.2250
0.4750
3
0.2000
0.6750
4
0.1750
0.8500
Drill
Hole Feature
Bore
Pre-Mill
Entry/Exit Cycle:
Drill
Feed In - Rapid Out
Tap
Tap
Peck
Var. Peck - Full Out
Other
Gun Drill
Mill Bore
Gun Drill
Bore
Fine Bore
Back Bore
Material
RPM
14000
Feed
95
Dwell
0
Clearance
Rev. Spin Dir. During Approach
Pilot Depth
0.5
Approach Feed
10
Approach RPM
100
Reduce Feed/Speed at Depth
Retract to Pilot, then Reduce
Final Feed
10
Final RPM
100
Prog. Stop after Approach
Prog. Stop at Depth
Prog. Stop after Retract
Stop Spindle Before Exit
Comment

General Turning

PrimeTurning

GibbsCAM 2023 supports Sandvik Coromant's PrimeTurning high-performance turning strategy and the CoroTurnPrime Type A and Type B inserts. Roughing and Finishing are both supported through the new **PrimeTurning** process. This strategy enters the part gently, and can cut either direction with the tool, automatically adjusting the feedrate as appropriate to maintain correct chip thickness. **PrimeTurning** in GibbsCAM works with all the usual GibbsCAM turning features, including Auto Clearance and Material Only.



The Minimum Cut Depth and Maximum Cut Depth for your selected CoroTurn Prime insert is shown on the process dialog for convenience. Select your desired Cut Depth, Lead-in Radius (recommended value is the Cut Depth), and how much to slow the tool during the

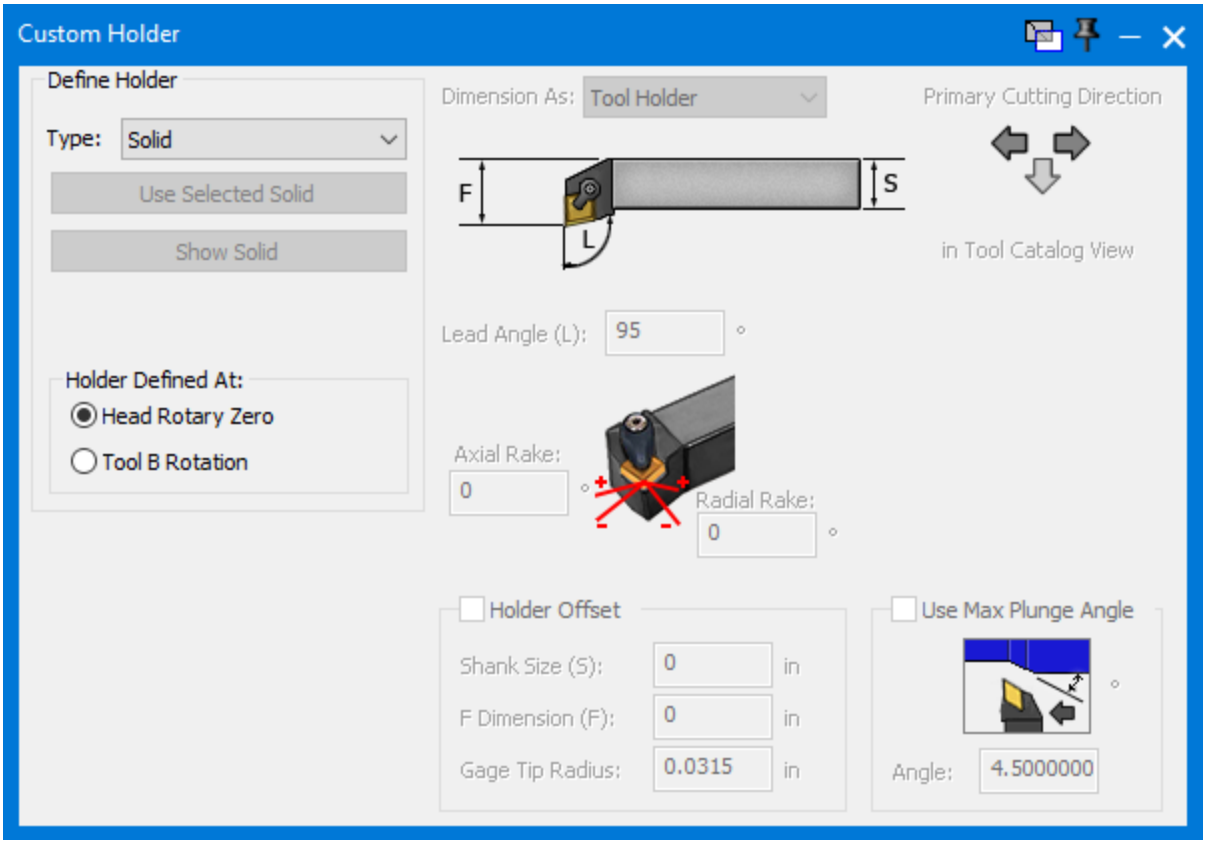
Lead-out. See your local Sandvik Coromant representative for additional advice about how to optimally configure this process.

Multifunction Insert Drill Turning and Offset Drilling

A new tool type is added: Multifunction Insert Drills. This tool type represents inserted drills that have a periphery insert that can perform turning and boring, such as the Sandvik Coromant CoroDrill 880 series. The tool body and periphery insert are defined in a single tool tile, allowing GibbsCAM to seamlessly switch between turning, lathe drilling, and mill drilling operations without extra retract and toolchange motion. Two tool offsets must be defined. The length offset programs the center of tool body (like a conventional drill), while the periphery offset programs the corner of the periphery insert (like a conventional turning tool).
A new option in Lathe Holes allows the use of MFI Drill tools for radial-offset drilling. Enter an offset amount between 0 and the tool radius to create a hole that is slightly larger than your tool. You can also select whether to output toolpath with the length offset at the center of the tool (typical for drilling) or with the periphery offset at the periphery insert corner, which may provide better dimensional control.

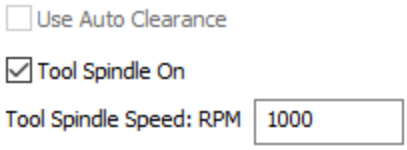
Turning Tool Rake and Back Relief

New fields have been added to turning tools to more accurately define the 3D geometry of the tool. Most types of turning inserts now have a Back Relief field to specify the insert's back relief in degrees. Additionally, Custom and Other toolholders can define that the insert is held with axial and radial rake (holding the insert slightly tilted out of the turning plane). Note that insert rake is used for visualization only and does not affect toolpath generation or the cutting plane used in the process. For tools that are used to cut in a different plane, such as Sandvik CoroCut QD Y-axis parting tools, you should instead use Intermediate Tooling to establish the new cutting plane.



Lathe Drilling With Counter-Rotating Live Tool

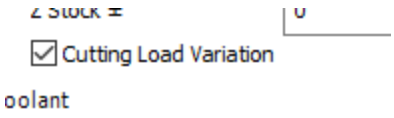
A new option in Lathe Drilling, Tool Spindle On, automatically commands the live tooling spindle to be spinning the opposite direction from the lathe spindle with a specified RPM. This increases the effective spindle speed and can significantly boost material removal rates, especially when performing simultaneous centerline drilling and OD turning with different toolgroups.



Please Note: This feature may require a post upgrade.

Cutting Load Variation

Most Turning processes now offer Cutting Load Variation functions. The exact nature of this function depends on your NC control and postprocessor, but it usually involves regular oscillation of either the cutting feedrate or spindle speed to suppress resonance-induced chatter and improve chipbreaking.

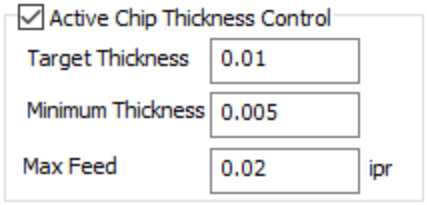


CNC manufacturers that offer options to vary the spindle speed include Haas and Soraluce (Spindle Speed Variation or SSV), Okuma (Harmonic Spindle Speed Control, HSSC, and Variable Spindle Speed Threading, VSST), and DMGMori (Alternating Speed).
CNC manufacturers that offer options to oscillate the feed axis include Star (High Frequency Turning or HFT), Citizen and Miyano (Low Frequency Vibration or LFV), and Tsugami (Oscillation Cutting).

Please Note: This feature requires a post upgrade.

VoluTurn: Active Chip Thickness Control

The VoluTurn process dialog offers Active Chip Thickness Control. ACTC allows VoluTurn to control the chip thickness during the cut by varying the feedrate as the instantaneous cut depth changes. Specify a target thickness, minimum chip thickness, and the maximum feedrate to be used when the instantaneous cut depth is very small.



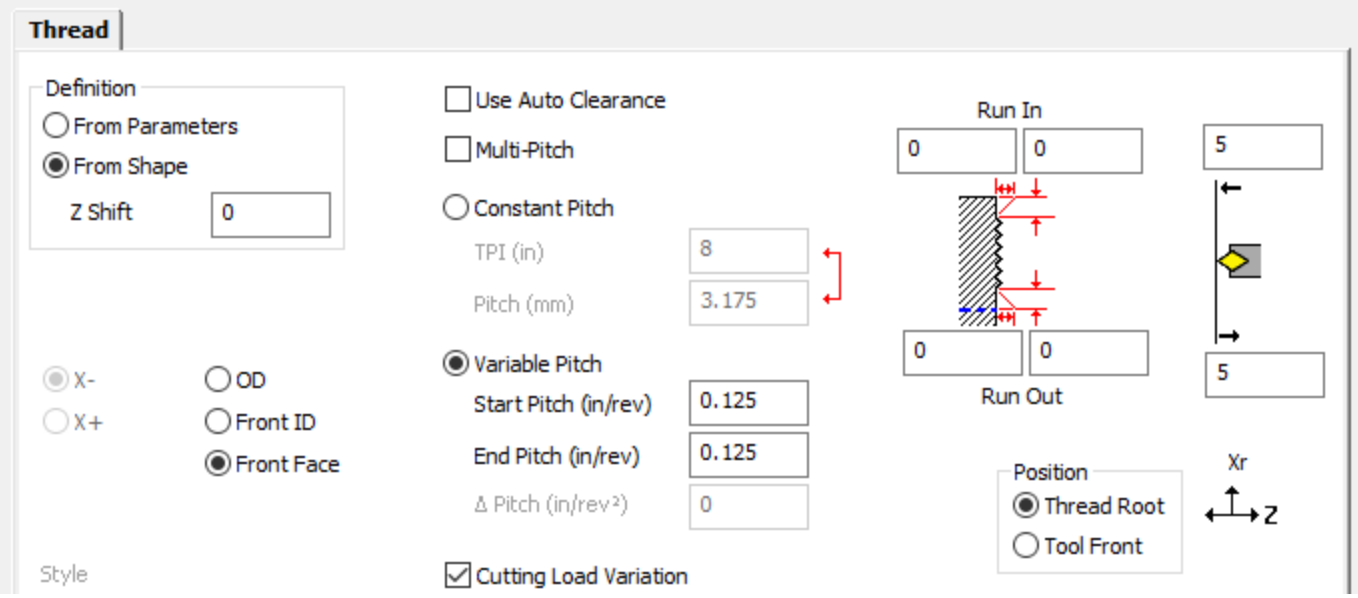
Elliptical Turning: Inside Diameter, and Xr Stock Offset

The Elliptical Contour process dialog offers the ID approach type to let you cut elliptical bores inside solid parts. You must select the surface to be machined, as well as a spine curve which runs within the bore. Creating a smooth spine curve close to the center of the bore will produce better quality toolpath.
Elliptical Contouring also now supports a radial (Xr) stock offset, for both ID and OD cutting.

Thread Turning

Face Threading

The Thread process dialog offers the Front Face approach type, for face threading or scroll threading. This type of machining produces a spiral thread on the face of the part.



Please Note: This feature may require a post modification if you require canned cycle output for your NC control.

Variable Pitch Threading

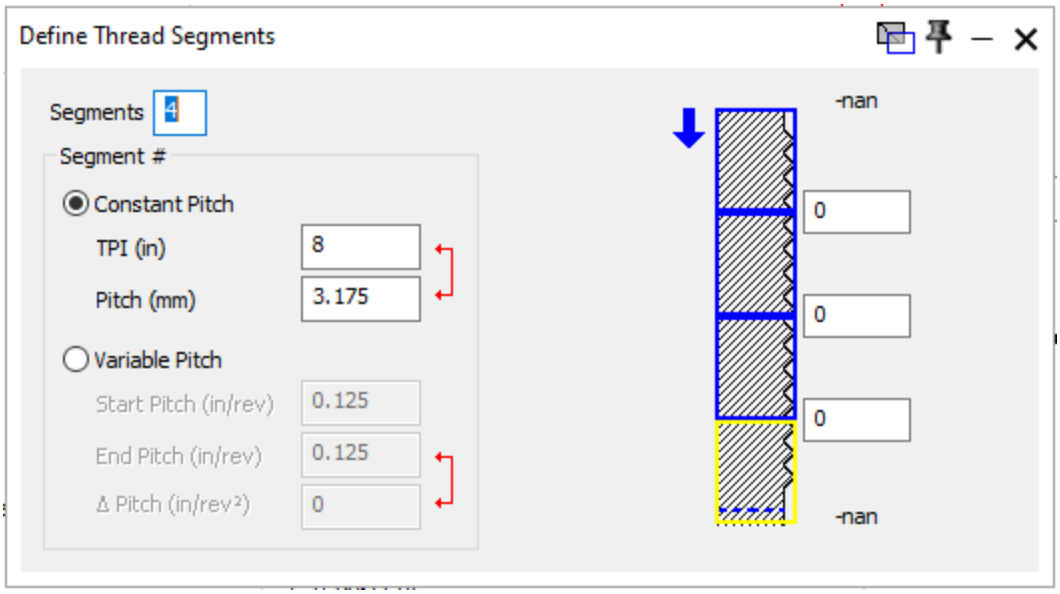
Select the Variable Pitch radio button in the Thread process to produce variable pitch threads. Specify values for Start Pitch and End Pitch to blend between them, or for Start Pitch and Δ Pitch to increment the pitch by the specified amount every revolution.

Please Note: This feature may require a post modification if you require canned cycle output for your NC control.

Multi-Pitch Threading

Check the Multi-Pitch checkbox to activate Multi-Pitch Threading. Click Define Segments to open the **Thread Segments** dialog where you specify the segments of your thread (in Z for OD/ID threading, or X for Face Threading). Enter the segment count and the boundaries between each segment. Selecting a segment lets you define the pitch for

that segment (either constant or variable pitch). Multiple thread features will be output along your thread with the specified pitch changes, to produce a continuous thread with different pitches.



Please Note: This feature may require a post modification if you require canned cycle output for your NC control.

Position Tool Front

The Position group now allows the selection of either Thread Root or Tool Front to define the values in the thread diagram. Thread Root means that the numbers define the thread itself; the Start Z value, for example, is where the thread starts on the part. Tool Front means that the numbers define the position of the front of the tool, so Start Z will be where the tool is when the cutting starts. This setting can be useful cutting threads up to a square shoulder, when the precise thread length is less important than the shoulder location. With a Laydown-style (LT) thread insert, selecting Position Tool Front and also selecting the alternate touchoff point at the front of the tool in the tool dialog will typically cause the exact start/end numbers that you type in this dialog to be output in the G-code.

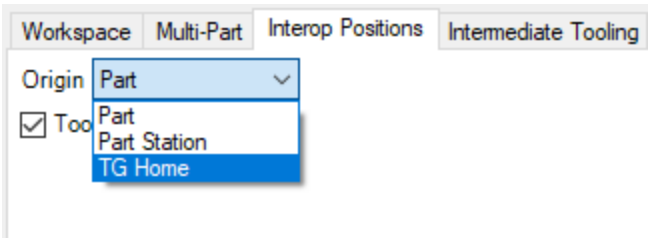
Other Enhancements and Improvements

Non-Z-Aligned Cylindrical Stock and Clearances

New options are enabled when creating an MDD in Machine Manager. Non-spinning part stations using cylindrical stock and Clearance Volumes enabled can now choose a Face Held of X, Y, or Z, minimum or maximum, instead of being restricted to Z minimum. This allows the use of Cylindrical or Part-defined stock types on 4-axis vertical mills while retaining the correct Z orientation.

User Toolchanges From Alternate Origins

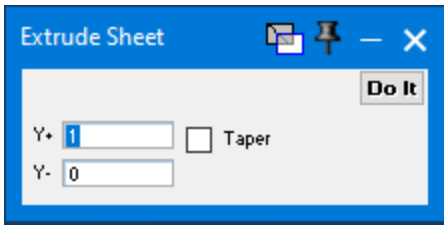
On machines supporting User Toolchanges, you can now select the origin for the toolchange on the **Interop Positions** tab of the DCD.



- Part is default behavior from before GC2023. It defines the toolchange location relative to the part origin. This is useful in many cases, such as close turret rotation on a typical lathe setup.
- Part Station defines toolchange location relative to the part station origin without the part offset.
- Machine (not available for Generic MDDs) defines toolchange location relative to the machine root, which lets toolchange occur in the same machine location regardless of the part setup.
- TG Home defines the toolchange location relative to the toolgroup home location. Note that selecting an alternate origin changes the meaning of the input coordinates but does not affect output by default. The postprocessor may also choose to change the output mode based on your origin selection; this will require a post modification.

Tapered Surface Extrusion

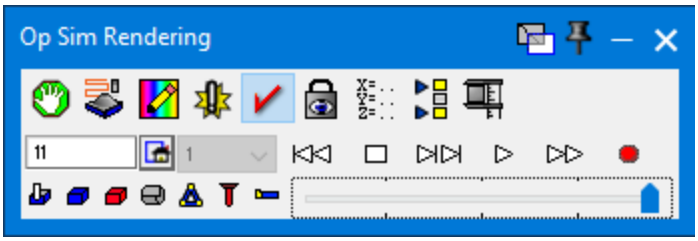
GibbsCAM 2022 introduced a new surface modeling command, Extrude. GibbsCAM 2023 extends this command with a Taper field, as in the equivalent solid modeling command.
Tapered solid extrusions are also more robust and efficient in this release



.

Fixture Visibility Toggle in Simulation

All simulation modes now include an extended Fixture Visibility toggle. Rather than only being able to set fixtures to opaque ( ) or partially transparent ( ), you can now also make them invisible ( ).



High-Feed Collision Reporting in Simulation

All simulation modes include a new Collision/Program Error setting: Cuts above ___ mmpm/ipm are collisions. By configuring this setting, you can tell simulation to report collisions if a process feeds into material at a feedrate higher than you believe is safe (for example, VoluMill and Advanced 3D both have High Feed modes). This can help you forsee and prevent broken tools and damaged parts, even when rapid is not being used.

Miscellaneous

Updated Support for Third-Party Libraries

All the libraries included in GibbsCAM 2023 have been updated to the latest version available to us at the time of integration.
GibbsCAM 2023 incorporates or is compatible with the following third-party libraries:

- ModuleWorks 2022.04
- VoluMill
- OPTICAM
- Tool libraries for Harvey Tools and Helical

CAD Add-Ins

GibbsCAM 2023 incorporates or is compatible with the following third-party Add-Ins:

- Cimatron 15
- Cimatron 16
- Autodesk Inventor 2010 and newer (64-bit)
- Solid Edge 100 (ST) and newer
- SOLIDWORKS 2011 and newer

Advanced Display

GibbsCAM 2023 leverages the capabilities of advanced video card drivers. Although it should be standard practice always to use the latest versions of all drivers and service packs, it is especially important to upgrade to the latest video card driver version when using GibbsCAM 2023 to assure the best display performance and functionality.

System Requirements

Minimum and recommended configurations for GibbsCAM 2023 are as follows:

	Minimum	*Recommended*
OS	Windows 10, Windows 11, or Windows Server 2019	
CPU	*Intel:* 5th-generation or newer (Core i3 or better) *AMD:* Desktop family 17h	*Intel:* Core i9, i7, or i5 with four or more cores *AMD:* Ryzen or Threadripper
RAM	4+ GB of total RAM	16 GB of total RAM
Video	3D-accelerated video card with 1+ GB of video memory	NVIDIA video card with 2+ GB of video memory
Disk	8+ GB free disk space to install the software	

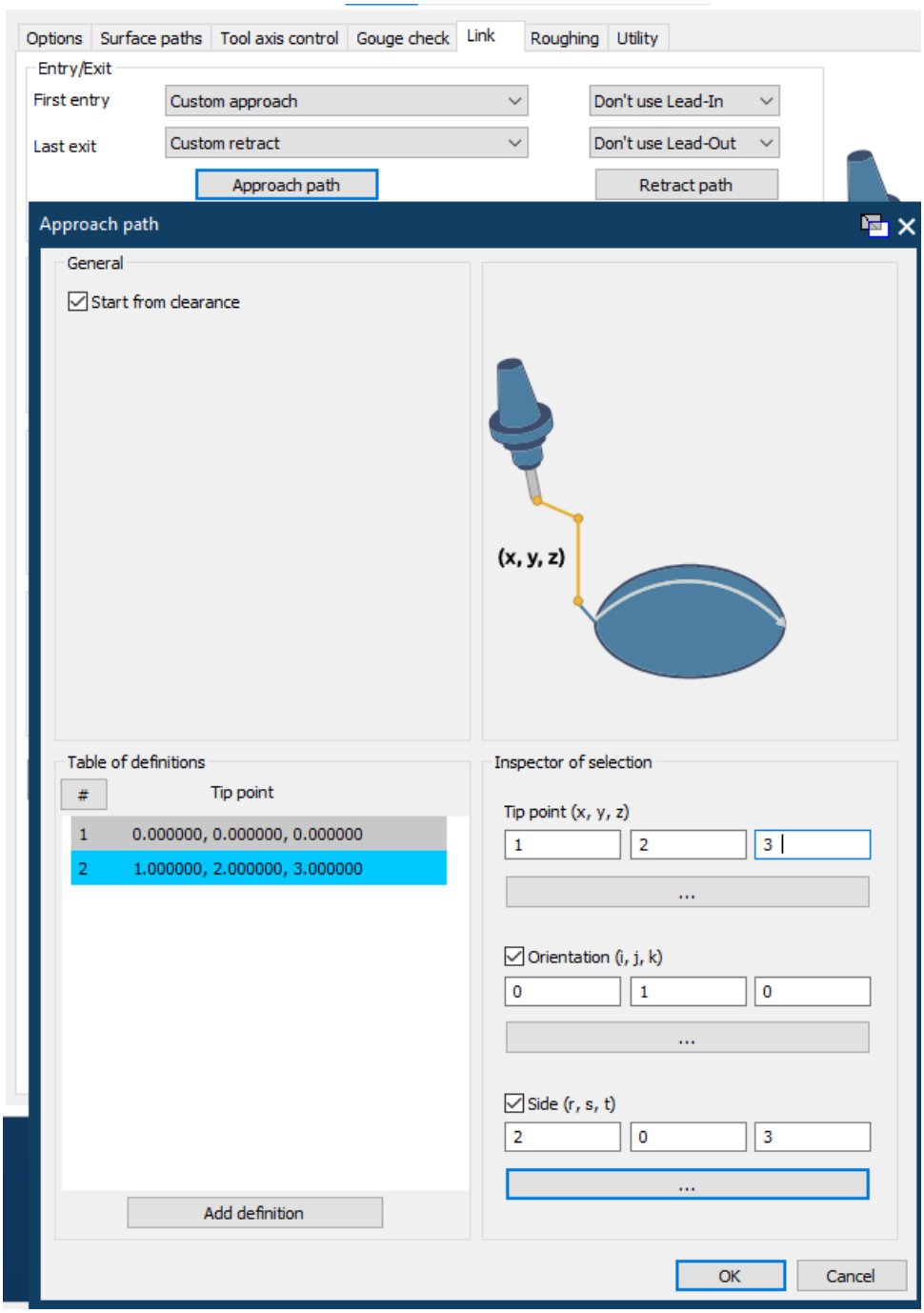
5-Axis

Enhancements and improvements since GibbsCAM 2022 include:

- **Surfaces, Wireframe, and Contouring:** Support for Custom approach / Custom retract
- **Surfaces > Flowline:** Flow direction
- **Multi blade machining:** Support for full collision check
- **Multi blade machining:** Parameters for First cut
- **Multi blade machining:** Separate selection of blades and fillets
- **Multiaxis machining:** Rest finishing
- **Multiaxis finishing:** Support for multiple surfaces
- **Multiaxis roughing:** Part definition of Fixture surfaces
- **Multiaxis roughing / Links > Retract:** Interpolation tilt angles
- **Rotary machining > Utility:** Feedrate for links
- **Rotary machining: Part definition >** Containment
- **Rotary machining:** Depth steps
- **Rotary machining:** Filtering out small regions
- **Deburring:** Limit edge detection area to a Mesh containment body
- **Deburring:** Spiralize closed contours
- **Wireframe**
  - Link tab > Retracts: Automatic clearance height
  - Area: Extend/Trim
  - Floor finishing > Sorting: Spiral cutting method
  - Engrave: Lead-In Type Vertical profile ramp
  - Engrave: Sorting options (Selection sequence, Left to right, etc.)
  - Engrave > Sorting: Start from options (Interior corner, etc.)
  - Engrave (Rough): Rest rough
  - 2-Axis Rough: Intermediate Slices
  - 2-Axis Rough: Draft Angle
- **Triangle Mesh**
  - Adaptive Roughing: Remove stock pillars
  - Adaptive Roughing: Over machine
  - Parallel cuts: Exclude flat areas
  - Constant Z Undercuts: Machine flatlands
  - Rough: Minimum depth step
  - Flatlands: Undercuts
  - Link tab / Retracts: Radial clearance
  - Roughing tab / Advanced: Shift distance (stepover %)

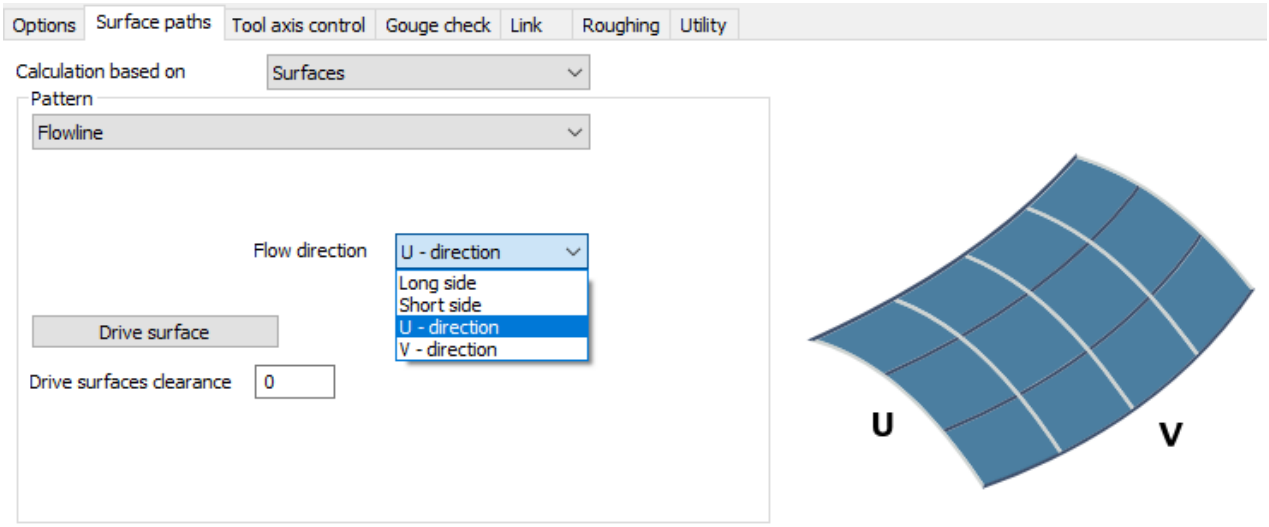
Link tab: Custom approach / Custom retract

For calculation strategies Surfaces, Wireframe, and Contouring, the **Link** tab now lets you manually define custom approach and/or retract links. For each custom entry/exit link, you can provide a list of *n* points and optionally, tool orientation and side vectors for each point. The approach and retract links follow the points and, from there, they connect to the contour. This overcomes a limitation where the regular retract would collide and could not be resolved automatically, and it allows for 6-axis re-orientation on the retract path.



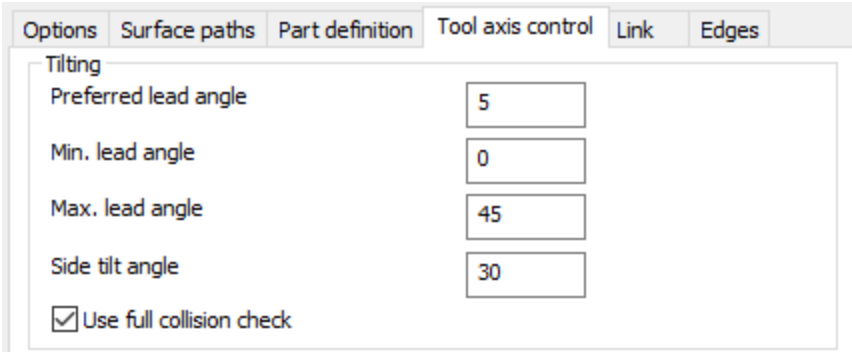
Surfaces > Flowline: Flow direction

For surface-based machining, pattern Flowline, new Flow direction choices let the pattern flow along the parametric dimensions (U or V) of the given surface. Benefits include following the natural shape of the surface, freedom from having to select bounding geometries or curves, and fast calculation time.



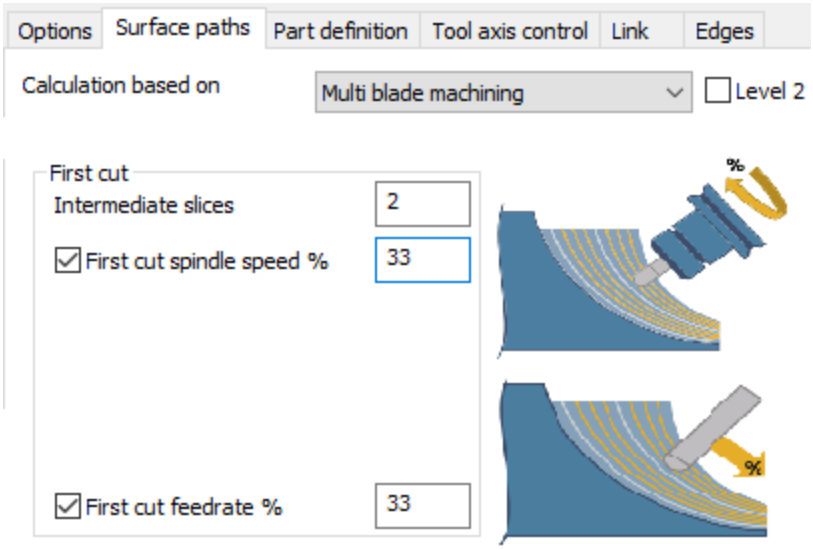
Multi blade machining: Support for full collision check

For Multi blade machining, the **Tool axis control** tab now offers checkbox Use full collision check for Roughing and Hub finishing.



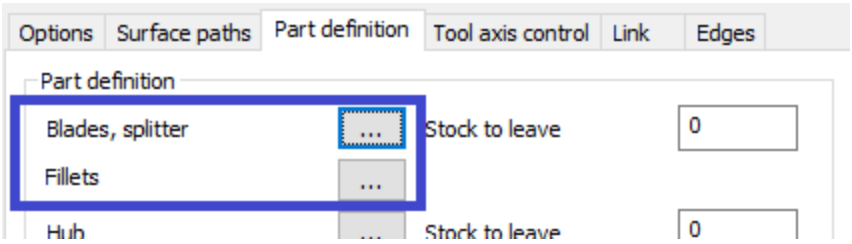
Multi blade machining: Parameters for First cut

In the Multi blade machining **Surface paths** tab, new parameters under First cut allow you to set a different spindle speed and/or feedrate for the first cuts in a new layer.



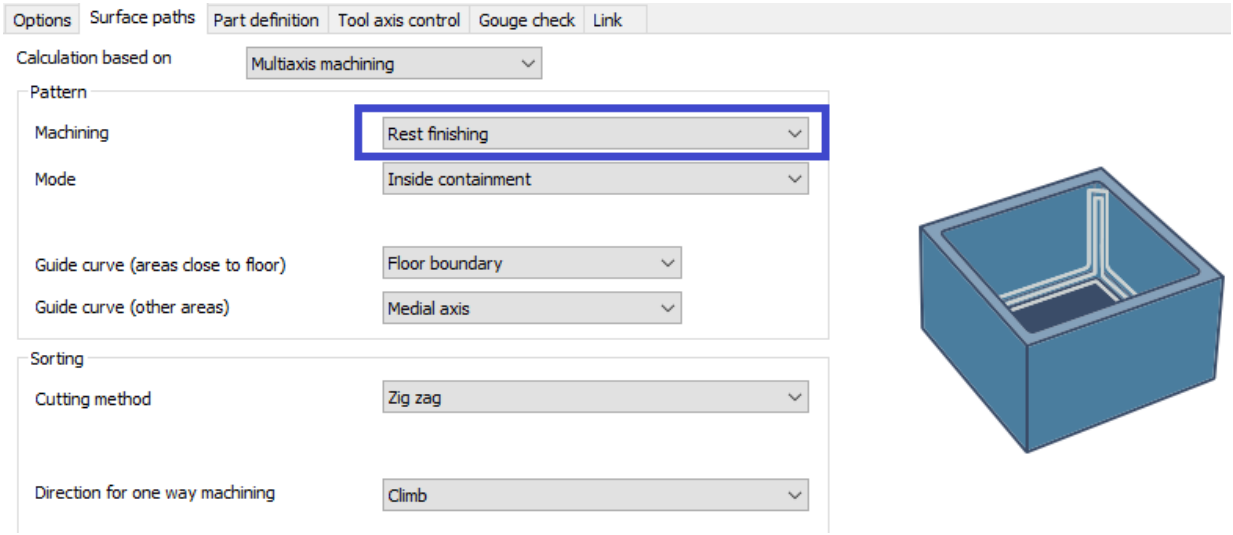
Multi blade machining: Separate selection of blades and fillets

For Multi blade machining, the **Part definition** tab now lets you select blades and fillets independently. This improves the quality of the toolpath by separating the given information, yielding a result closer to contour roughing with bull nose and flat tools.



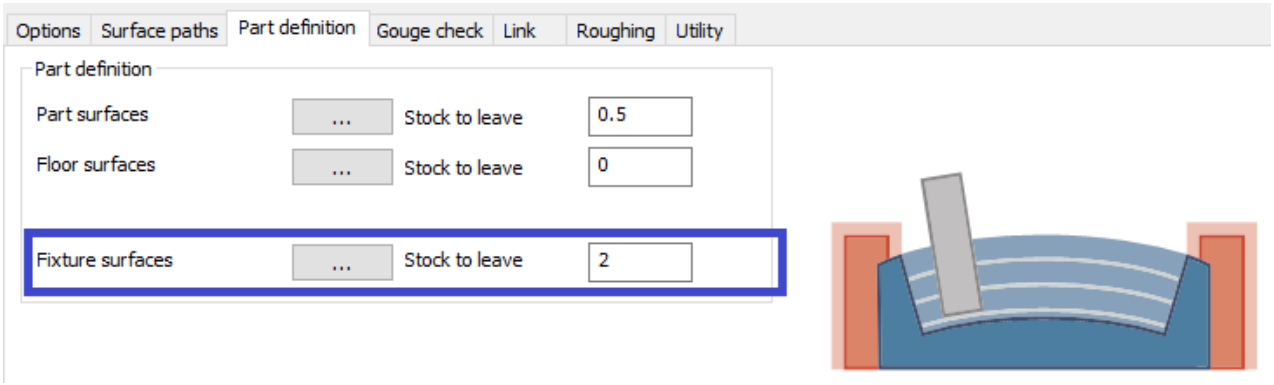
Multiaxis: Rest finishing

Multiaxis machining now offers a new machining pattern: Rest finishing. This new mode adds finish cuts along the containment curve. The containment can be the boundary around the entire machining surface (for example, wall or floor surface) or an automatic or user-defined containment around an unmachined area. These additional cuts remove scallops on the boundary of the infill pattern.



Multiaxis Roughing: Part definition of Fixture surfaces

When using the Roughing pattern of Multiaxis machining, the **Part definition** tab now lets you identify Fixture surfaces, so that toolpath calculation will avoid collisions with tools, clamps, jaws, etc.



Multiaxis Roughing / Links > Retracts: Interpolation tilt angles

When using the Roughing pattern of Multiaxis machining, new options in the **Retracts** dialog let you define discrete Angle step parameters that determine the angle that will be respected during tool transitions along the retract links. You decide the exact values to be used for link movements.

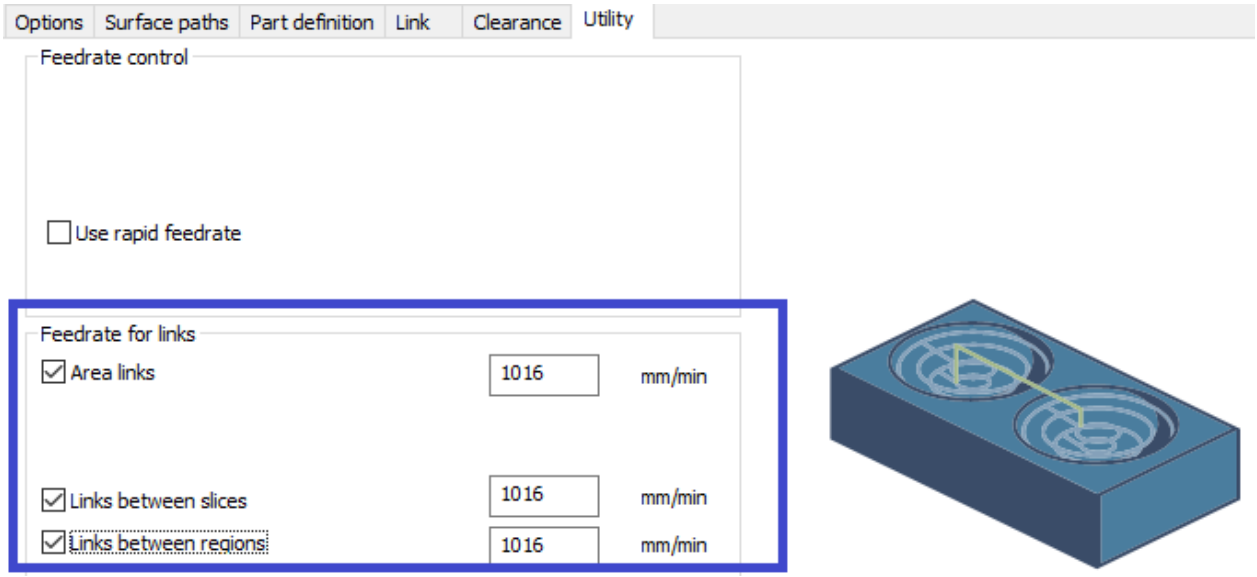
Multiaxis Finishing: Support for Multiple Surfaces

In all finishing patters of Multiaxis machining, you can now create toolpaths on multiple unconnected surfaces within one operation with the same tools and settings, increasing programming performance and reducing interoperation linking.

Rotary machining > Utility: Feedrate for links

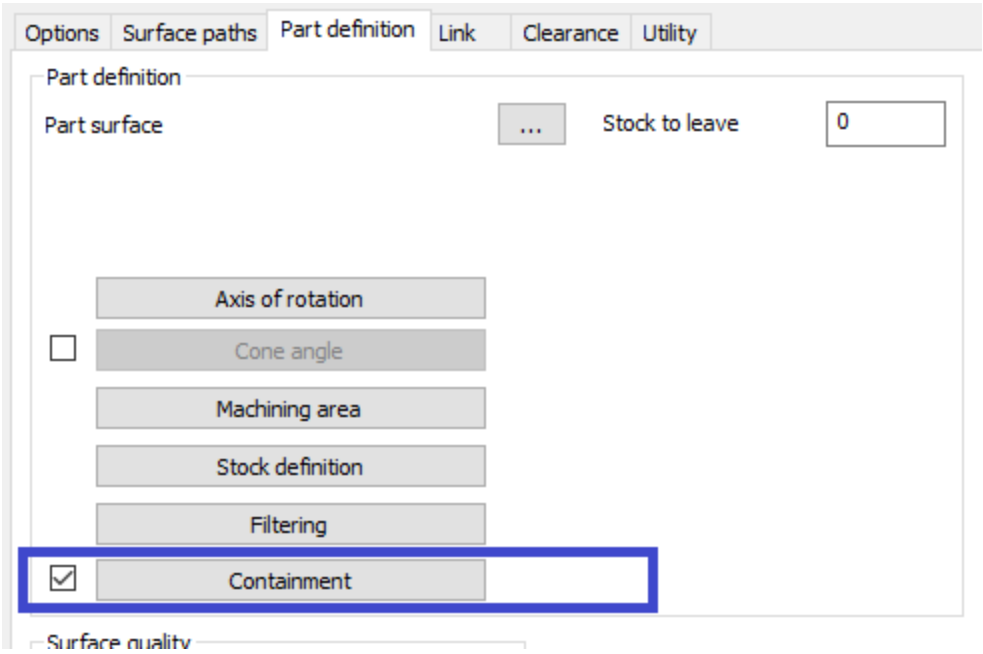
For Rotary machining, the **Utility** tab now offers new options for setting feedrate for different link movements. The first option lets you override the rapid feedrate with a user-defined feedrate. Other options let you define feedrates for direct or blend spline links. Benefits include:

- Instead of using the exact stop G00 movements, you can now override these with feedrates to create smooth machine movements.
- You have control over individual direct and blend spline links. Depending on the application, this can either reduce the cycle time or enable lower feedrates on links for slot cuts.



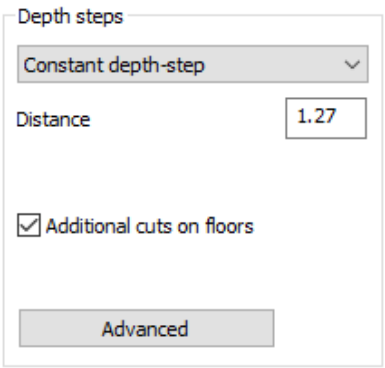
Rotary machining: Part definition > Containment

For Rotary machining, the **Part definition** tab now offers a new option: Containment. This lets you select 3D or 2D curves that define the containment area of the toolpath. You can add offset values with a positive or negative range, and you can invert the containment area.



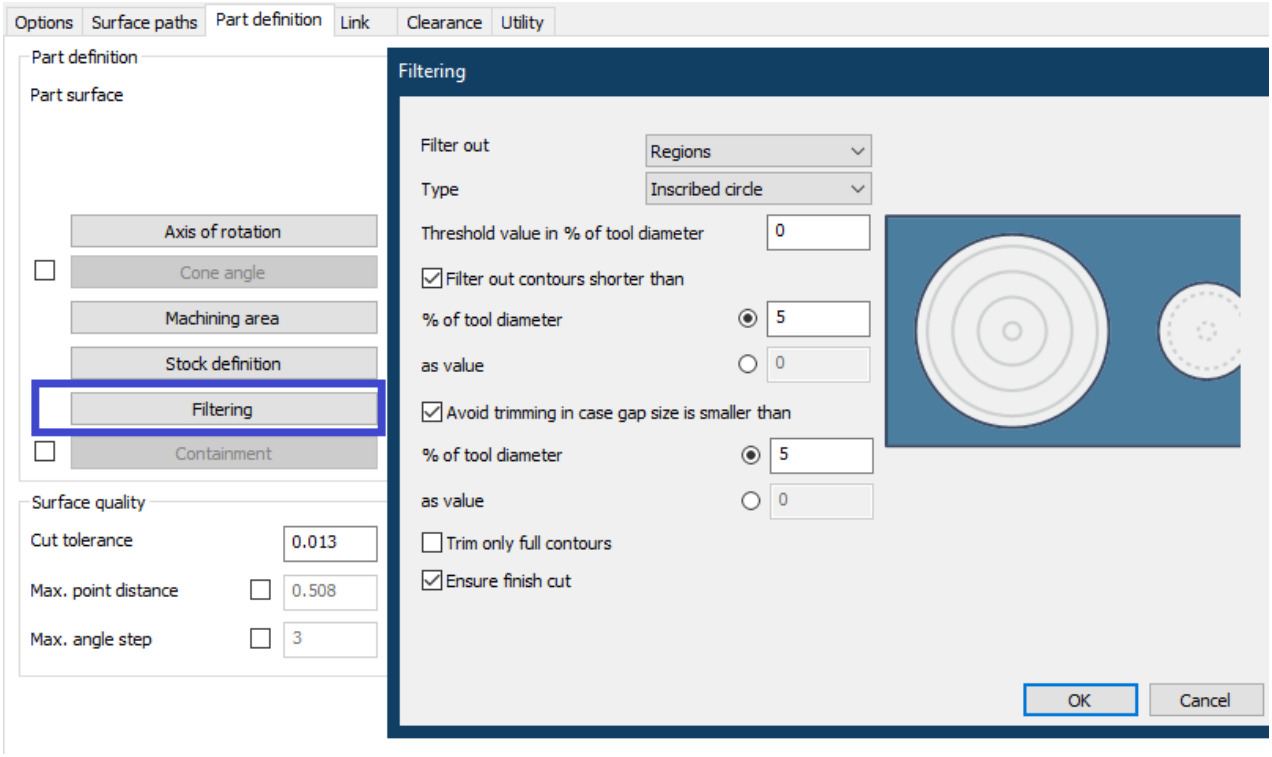
Rotary machining: Depth steps

For Rotary machining, the **Surface paths** tab offers options under Depth steps: Constant depth-step and Adaptive depth-step. These let you use additional cuts in combination with a constant depth step.



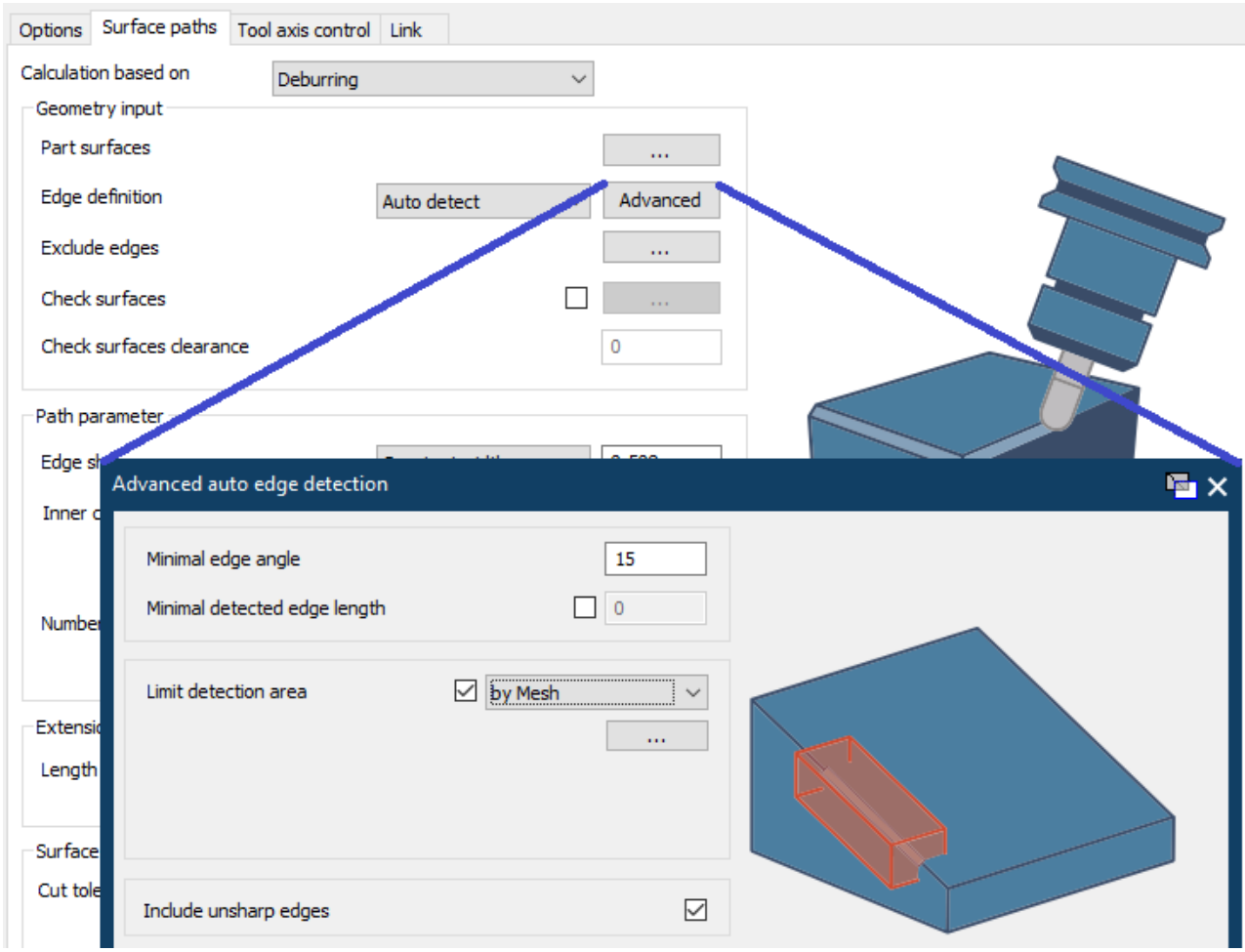
Rotary machining: Filtering out small regions

For Rotary machining, in the Part definition tab, the new **Filtering** dialog let you filter out toolpath regions that are smaller then the threshold values you specify.



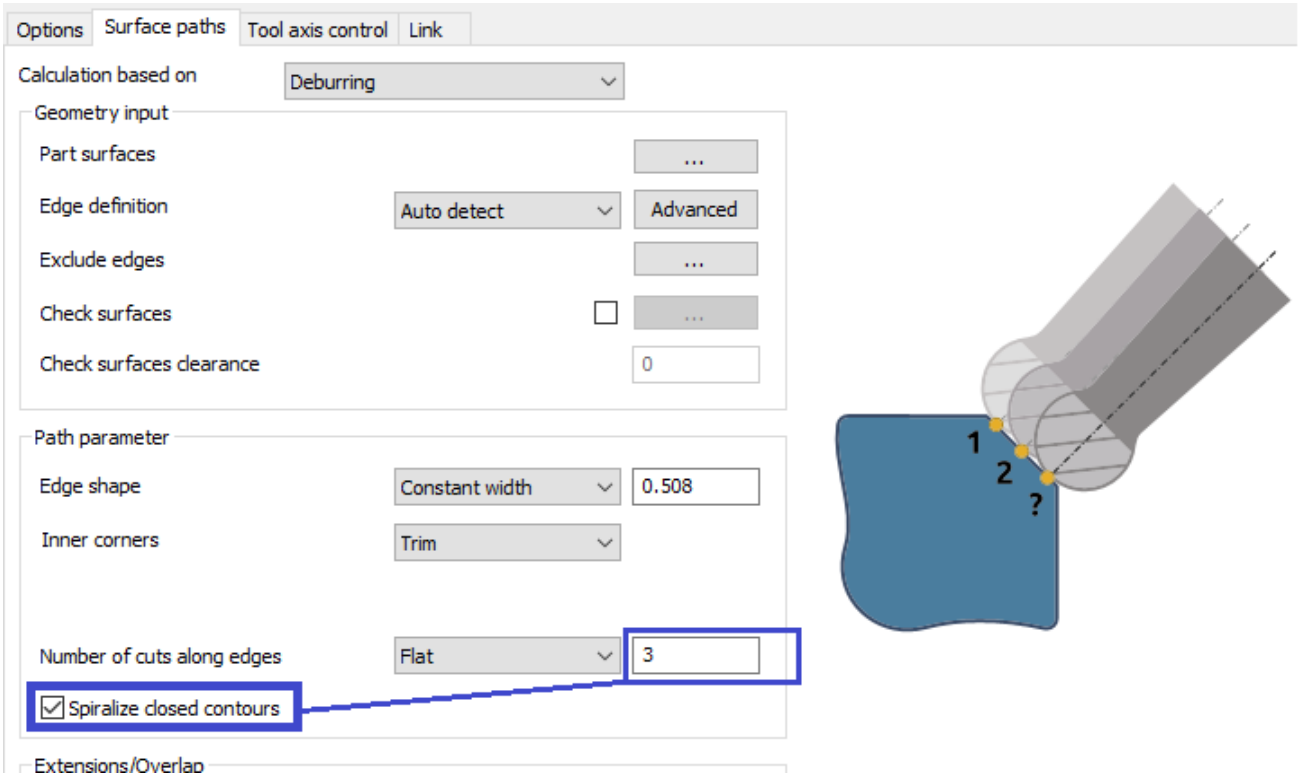
Deburring: Limit edge detection area to Mesh containment bodies

In Deburring > Surface paths > Edge definition, the **Advanced auto edge detection** dialog now lets you select one or more mesh containment bodies, where each containment body is a closed mesh. The only edges that will be machined are those inside the selected containment body or bodies. This feature makes it easier to select the areas to be machined, and it speeds up programming.



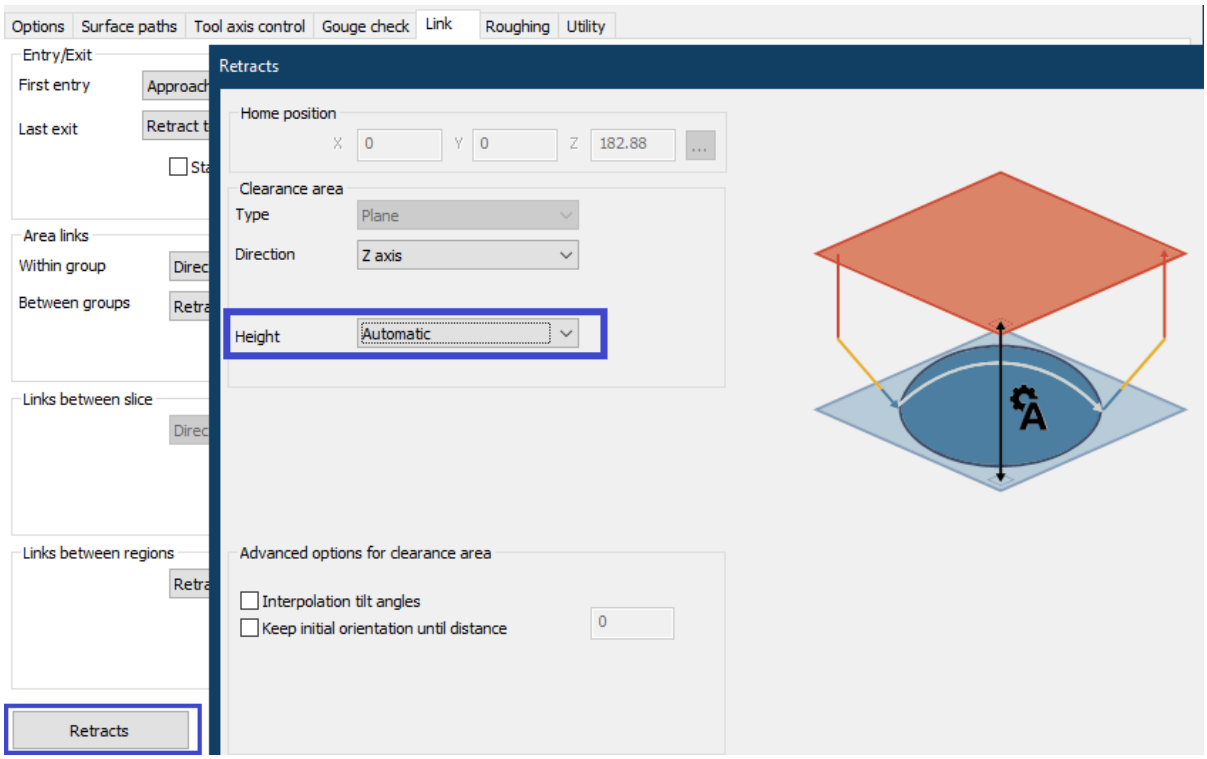
Deburring: Spiralize closed contours

In Deburring > **Surface paths** > Path parameter, when Number of cuts along edges is >1, new option Spiralize closed contours allows multiple cuts to use a spiral pattern on closed contours. This avoids stepover marks on the deburred edge, resulting in smoother toolpath and less machine movement.


Options
Surface paths
Tool axis control
Link
Calculation based on
Deburring
Geometry input
Part surfaces
...
Edge definition
Auto detect
Advanced
Exclude edges
...
Check surfaces
...
Check surfaces clearance
0
Path parameter
Edge shape
Constant width
0.508
Inner corners
Trim
Number of cuts along edges
Flat
3
Spiralize closed contours
Extensions/Overlap
1
2
?

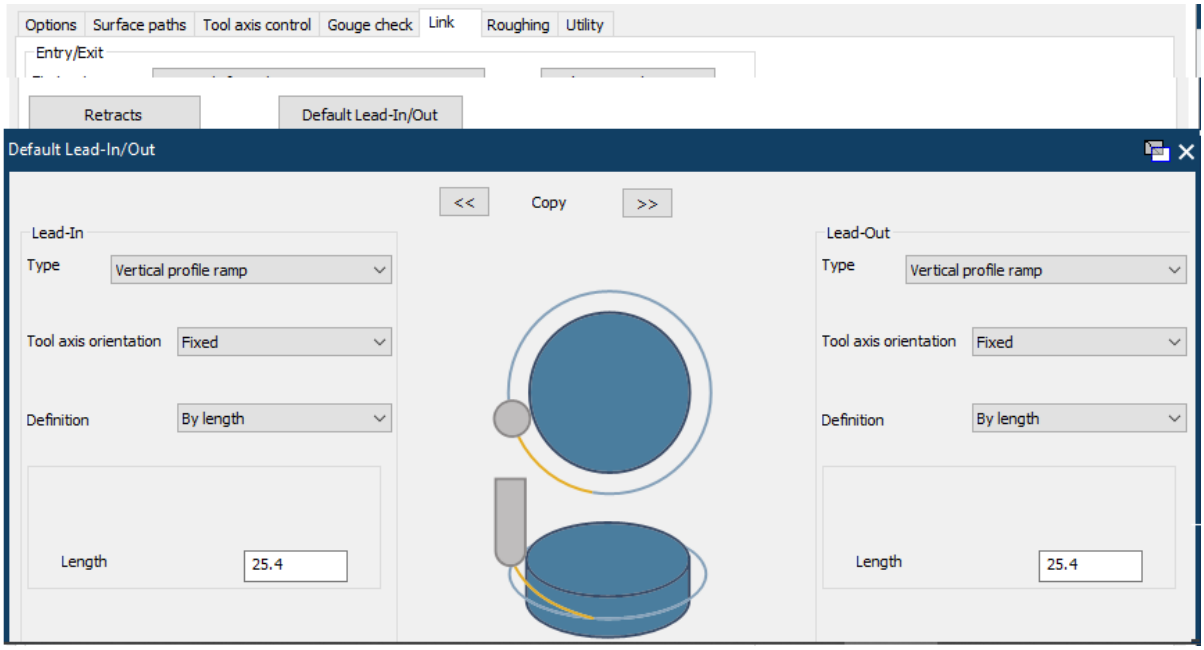
Wireframe > Link tab > Retracts: Automatic clearance height

For all Wireframe patterns, the **Retracts** dialog now offers a new Automatic choice for Height, to automatically define clearance plane height according to the range of the specified machining height and the spatial orientation of the given curves. This makes it easier to control the clearance area.



Wireframe Engrave: Lead-In Type Vertical profile ramp

For Wireframe pattern Engrave, the **Link** tab offers a new Lead-In type: Vertical profile ramp. This lets you define a vertical profile ramp entry in the material that follows the shape of first toolpath contour until it reaches a specified length and height, making it easier to control the approach motions.

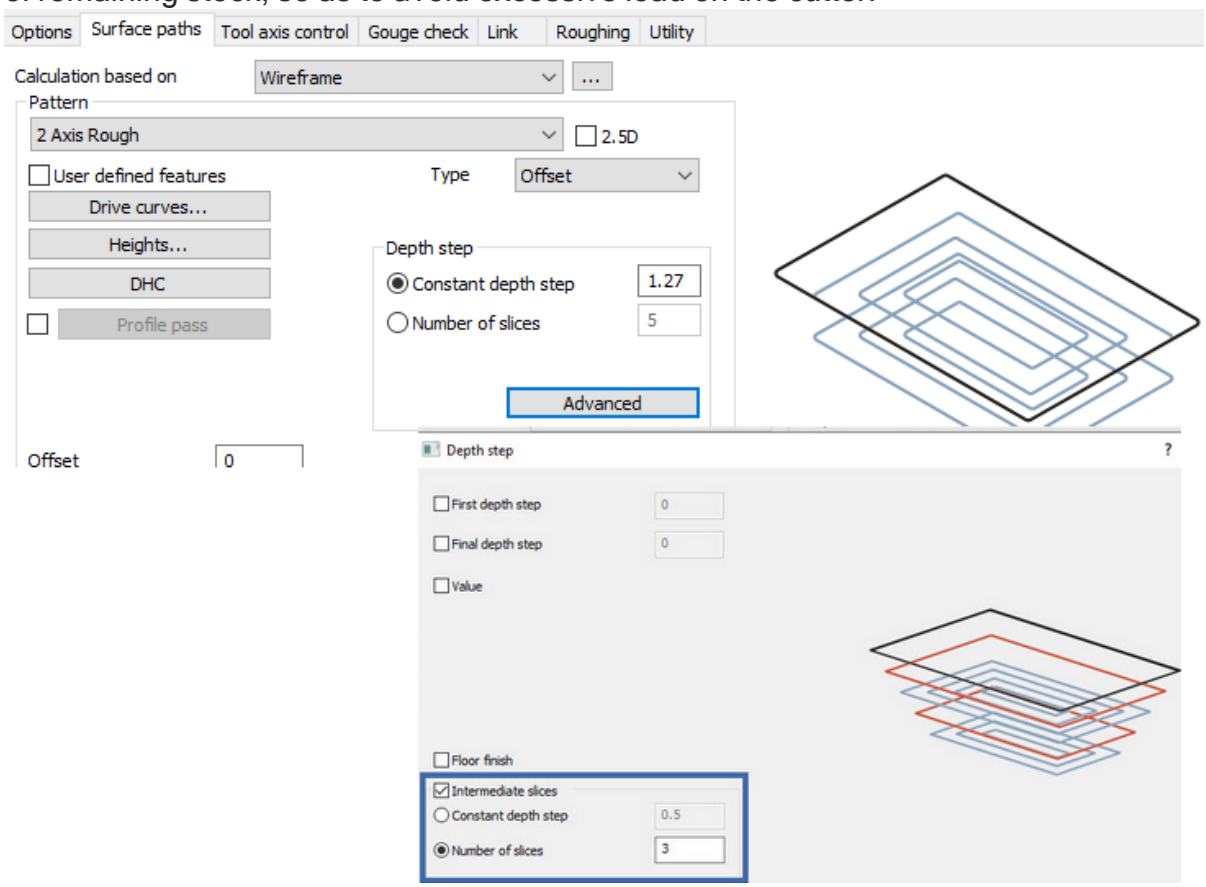


Wireframe 2-Axis Rough: Intermediate Slices

For Wireframe 2D/2.5D roughing, the Depth step Advanced dialog lets you add intermediate slices to reduce the staircase effect and optimize 2.5D parts using the draft angle feature. Benefits include:

- Less material is left when using smaller tools for rest-roughing.
- A uniform thickness is used for semi-finishing toolpaths.
- A uniform tool load is placed on semi-finishing tools.
- More stock removal with larger tools and fewer steps.
- Faster rough machining.

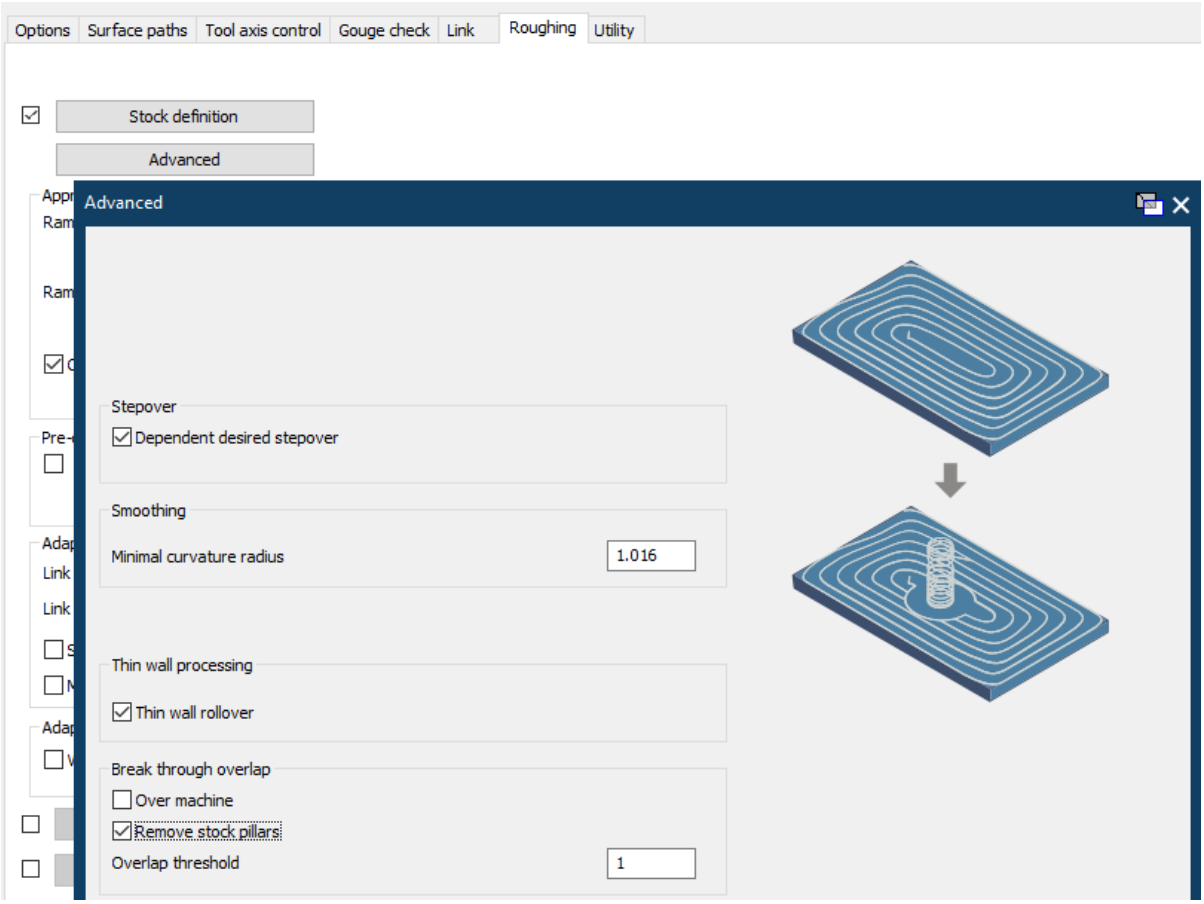
Intermediate slices are rest-roughed. Additional offset passes are added if there is a lot of remaining stock, so as to avoid excessive load on the cutter.



Triangle Mesh

Adaptive Roughing: Remove stock pillars

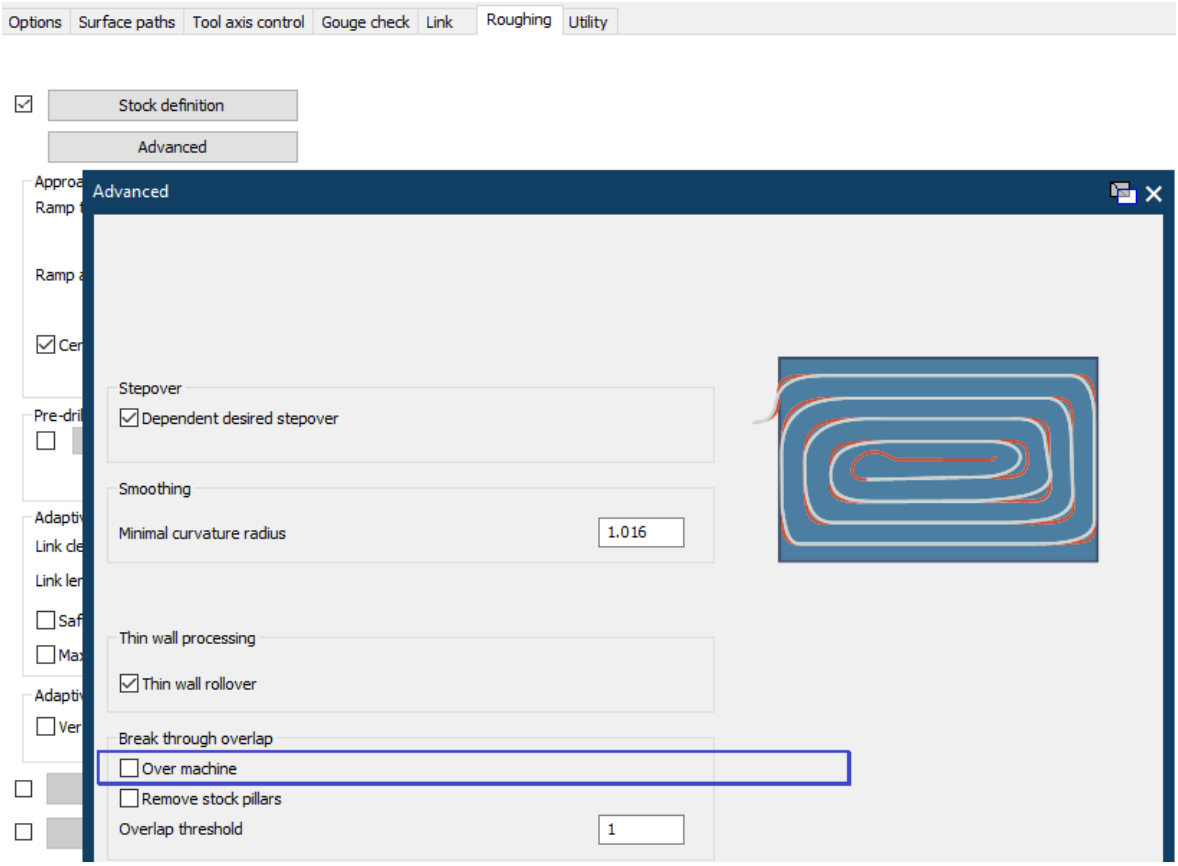
Roughing tab (Triangle Mesh> pattern Rough, type Adaptive) > Advanced: **Remove stock pillars**. This option improves adaptive roughing toolpath for machining open regions. Using standard settings, adaptive roughing for open areas can create a pillar at the center of the region in in-process stock. Remove stock pillars solves this problem by leaving a pillar at the center during adaptive roughing and then removing the pillar later by means a spiral ramp motion for better cutting conditions and tool life.



Adaptive Roughing: Over machine

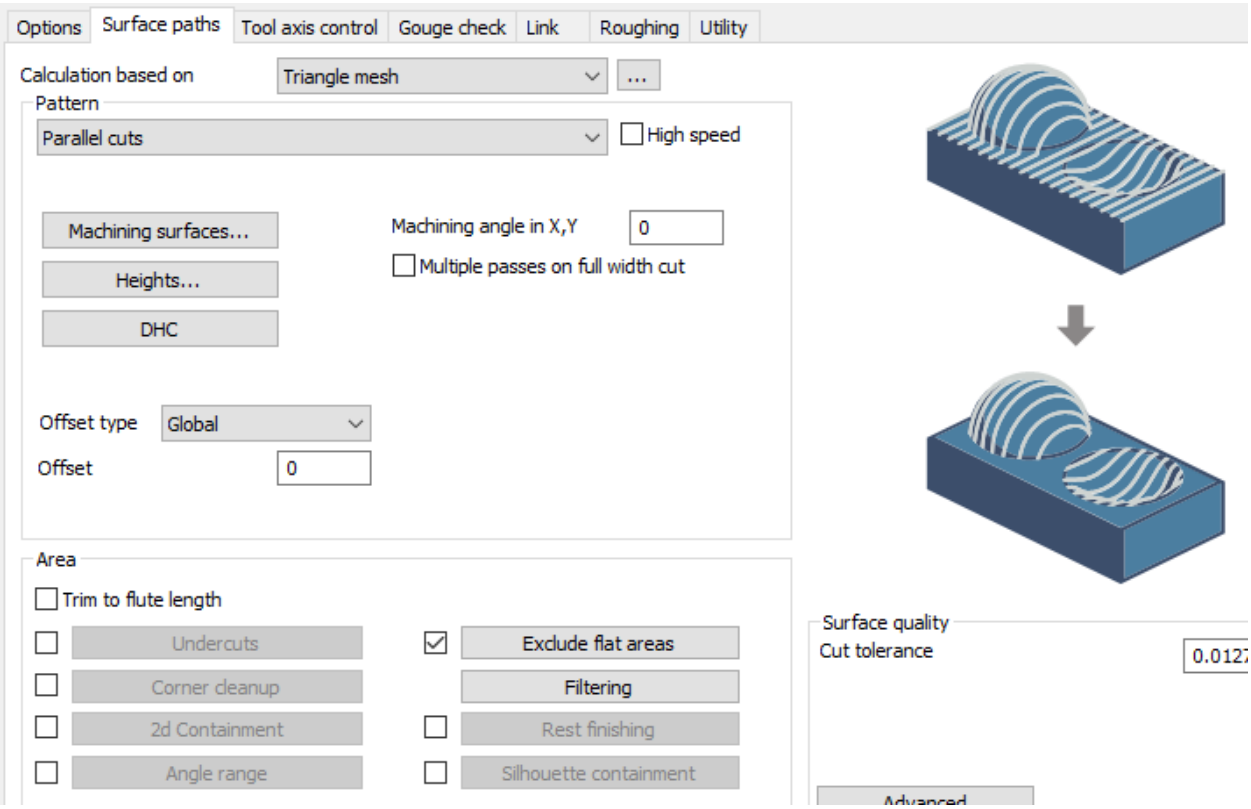
Roughing tab (Triangle Mesh> pattern Rough, type Adaptive) > Advanced: **Over machine**. This option enhances adaptive roughing toolpath to optimize milling conditions for hard metal milling. It changes the shape of toolpath slices to compensate for tool deflections due to high cutting forces. Preventing thin walls generated by tool deflections

lets you avoid vibrations and chatter, prevent thin stock from damaging the tool and spindle, and avoid defects in the workpiece.



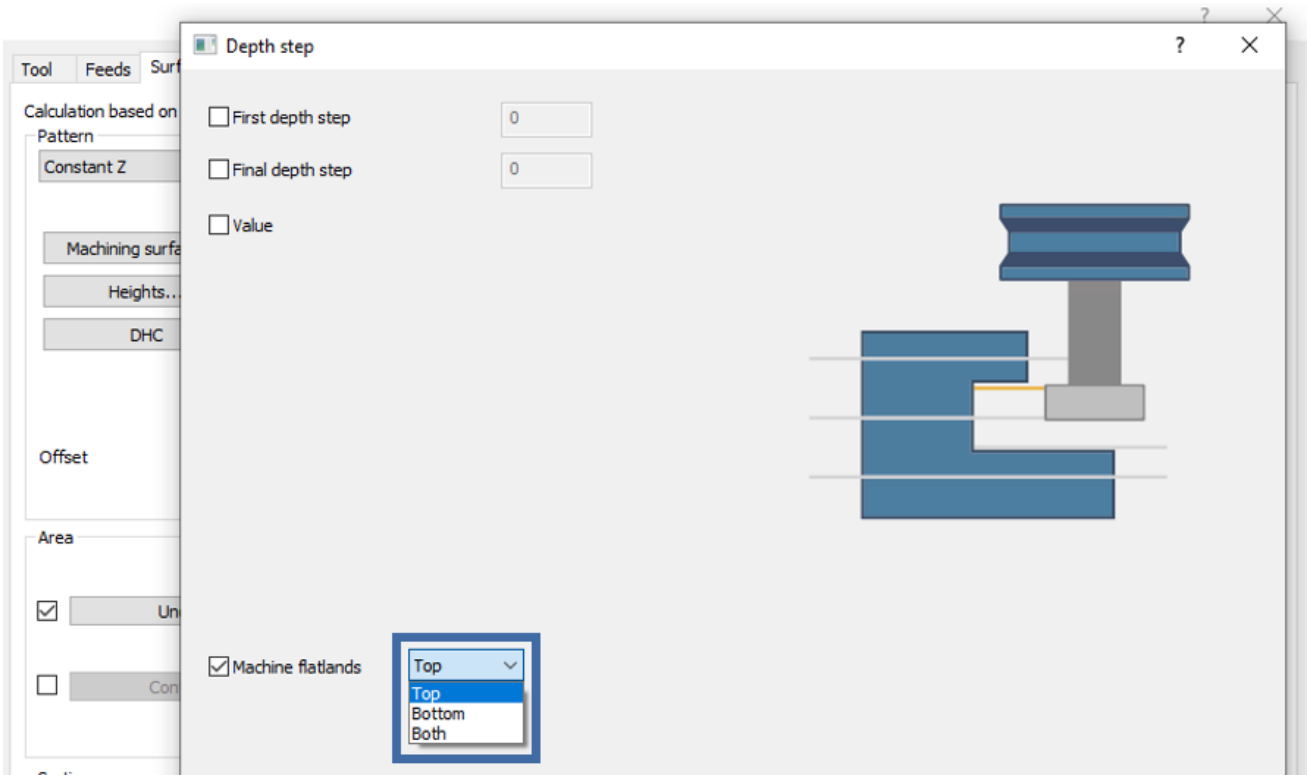
Parallel cuts: Exclude flat areas

Triangle Mesh pattern Parallel cuts > Area: **Exclude flat areas** lets you exclude flat areas while machining a part with spherical or bullnose cutters, thus optimizing machining time. A separate flatlands toolpath can be applied to such areas using an endmill or facemill tool.



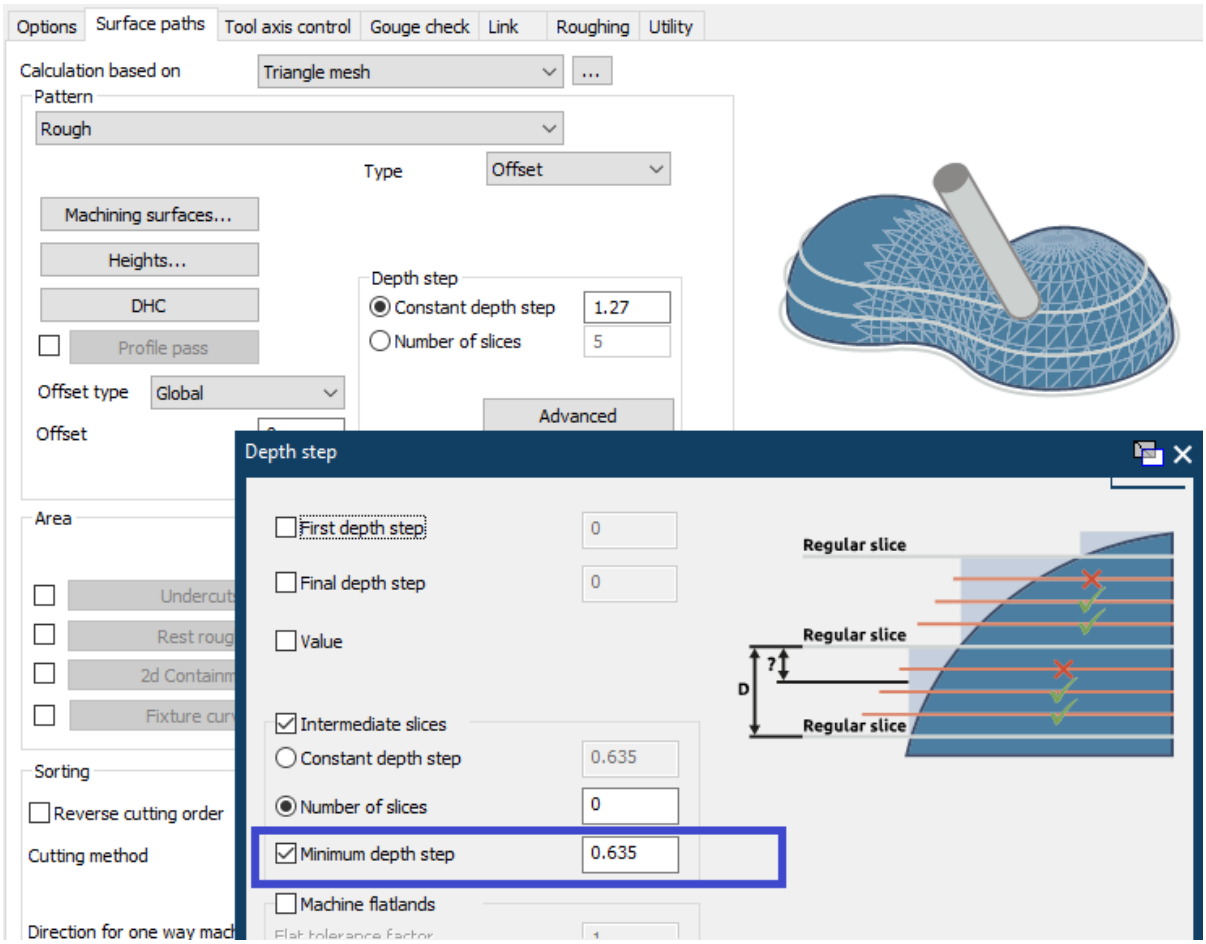
Constant Z Undercuts: Machine flatlands

Triangle Mesh pattern Constant Z > Area > Undercuts: New option **Machine flatlands** enhances the machining of flat areas in undercut regions by detecting the exact Z height of the flat areas. Using slot mills, you now have the option to machine flats at the top, bottom, or on both sides of the undercut areas. Additional toolpath slices are generated on the flat regions of the undercut area, using outside-in ordering to remove the material gradually.



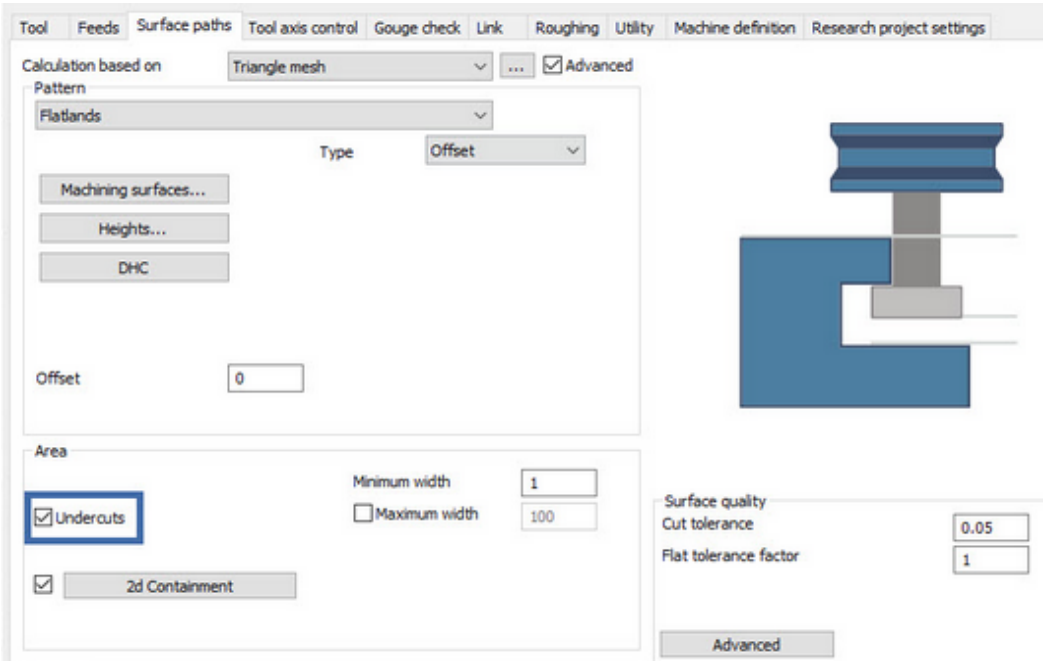
Rough: Minimum depth step

Triangle Mesh pattern Rough > Depth step > Advanced: New option Minimum depth step filters out all intermediate slices that have a depth step smaller than the user-defined threshold for the minimum depth step, to enhance cutting conditions during roughing and reduce vibration and chatter.



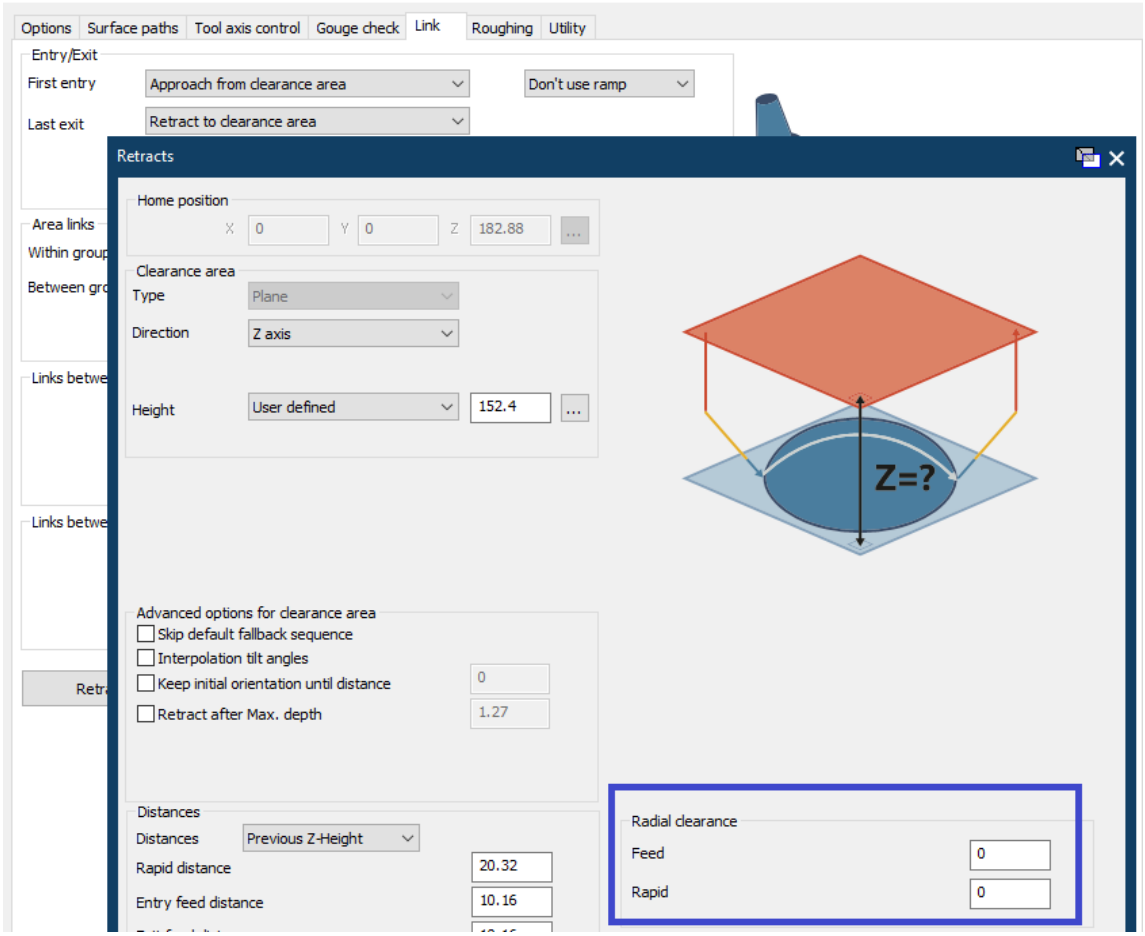
Flatlands: Undercuts

Triangle Mesh pattern Flatlands > Area: New option Undercuts makes it possible to machine undercut flat areas on a part using slot mills. Additional toolpath slices are generated on the flat regions of the undercut area, using outside-in ordering to remove the material gradually.



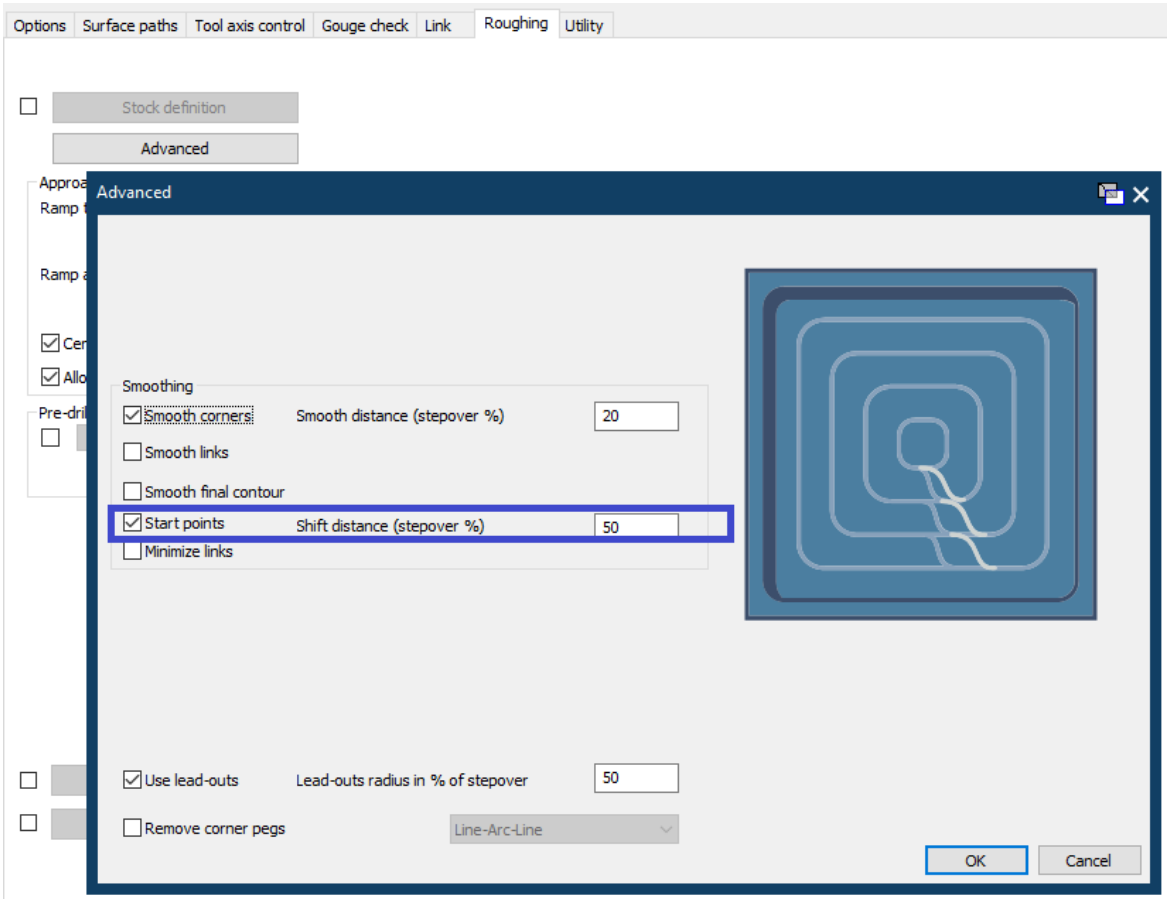
Link tab / Retracts: Radial clearance

In the **Retracts** dialog, you can now specify additional clearances for retract links. The tool is expanded by the specified values, which are used for link checking on collisions. If the retract link height is not sufficient to avoid collisions with the expanded tool, then it is automatically adjusted. *Result:* Links stay farther away from the wall, preventing potential collisions against unmachined remaining stock.



Roughing tab / Advanced: Shift distance (stepover %)

In the **Roughing** tab for Triangle mesh roughing patterns, option Start points provides a way to shift the start point for subsequent cuts by a defined value. This provides a smoother transition to the next pass, resulting in faster milling times because of less deceleration/acceleration during machining.


Options
Surface paths
Tool axis control
Gouge check
Link
Roughing
Utility
Stock definition
Advanced
Advanced
Smoothing
Smooth corners
Smooth distance (stepover %)
20
Smooth links
Smooth final contour
Start points
Shift distance (stepover %)
50
Minimize links
Use lead-outs
Lead-outs radius in % of stepover
50
Remove corner pegs
Line-Arc-Line
OK
Cancel