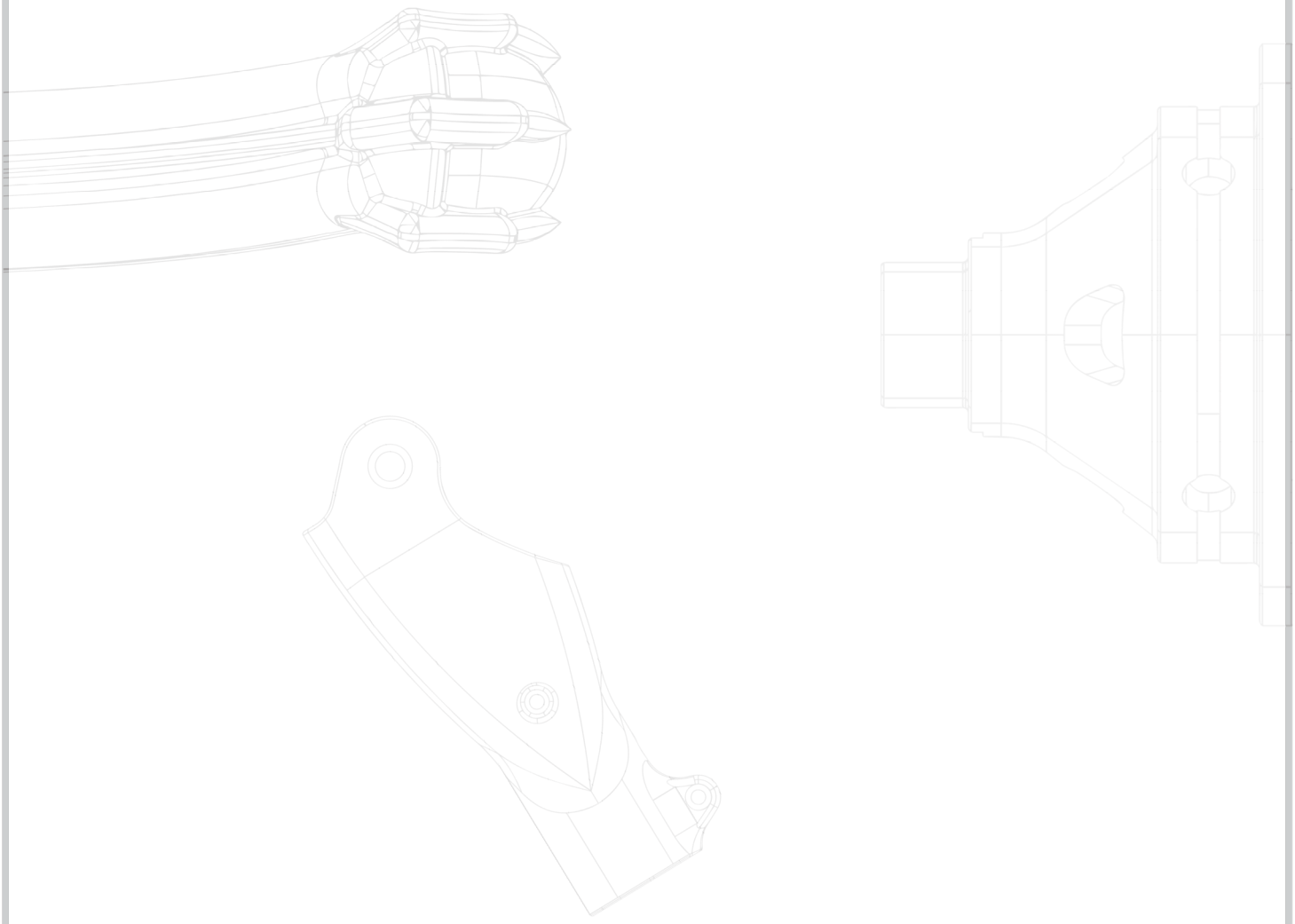
WHAT'S NEW IN MASTERCAM 2021

March 2020



POWERED BY MASTERCAM'S DYNAMIC MOTION™ TECHNOLOGY

Mastercam

WHAT'S NEW IN MASTERCAM 2021

March 2020

Software: Mastercam 2021

Terms of Use

Be sure you have the latest information!

Information might have changed or been added since this document was published. The latest version of the document is installed with Mastercam or can be obtained from your local Reseller. A ReadMe file (ReadMe.PDF) – installed with each release – includes the latest information about Mastercam features and enhancements.

TABLE OF CONTENTS

INTRODUCTION

CAUTION
Please be aware that any information described in this document is subject to change at any time. Features may be removed, added, or changed over the course of the Mastercam Betas.

Welcome to Mastercam 2021! Mastercam 2021 features new functionality focused on delivering speed and efficiency to your machining jobs.

Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Documentation*—Mastercam installs a number of helpful documents for your version of software in the Documentation folder of your Mastercam 2021 installation.
- *Mastercam Help*—Access Mastercam Help by selecting **Help**, **Contents** from Mastercam's **File** tab or by pressing [**Alt+H**] on your keyboard.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—Our Technical Support department (+1 860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Mastercam Tutorials*—We offer a series of tutorials to help registered users become familiar with basic Mastercam features and functions. Visit our website, or select **Help**, **Tutorials** from Mastercam's **File** tab to see the latest publications.
- *Mastercam University*—Mastercam University, an affordable online learning platform, gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master skills at your own pace and help prepare for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit university.mastercam.com/, or email training@mastercam.com.
- *Online Communities*—You can find a wealth of information at www.mastercam.com.
    - Follow us on Facebook (www.facebook.com/Mastercam), Twitter (twitter.com/Mastercam), and Instagram (www.instagram.com/mastercamcadcam/) for the latest tech tips and Mastercam news.
    - See Mastercam in action on YouTube (www.youtube.com/user/MastercamCadCam).
    - For more information on CNC Software, Inc., to find and apply to jobs, and connect with people using Mastercam, visit us on LinkedIn (www.linkedin.com/company/cnc-software/).
    - Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the Mastercam Knowledge Base at kb.mastercam.com.

Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at techdocs@mastercam.com.

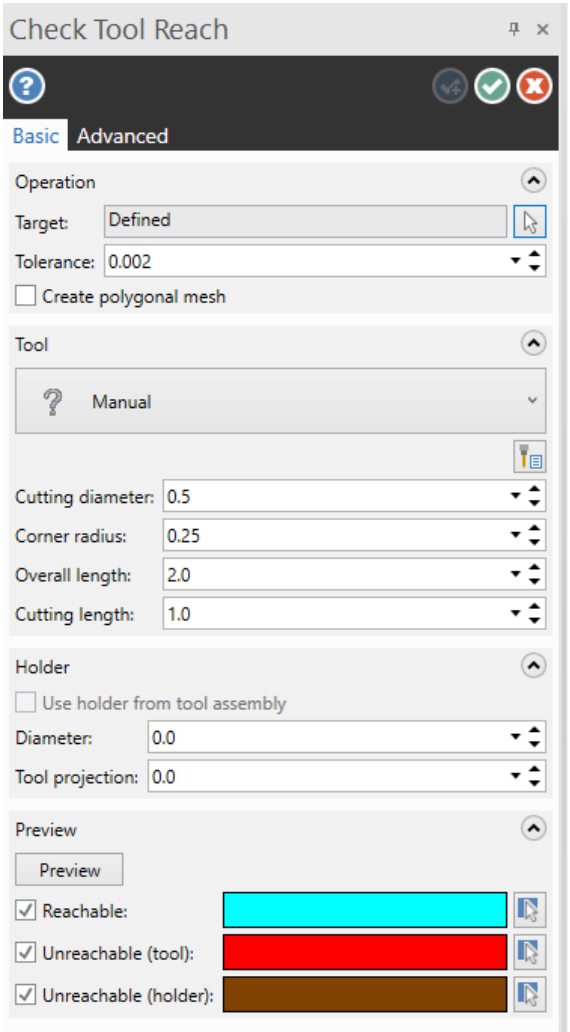
MILLING

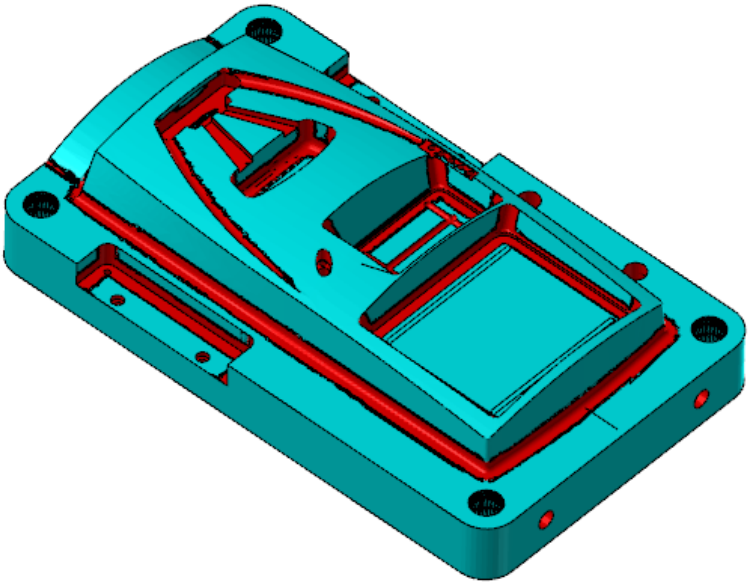
Listed below are major enhancements made to the Mill product. These include improvements to 2D, 3D, and Multiaxis toolpaths.

> **NOTE**
> Unless otherwise stated, the new features and functionality listed in this section also apply if you own a Lathe or Mill-Turn license that has access to the listed Mill toolpaths.

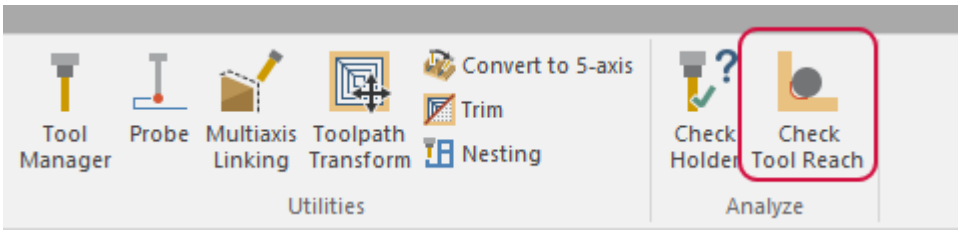
Checking Your Tool and Holder Against Model Geometry

A new function, Check Tool Reach, is now available for Mastercam 2021. Use Check Tool Reach to check your tool and holder against the selected model geometry to view where the tool can and cannot reach. The selected model geometry is then colored based on the selected preview options: **Reachable**, **Unreachable (tool)**, and **Unreachable (holder)**.



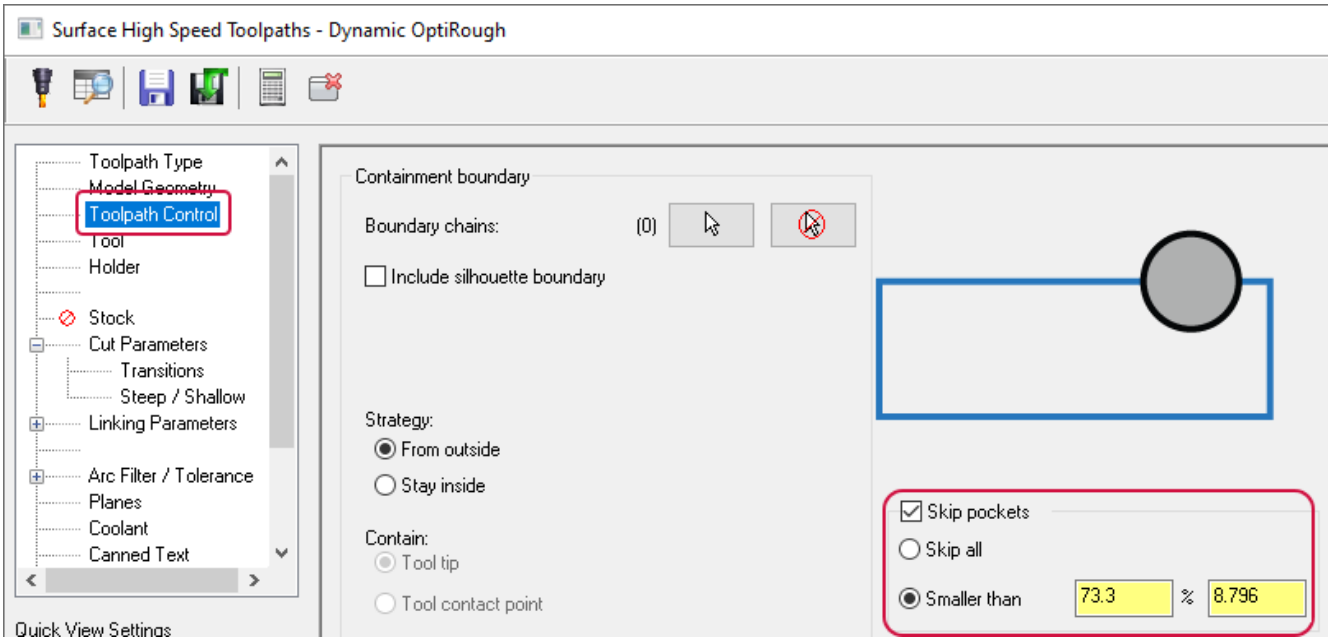


You do not need to select a toolpath operation before launching Check Tool Reach. Check Tool Reach is available on the **Mill Toolpaths** contextual tab.



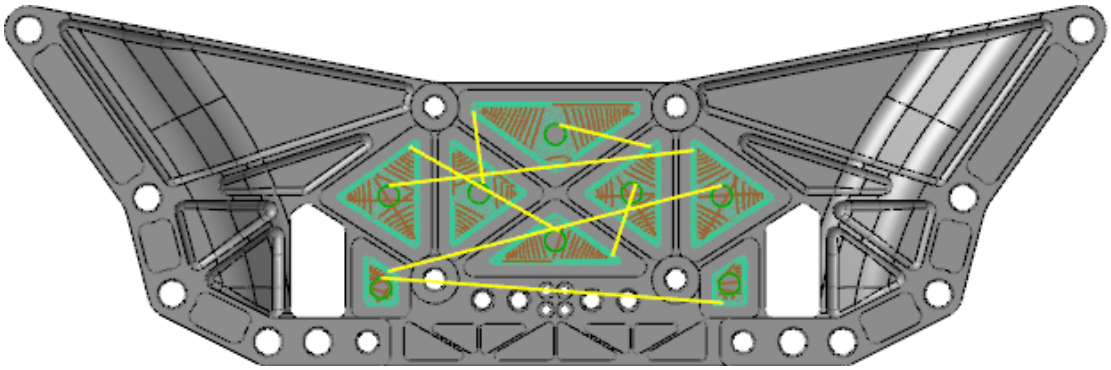
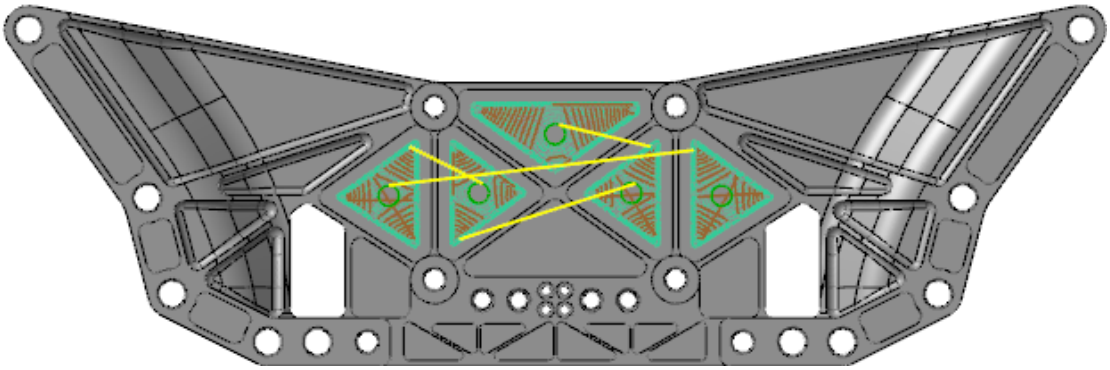
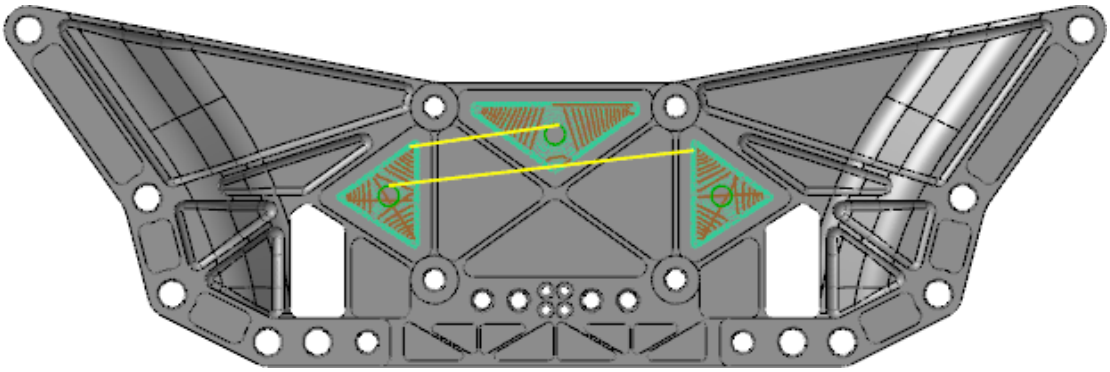
Skip Pockets Based on Tool Diameter or Pocket Size

The **Skip pockets smaller than** option is now available for 2D Dynamic Mill. Additionally, for Dynamic Mill and Dynamic OptiRough, there is a new option to skip all pockets. These options are located on the **Toolpath Control** page for Dynamic OptiRough and the **Entry Motion** page for Dynamic Mill.



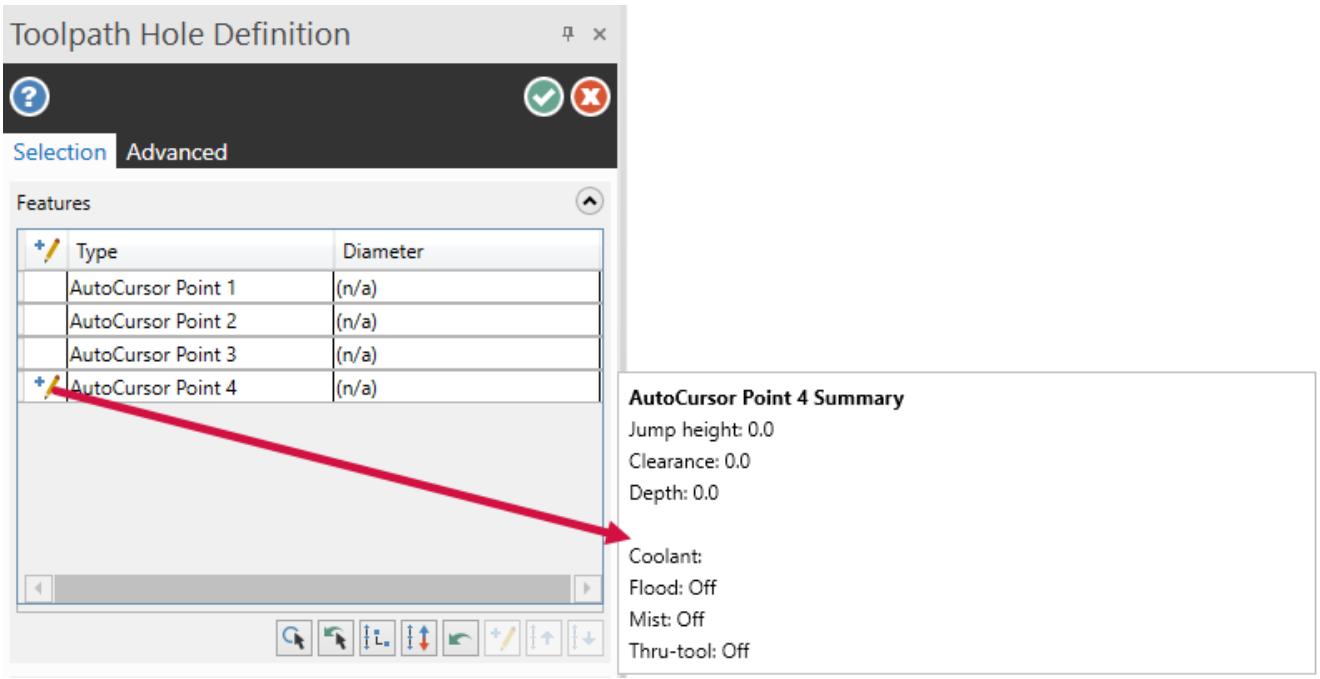
Smaller than allows you to skip pockets based on the entered tool diameter percentage or a minimum pocket size. When you enter a value for either the **Tool diameter percentage** (on the left) or for the **Minimum pocket size** (on the right), the other parameter updates. **Skip all** skips all pockets.

The images below show how different **Smaller than** percentages affect a Dynamic Mill toolpath using a flat end mill tool with a tool diameter of **12.0** (mm).

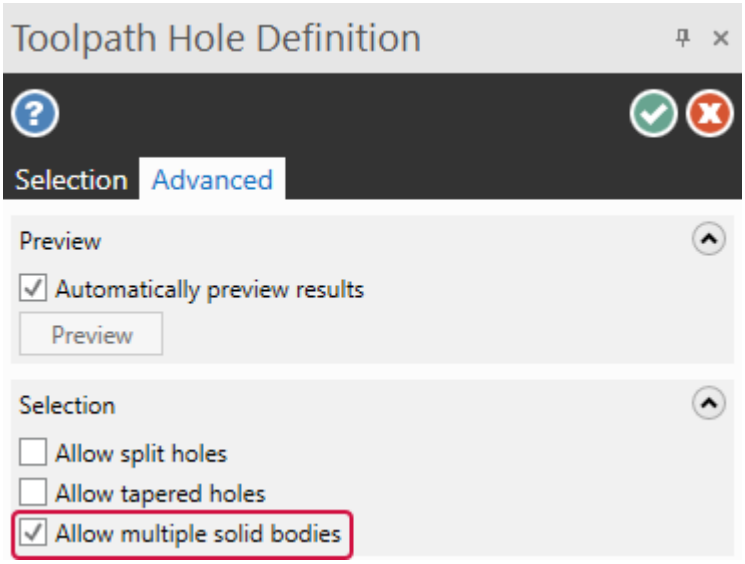
Tool diameter percentage	Resulting toolpath
0%	
300%	
350%	

Selection Improvements for Toolpath Hole Definition

The Toolpath Hole Definition function panel includes two enhancements. On the **Selection** tab, a summary of all edited **Change At Point** parameters are now conveniently displayed on a tooltip in the **Features** list.



The **Advanced** tab now features the **Allow multiple solid bodies** option. This option recognizes same-sized holes across multiple bodies when selecting with [**Ctrl+Click**].



Mill Tools

Listed below are enhancements to Mill tools.

Improved Barrel Form Tool Definition

The Barrel Form tool is now parametrized from the **Tip flat radius**, like the Taper Form tool, instead of from the **Cutting length**. This provides a straightforward method for you to specify a Barrel Form tool. This also decouples the cutting length from the cutting geometry, which is more consistent with most of our tool definitions.

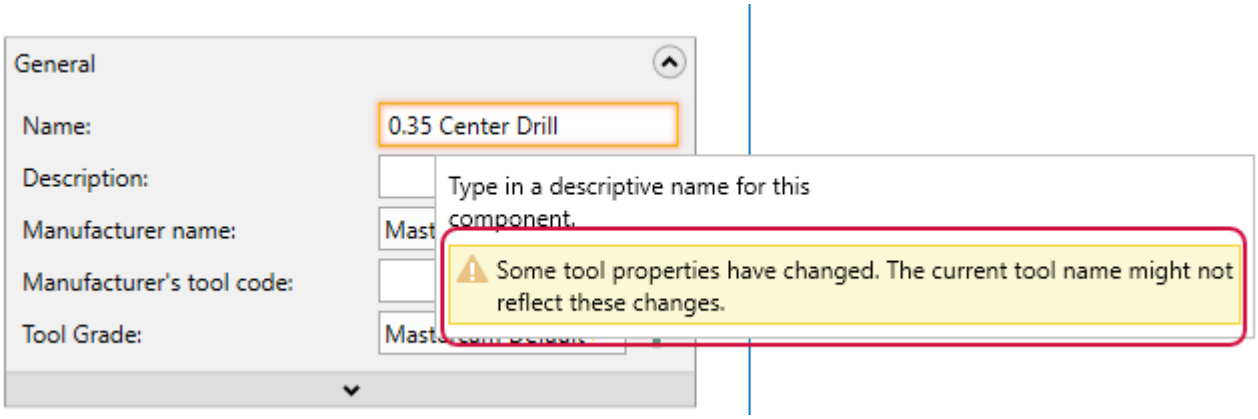
Auto-naming Consistency

Tool auto-naming now follows the rules listed here:

- All inch-unit default tools and catalog tools now have the same name format. The diameter of the tool is first, followed by the tool type.
- If the tool is selected from the catalog list, the naming order is diameter, catalog name, and then tool type.
- Metric catalog tools do not have the diameter added to the catalog name. The tool's name is only the catalog name followed by the tool type.
- Editing a drill and changing the cutting diameter updates the drill's name.
- Editing a drill and selecting a catalog drill also updates the drill's name.

Additional Feedback When Naming Tools

When you change certain tool attributes in the Tool Wizard, the **Name** field on the **Finalize Properties** page becomes highlighted, warning you that the tool's name may need to be updated. Also, when the **Name** field is highlighted, additional information appears in the tooltip, explaining why the field needs your attention.

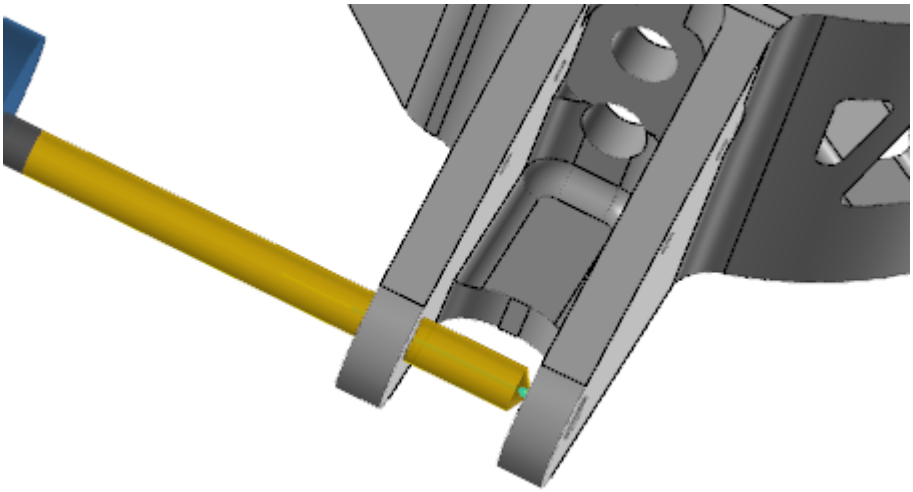


2D Enhancements

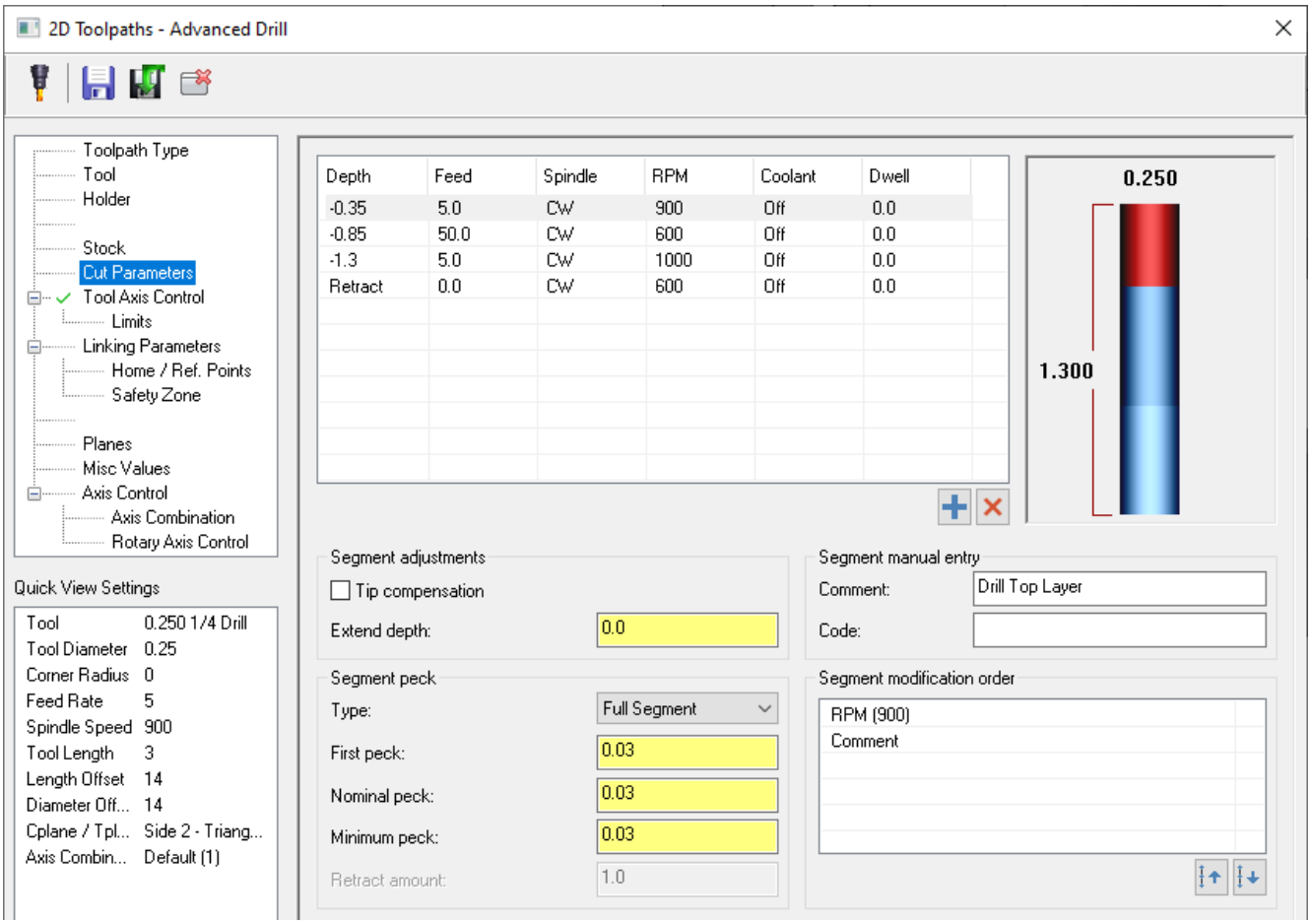
Listed below are enhancements made to the 2D toolpaths.

Machining with a Customizable Multi-Segment Drill Cycle

Mastercam 2021 features a new toolpath, Advanced Drill.



Advanced Drill is a customizable multi-segment drill cycle that is useful for spot drilling, deep hole drilling, and back spot facing. Each segment of the drill cycle can be defined in a table on the toolpath's **Cut Parameters** page.

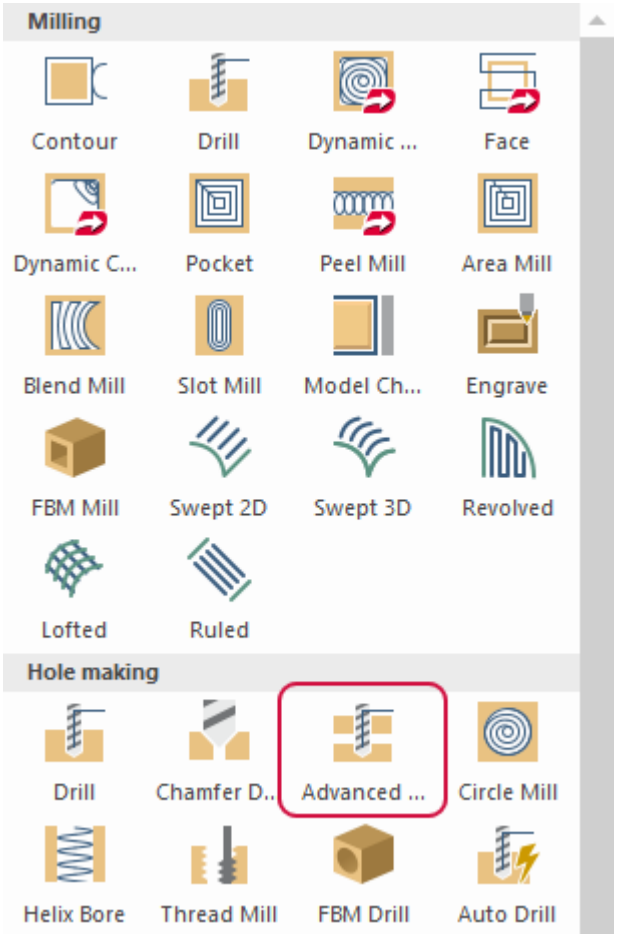


For each segment, you can customize any or all of the following conditions:

- Define individual pecks within the segment, including retracts to simulate a chip break cycle.
- Set the feed rate or choose rapid motion.
- Change the spindle speed and direction.
- Turn coolant options on or off.
- Add manual entry comments or code.
- Fine-tune the order in which the commands are output.

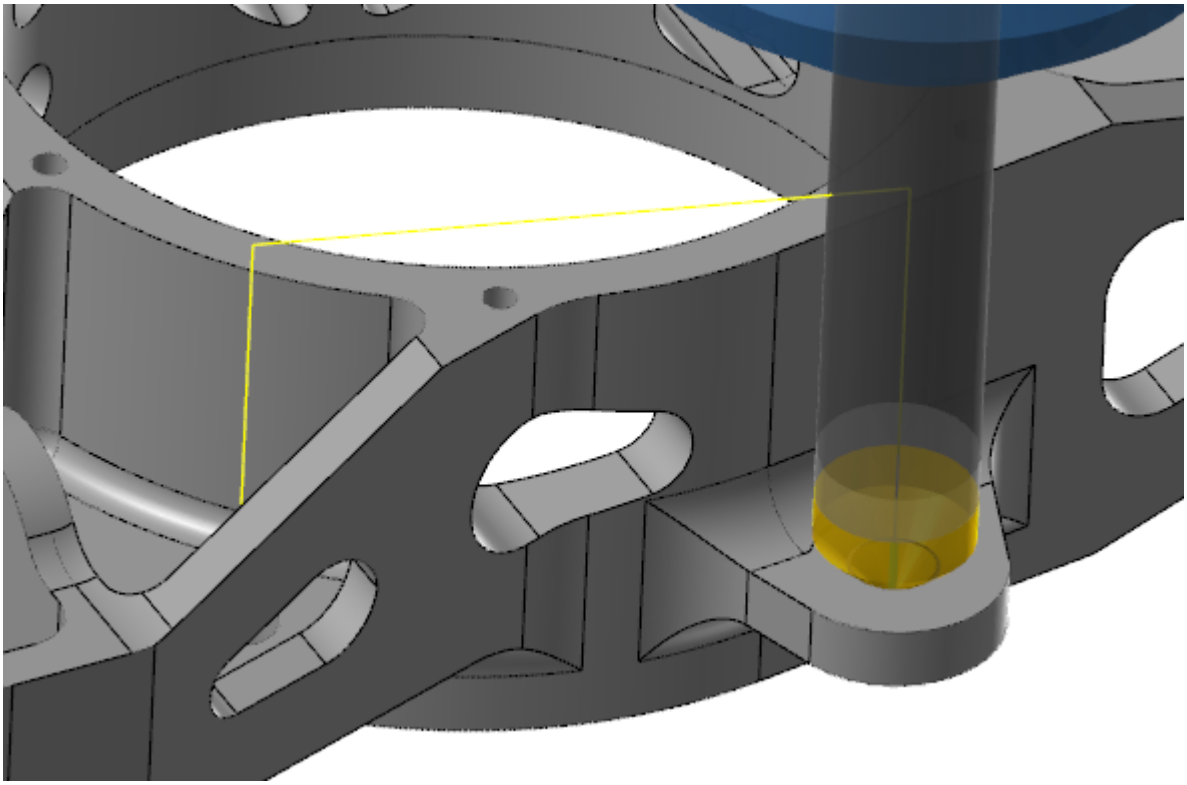
Like other drill operations, you can use the **Tool Axis Control** page to choose **3-**, **4-**, or **5-axis** output. Choosing **4-axis** or **5-axis** output gives you access to additional multiaxis features, like Safety Zone.

Advanced Drill is located in the **2D** gallery on the **Mill Toolpaths** contextual tab.



Machining Chamfer Holes Based on Width or Depth

Mastercam 2021 features a new toolpath, Chamfer Drill. Using tools with a tip angle, the Chamfer Drill toolpath chamfers holes after calculating the correct depth based on the desired width or depth. The Chamfer Drill toolpath also lets you select holes of different diameters or sizes—or that lie in different planes—and machine them in a single operation with a single tool.

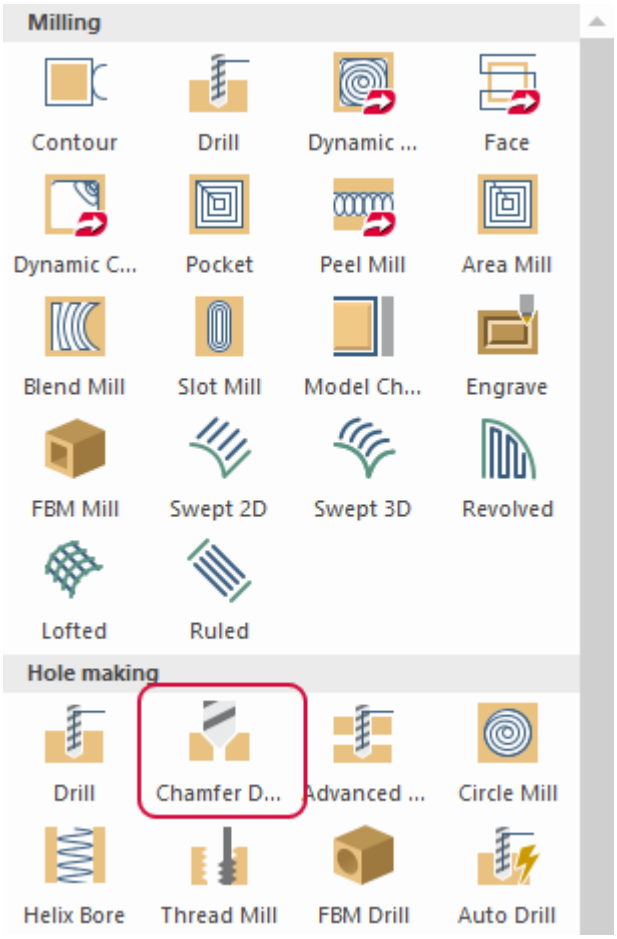


You can use any tool with a tapered tip, not just a drill. Like other drill operations, you can use the **Tool Axis Control** page to choose **3-**, **4-**, or **5-axis** output. Choosing **4-axis** or **5-axis** output gives you access to additional multiaxis features, like Safety Zone.

To create the toolpath, select Chamfer Drill in the Hole making section of the 2D Mill toolpaths. Then, in the graphics window, select the entities you want to add to the **Features** list. Use the following methods to make and manipulate your selections:

- Select entities to add or delete them from the **Features** list.
- Click or use window selection to choose solid holes, solid arc edges, wireframe arcs, lines, points, or AutoCursor positions.
- [**Ctrl+Click**] to select all matching radius solid features.
- [**Ctrl+Shift+Click**] to select all matching radius solid features on the same vector as the initial selection.
- Click a selected solid feature's arrow to change direction.

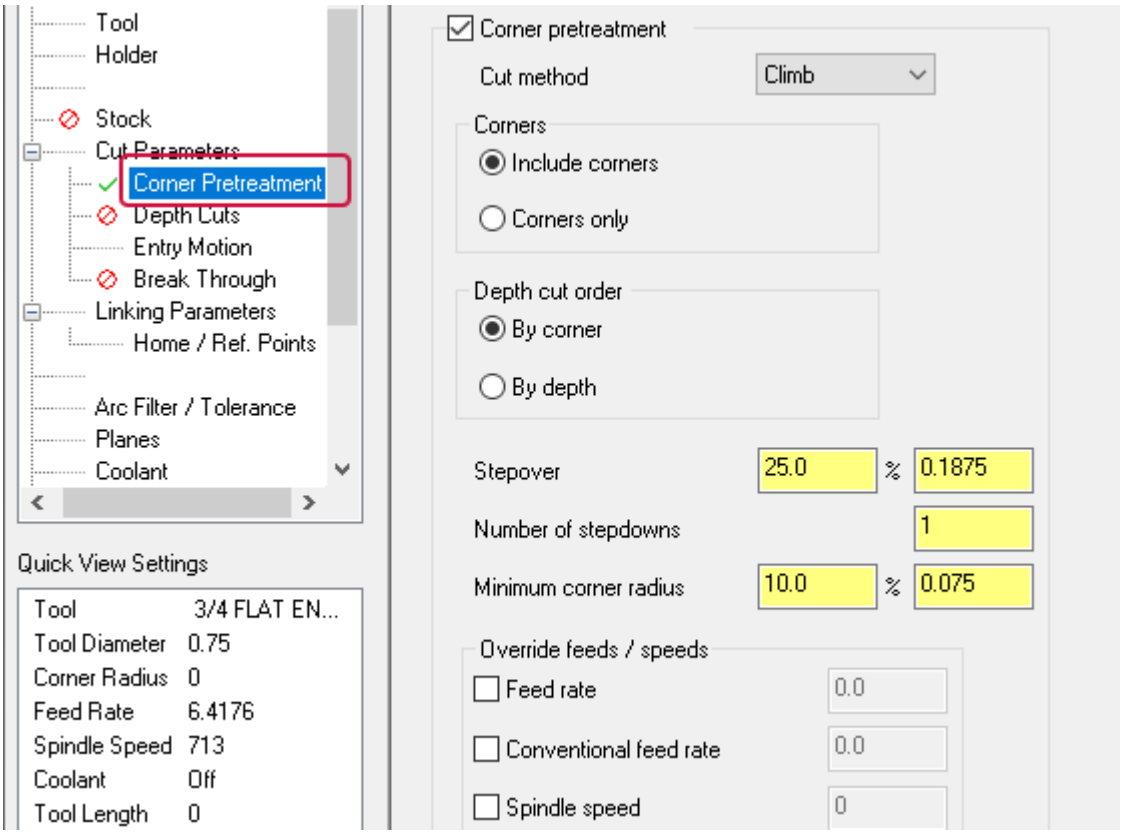
Chamfer Drill is located in the **2D** gallery on the **Mill Toolpaths** contextual tab.



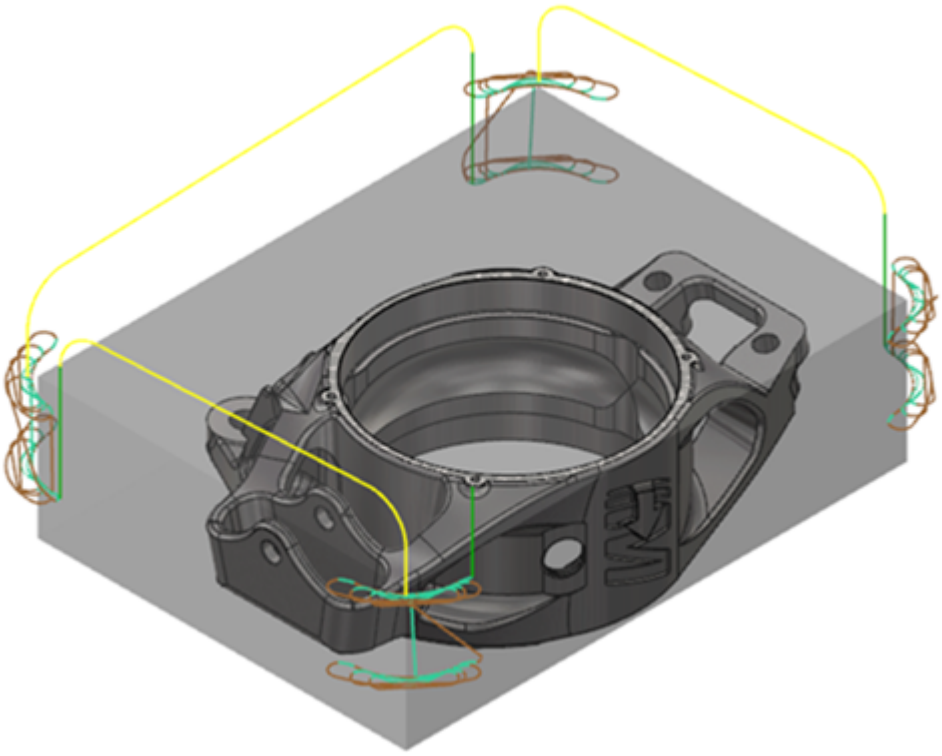
NOTE
Currently, Chamfer Drill uses longhand output. Canned cycles are not available at this time.

Pre-Machining Corners for Dynamic Mill Toolpaths

A new page, **Corner Pretreatment**, has been added to the Dynamic Mill toolpath. Use the options on this page to set machining parameters on any corners in the selected machining regions before machining the rest of the part. These new options are particularly helpful if you have sharp corners in your part.

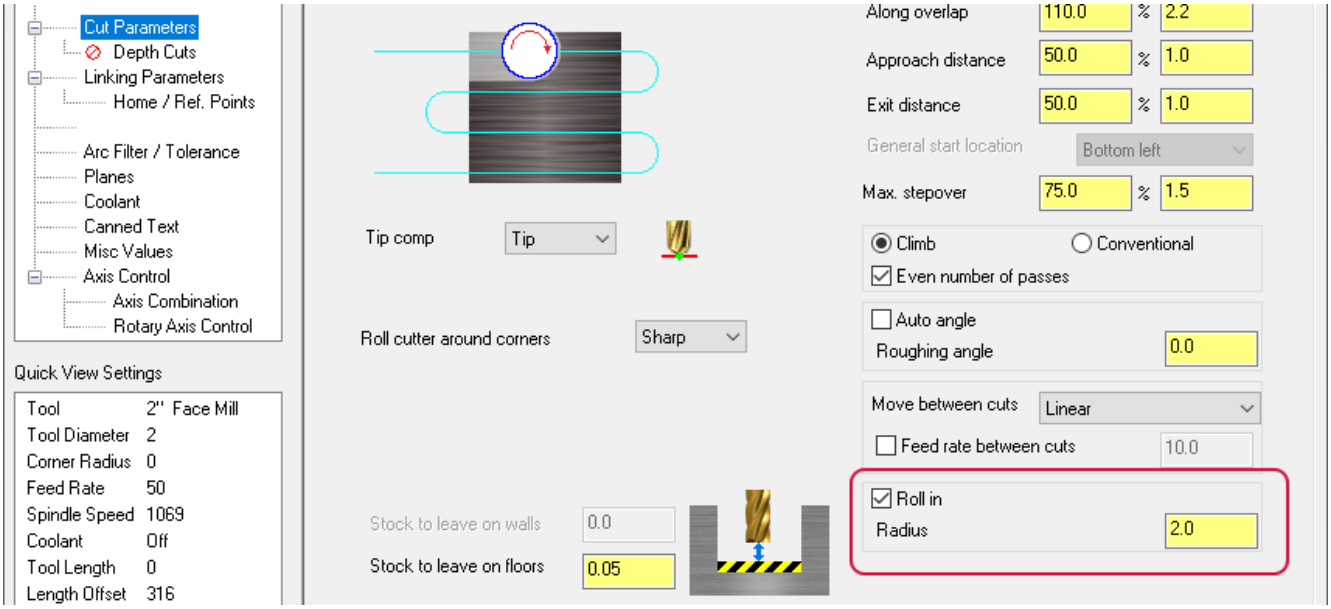


Additionally, this can reduce the amount of thin wall islands that are created when machining.

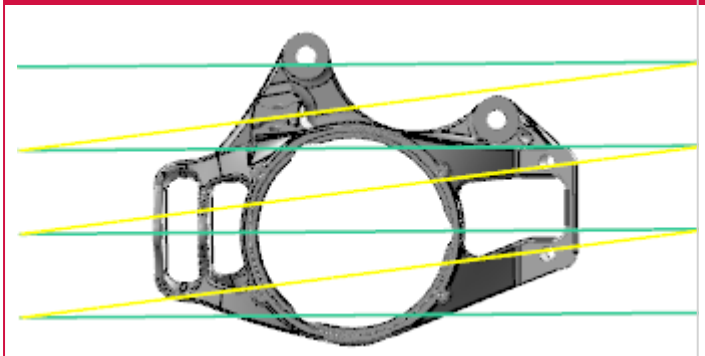
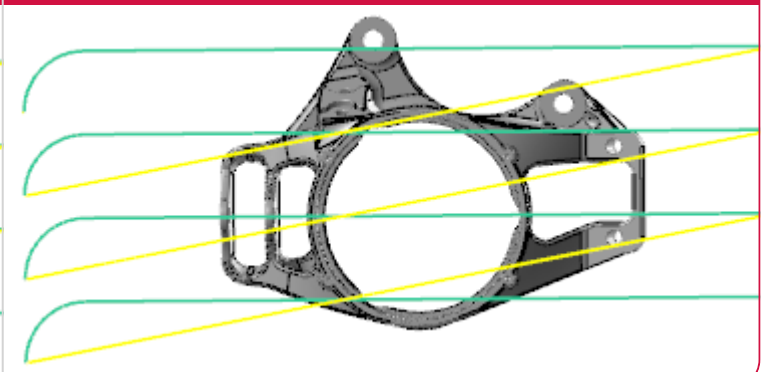


Adding a New Roll In Move for Face Toolpaths

A new option, **Roll in**, has been added to the **Cut Parameters** page for Face toolpaths.



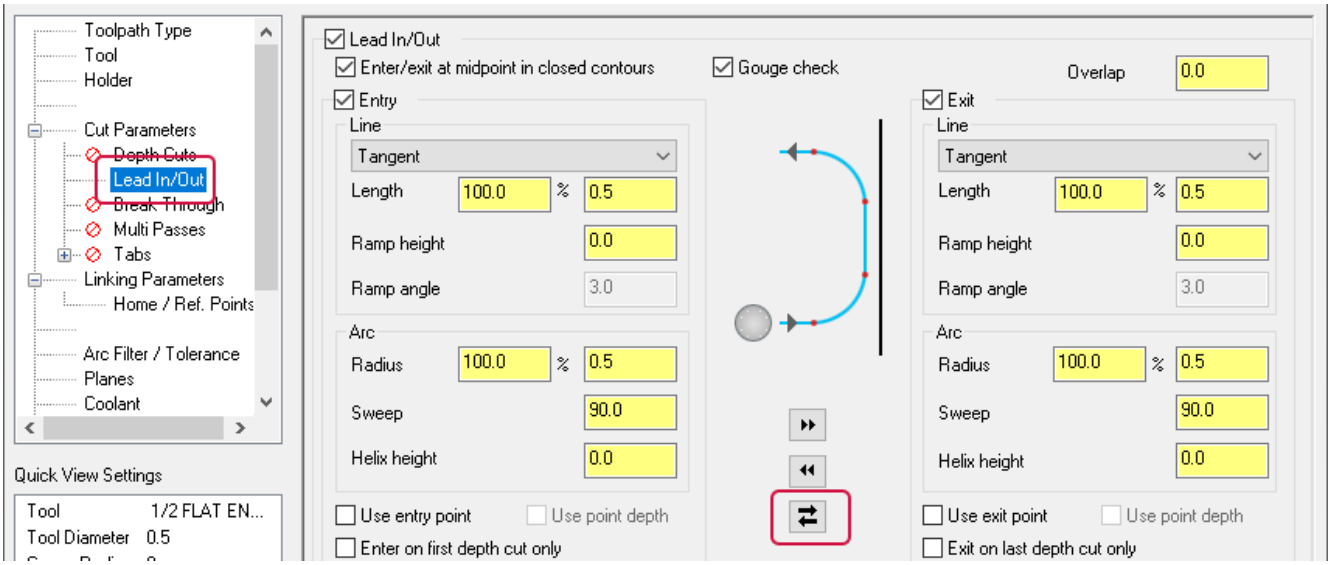
Roll in adds an arc move with the entered **Radius** to the lead in move. This move is always opposite the stepover direction. This option is particularly helpful with tool engagement, resulting in less tool stress.

Roll in off	Roll in on (Radius set to 1.0)
	

NOTE
When **Style** is set to **One way**, the arc is applied to each cut. When set to **Zigzag**, the arc is only applied to the first cut. This option is unavailable for **One pass** and **Dynamic**.

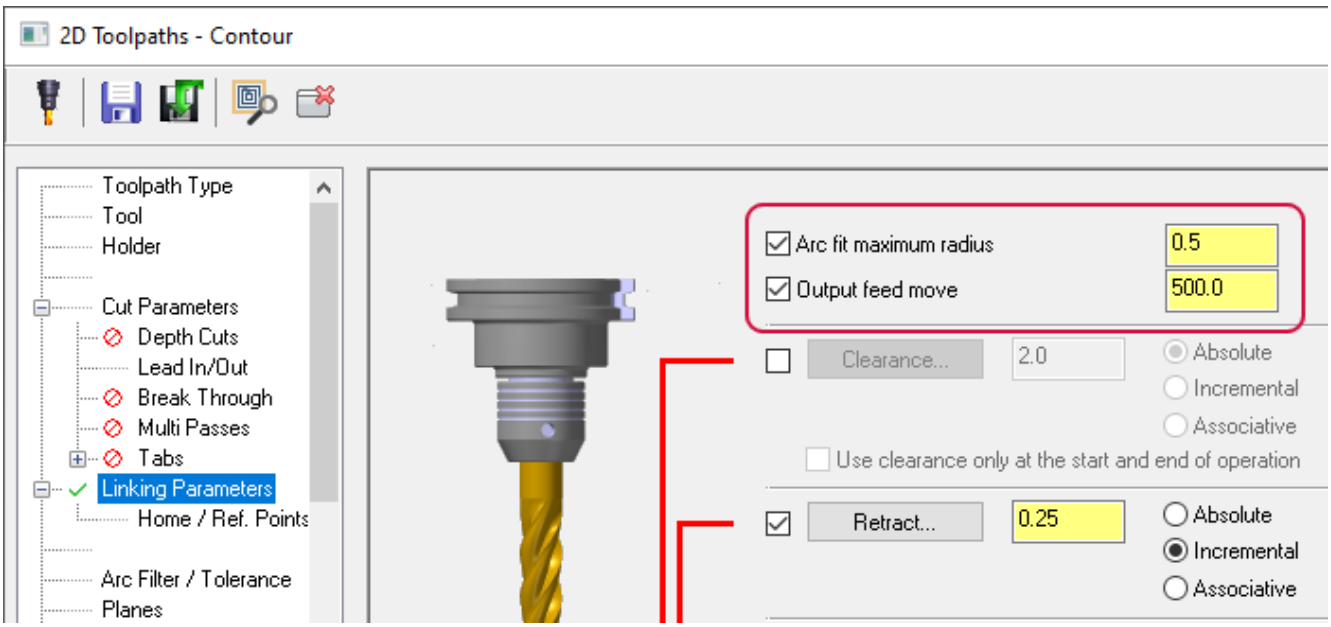
Swapping Lead In and Lead Out Parameters

A new option has been added to the **Lead In/Out** page for 2D toolpaths. This new button lets you swap the parameters values for the **Entry** or **Exit** sections without losing information.



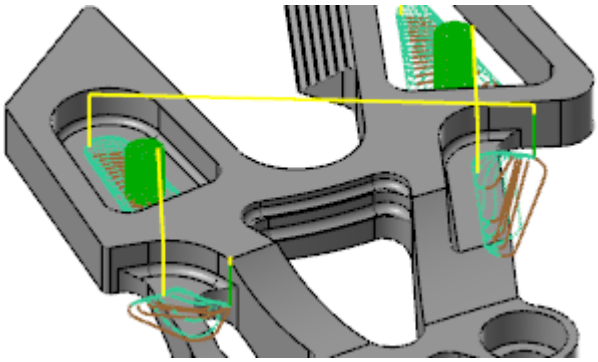
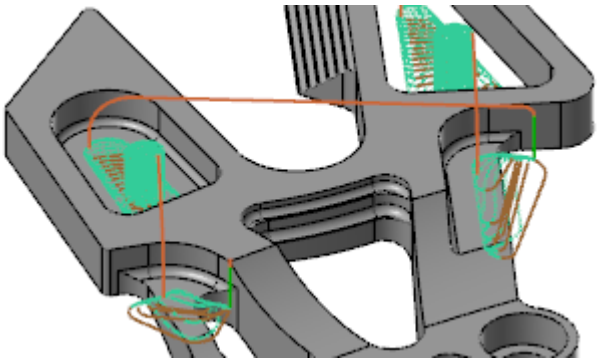
Using New Arc Fit Linking and Feed Move Override

Two new options are available on the **Linking Parameters** page for all 2D toolpaths.



The **Arc fit maximum radius** option attempts to fit 90-degree arc moves into rapid and clearance moves. If the entered radius does not fit, then the output is created as an 180-degree arc.

Use the **Output feed move** option to output the rapid move between passes as a feed rate move instead of a rapid move. This may be helpful when the tool needs to make many irregular moves per pass to jump between different areas of the part. This option is also useful for older machines which create dogleg moves for rapids.

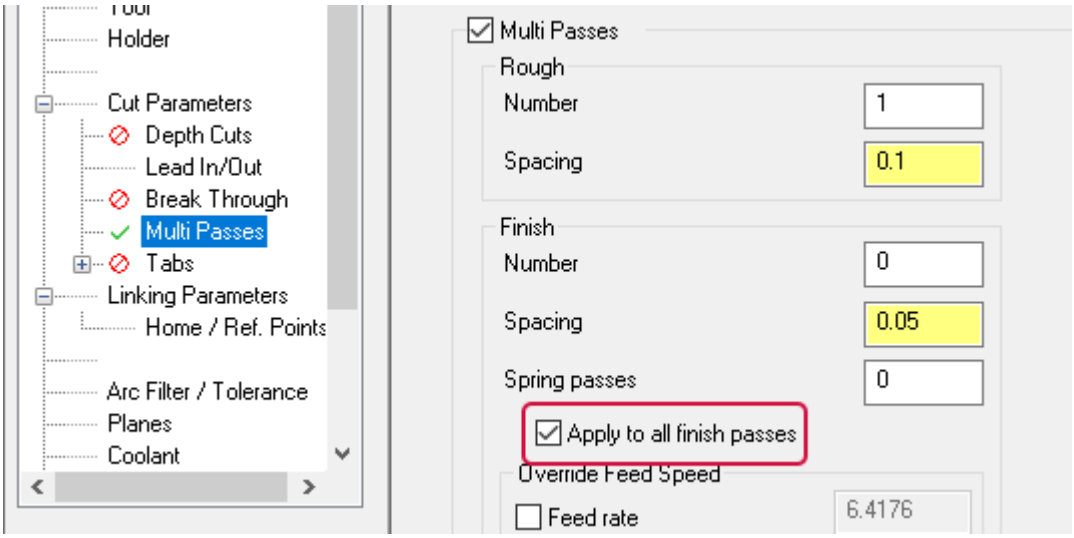
Both options off	Both options on
	

Contour Enhancements

The 2D Contour toolpath now includes the following enhancements.

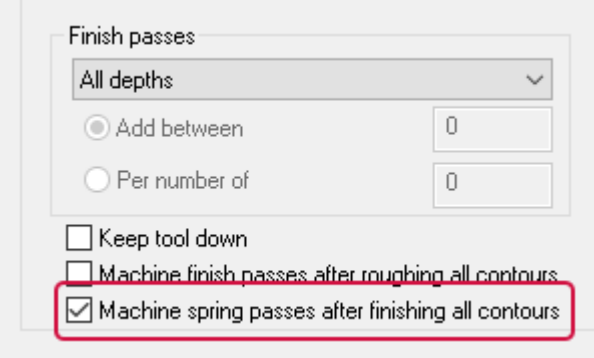
Additional Spring Passes Improvements

A new option, **Apply to all finish passes**, on the **Multi Passes** page applies spring passes after the last finish pass for each defined depth set by the **Finish passes** parameter.



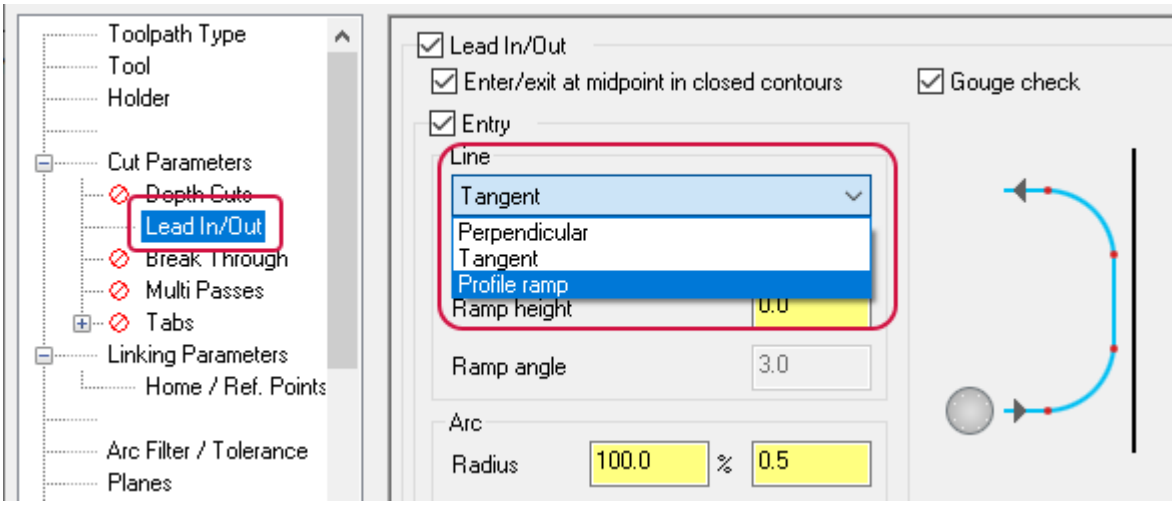
When deselected, spring passes are only applied after the last finish pass at the final depth.

Another new option on the **Multi Passes** page, **Machine spring passes after finishing all contours**, allows you to perform spring passes after all finishing cuts and depth cuts have been completed.

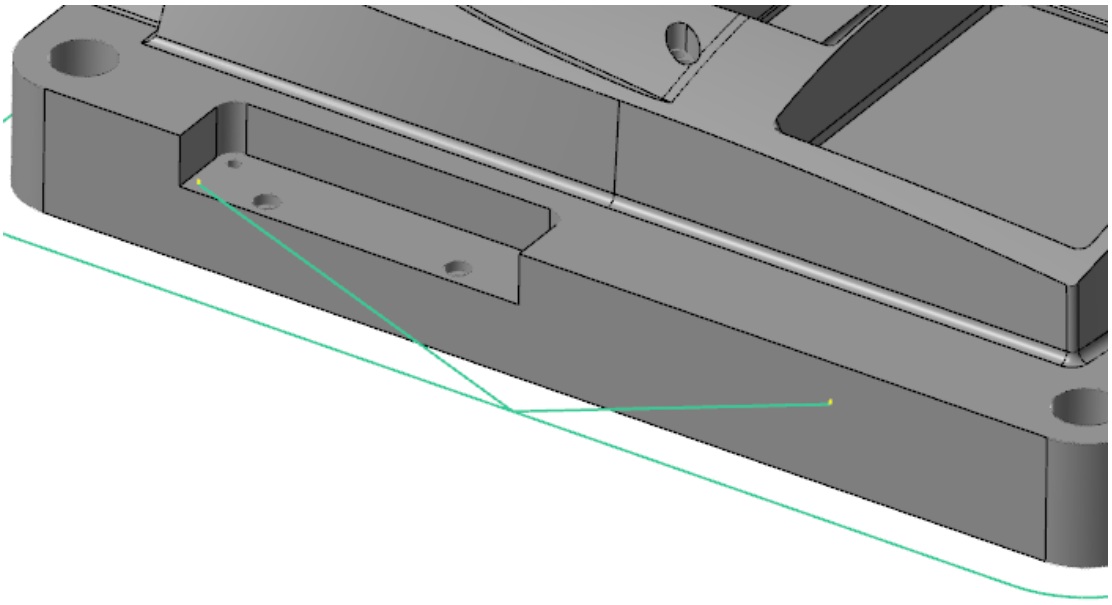


Profile Ramping for Lead In/Out Moves

Profile ramp is a new option in the **Line** drop-down on the **Lead In/Out** page for 2D Contour toolpaths.

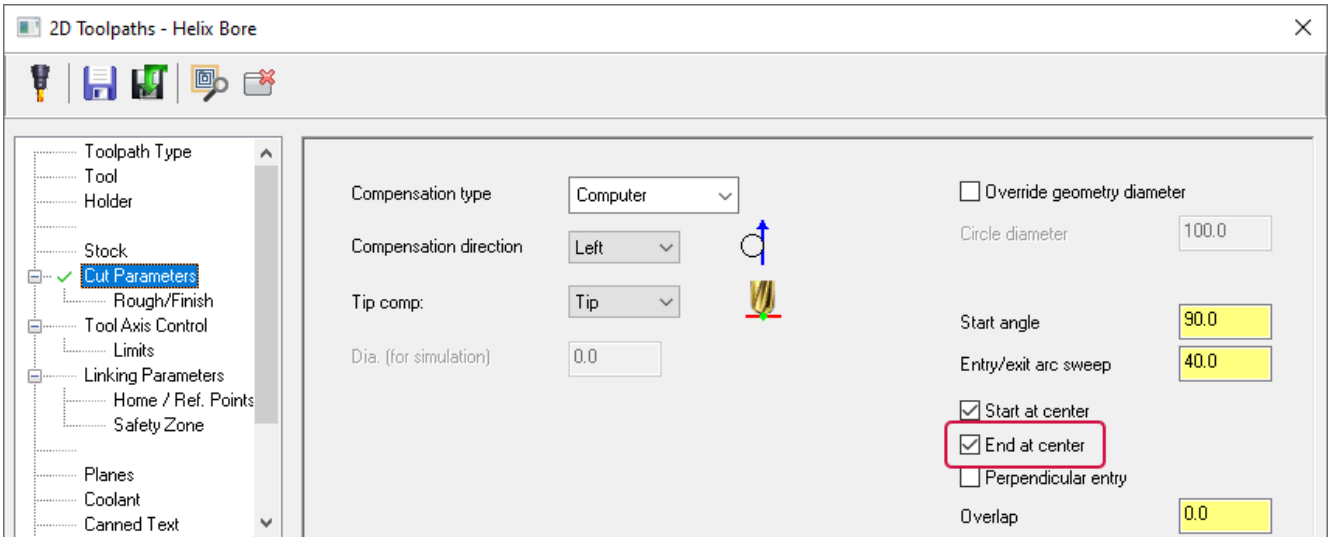


When you select this option, you can only modify the **Ramp angle** value. **Profile ramp** uses a continuous ramp to transition smoothly between lead in and lead out moves.



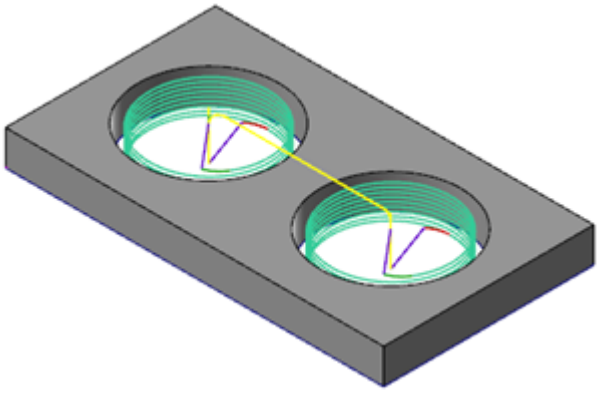
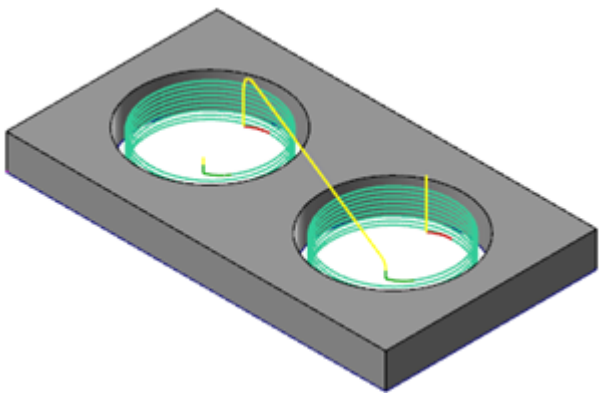
Ending at Center for Circle Mill and Helix Bore Toolpaths

For more consistency between toolpath options, an **End at center** option has been added to the **Transitions** page of the Circle Mill toolpath and the **Cut Parameters** page of the Helix Bore toolpath.



You can now choose both **Start at center** and **End at center** in these toolpaths.

- **Start at center** begins the toolpath at the center of each selected entity. Once that entity has been machined, the toolpath moves to the next entity.
- **End at center** ends the toolpath at the center of each selected entity. Once that entity has been machined, the toolpath moves to the next entity.

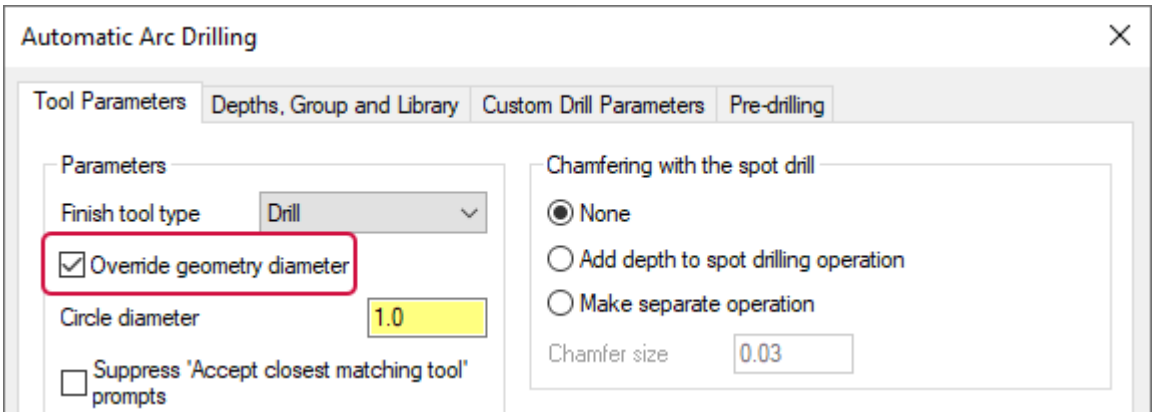
Start and End at center on	Start and End at center off
	

Improved Arc and Wireframe Point Selection for Auto Drill Toolpaths

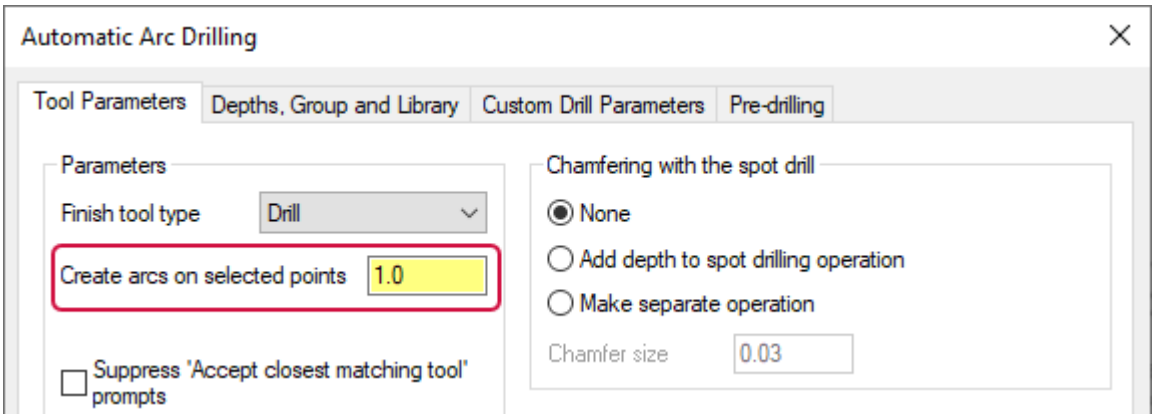
> **NOTE**
> The Auto Drill toolpath is not available for Mill-Turn.

When using the Auto Drill toolpath to create operations for tapped holes, Mastercam chooses the tap size based on the diameter of the selected arcs. This can be a problem when the geometry is based on the minor diameter of a tapped hole.

Now, when you select only arcs, Mastercam displays the **Override geometry diameter** parameter in place of the **Create arcs on selected points** on the **Tool Parameters** tab. Mastercam uses the existing hole diameter unless you select the override option.

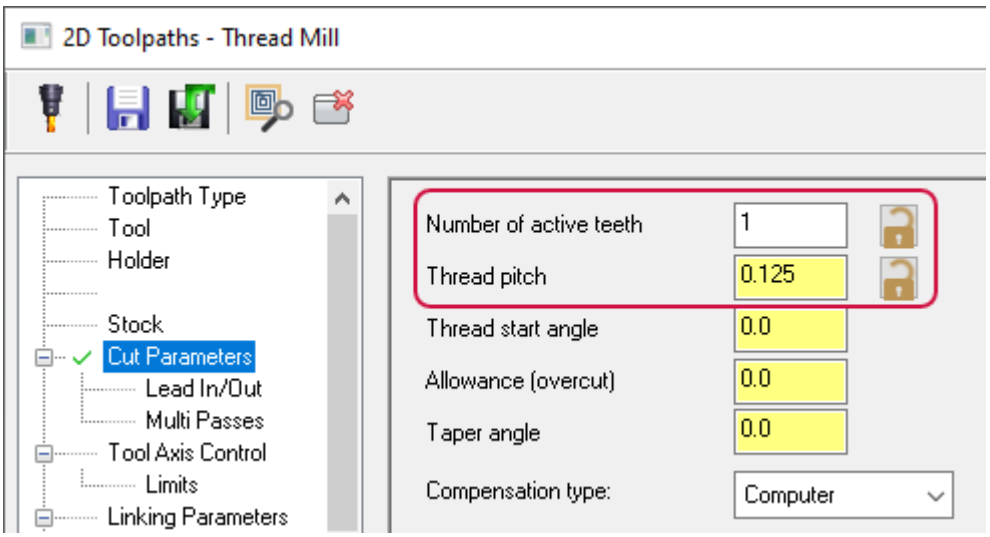


If any wireframe points are selected, the **Create arcs on selected points** parameter displays, and the user-entered diameter associated with that field defines the hole diameter.



Locking Parameters for Thread Mill Toolpaths

Because Mastercam automatically calculates fields like **Thread pitch**, manual changes to such fields can be overwritten whenever Mastercam is called to recalculate these values. You can now lock **Number of active teeth** and **Thread pitch** to ensure that Mastercam does not recalculate these values. Click the lock icons to toggle their status. Locked fields cannot be edited.



Engraving Enhancements

> **NOTE**
> The Engraving toolpath is not available for Mill-Turn.

The Engrave toolpath has been improved to address many long-standing issues. Due to these changes, the toolpath motion may look different or the number of retracts may change but the end result will be better than previous releases. For more information on the changes made to how certain parameters function, refer to the Mastercam Help.

3D Enhancements

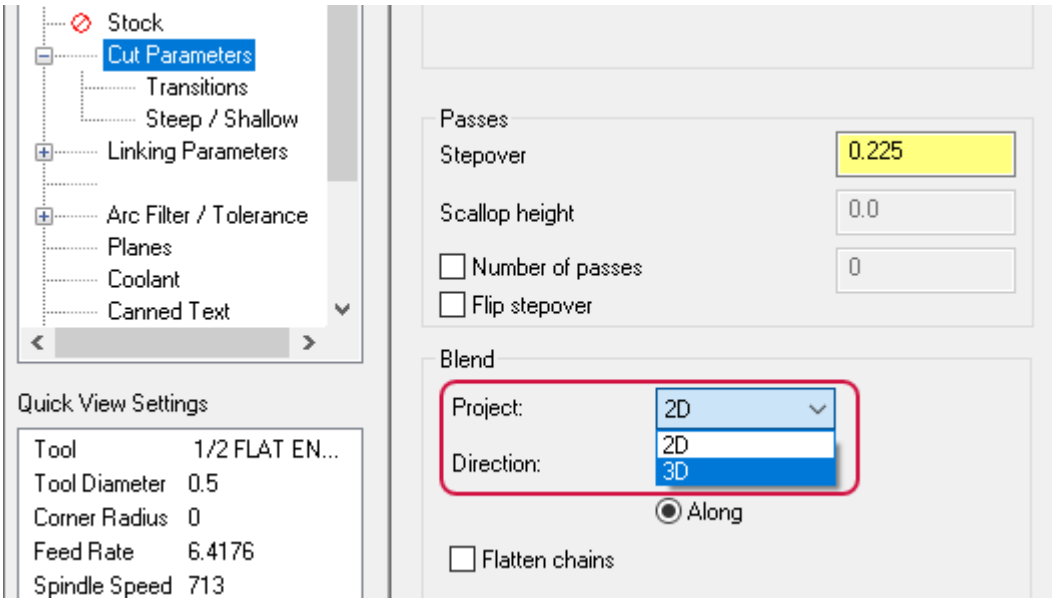
Listed below are enhancements made to the 3D toolpaths.

Blend Enhancements

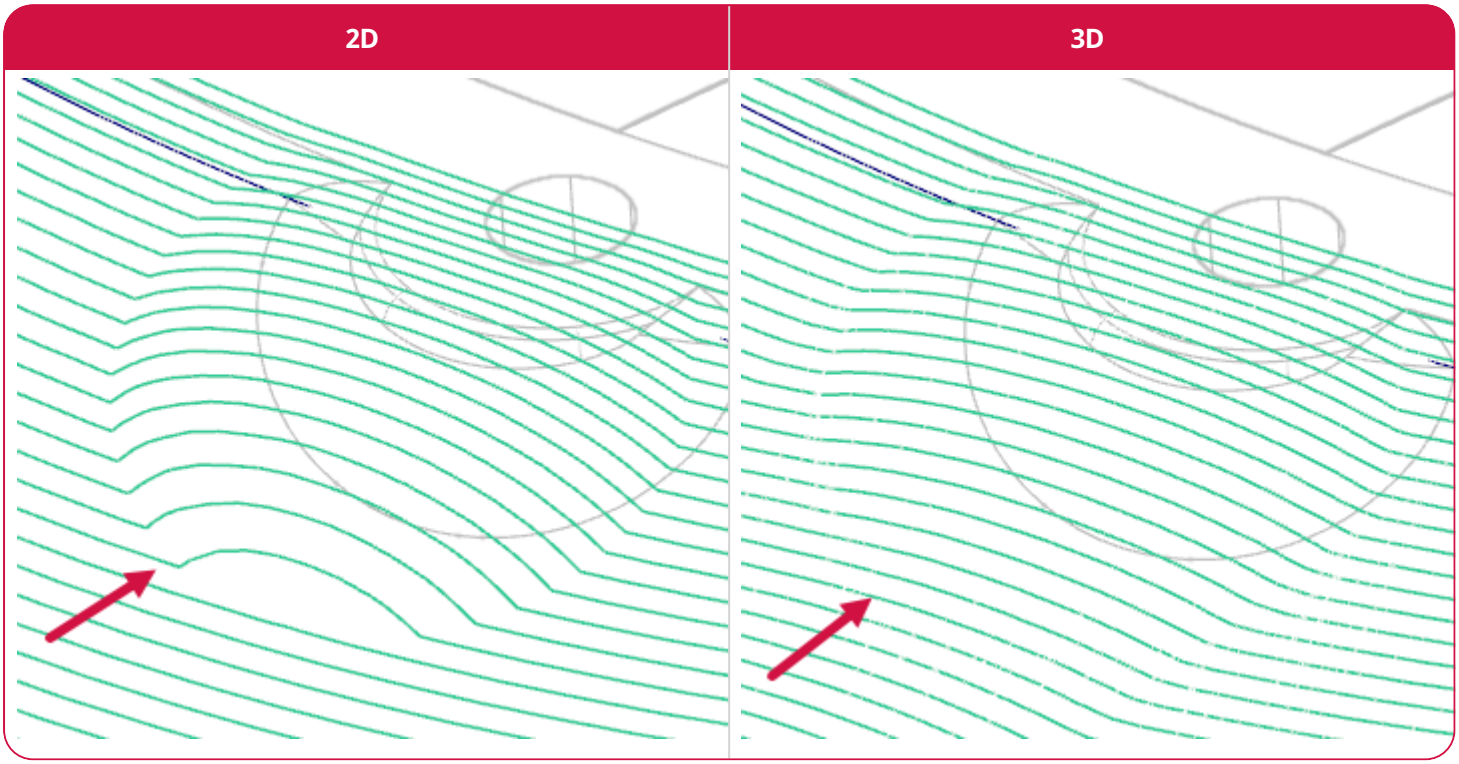
The 3D High Speed Blend toolpath now includes the following enhancements.

Creating Equidistant Passes

The **3D** option in the **Project** drop-down on the **Cut Parameters** page is now available.

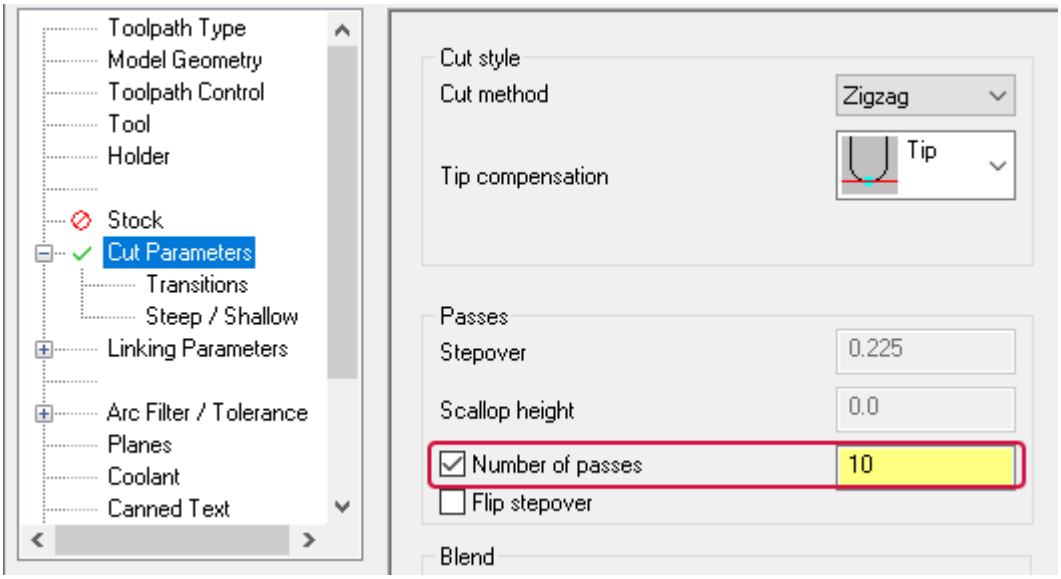


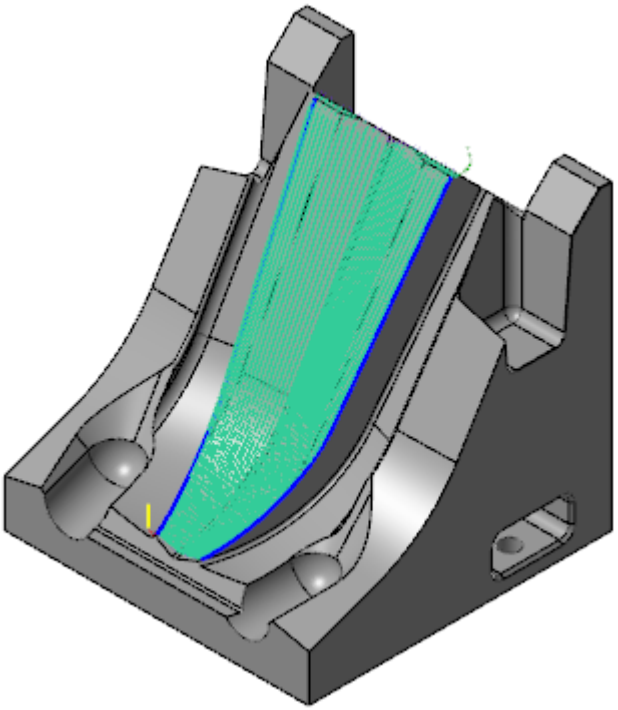
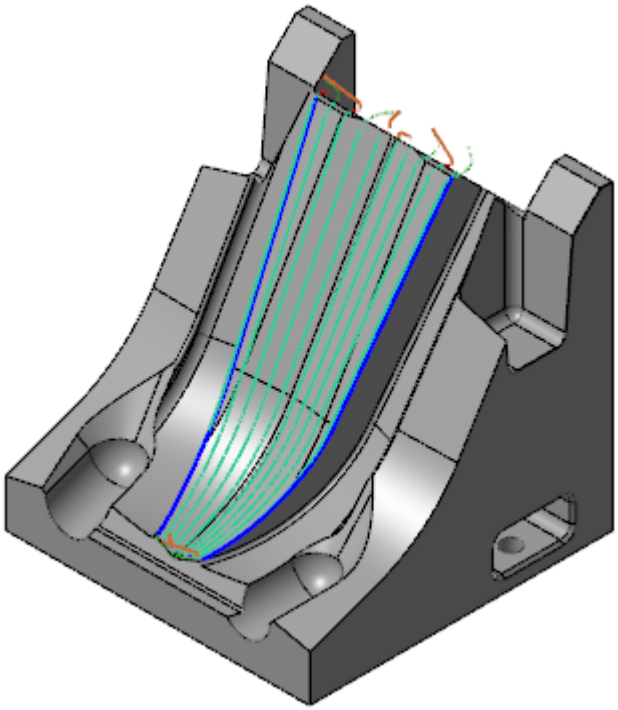
When selected, Mastercam keeps the created passes equidistant in 3D, adding cuts in steep areas.



Setting Exact Number of Cutting Passes

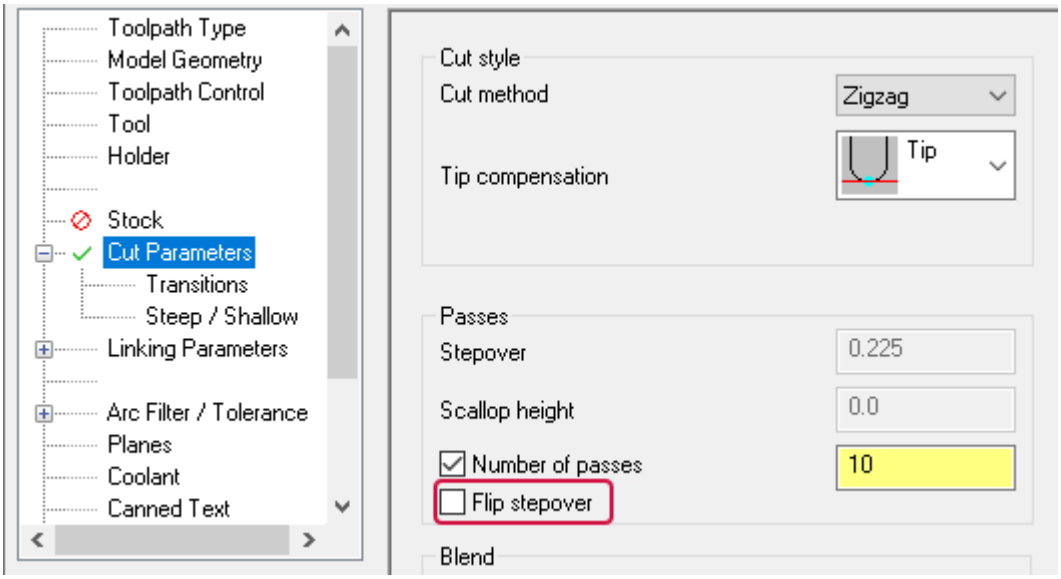
The new **Number of passes** checkbox, located on the **Cut Parameters** page, lets you set the exact number of cutting passes.



Number of Passes Off	Number of Passes On (Set to 10)
	

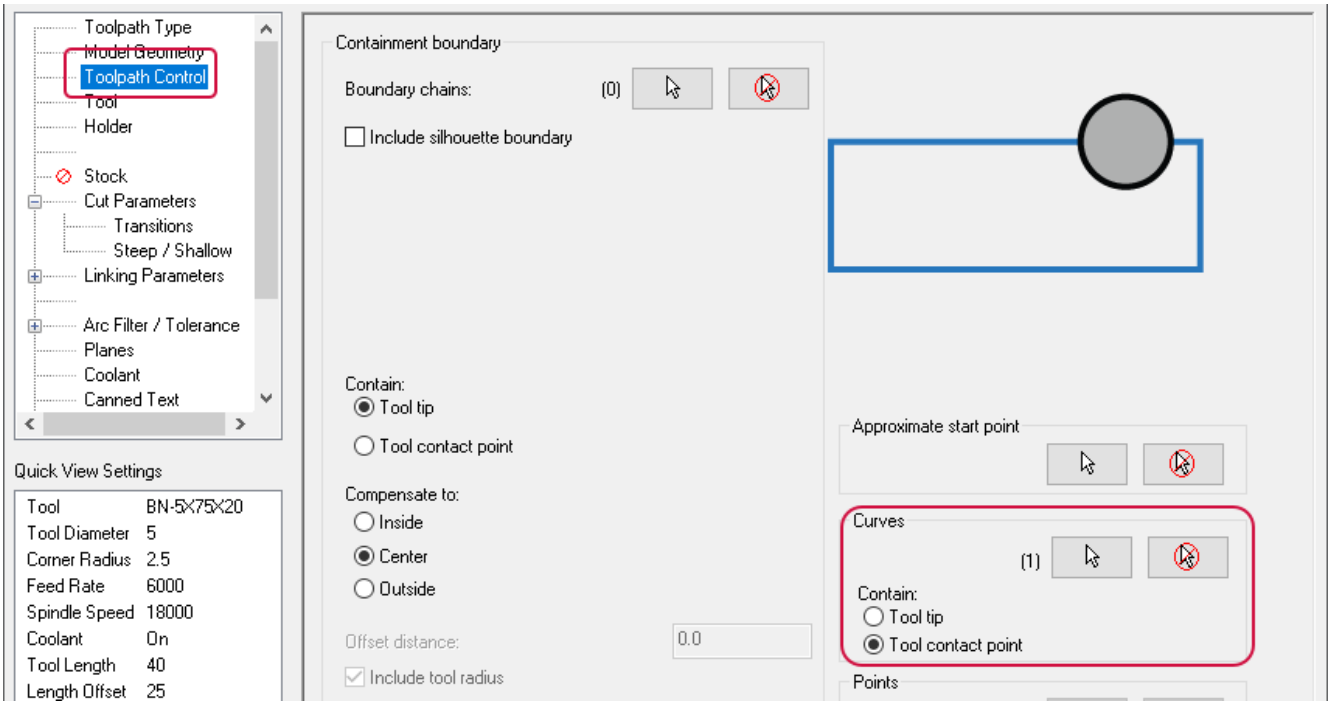
Reversing the Cutting Direction

The new **Flip stepover** checkbox, located on the **Cut Parameters** page, reverses the cutting direction of the Blend toolpath.

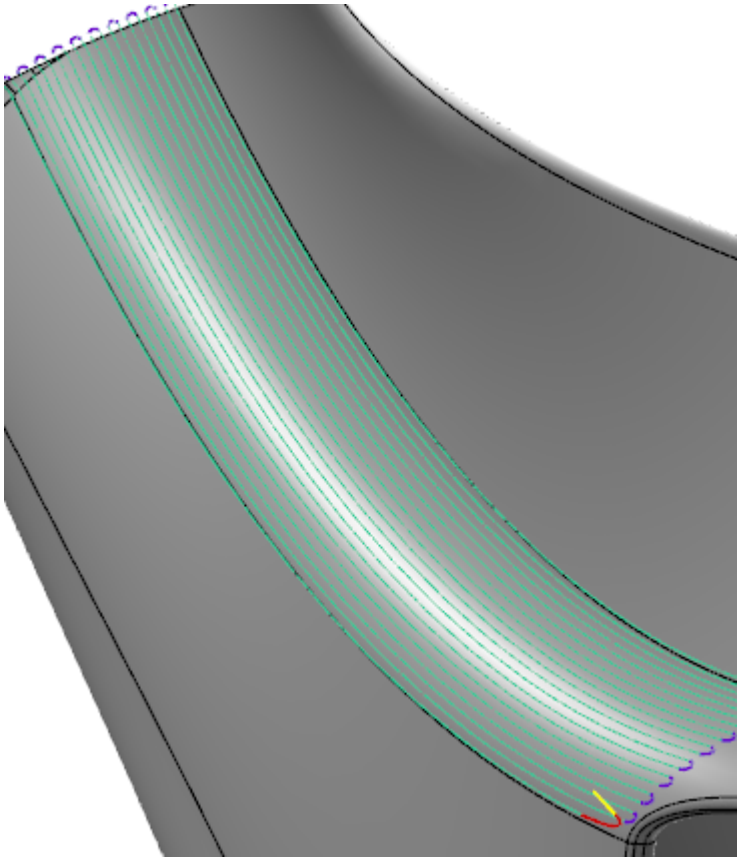


Confining Curves to Tool Tip or Tool Contact Point

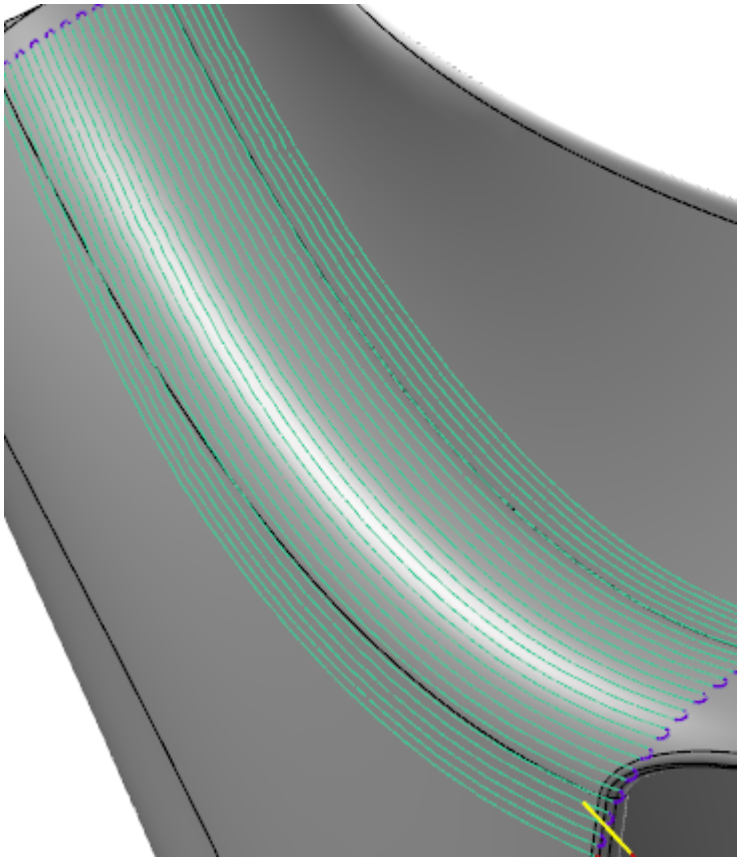
You can now confine selected **Curves** on the **Toolpath Control** page to either the **Tool tip** or **Tool contact point**.



- **Tool tip**: Confines the tool center inside the selected curves.

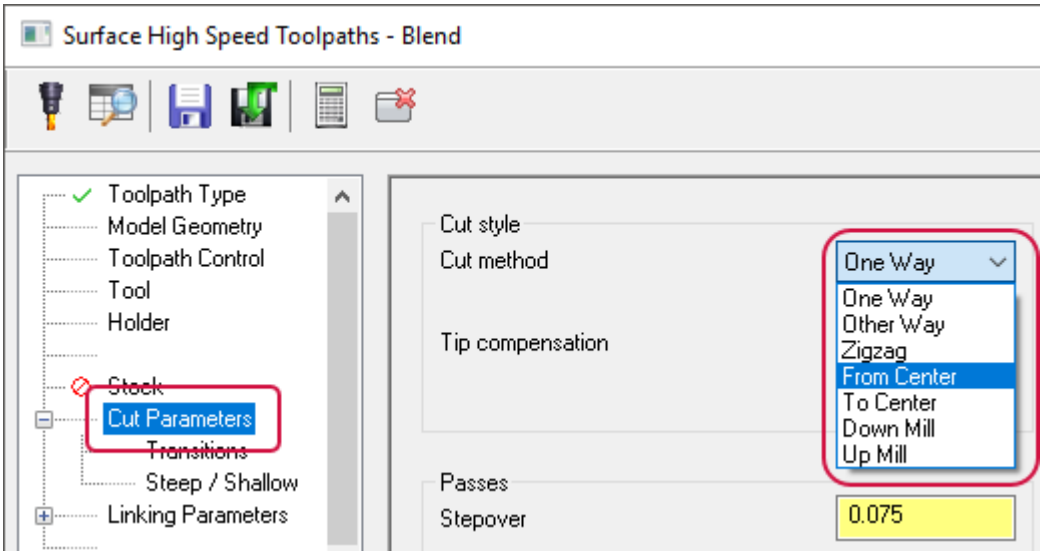


- **Tool contact point**: Confines the tool contact points inside the selected curves. The tool center may run outside the selected curves, but the contact point of the tool will not.



Machining From Center and To Center

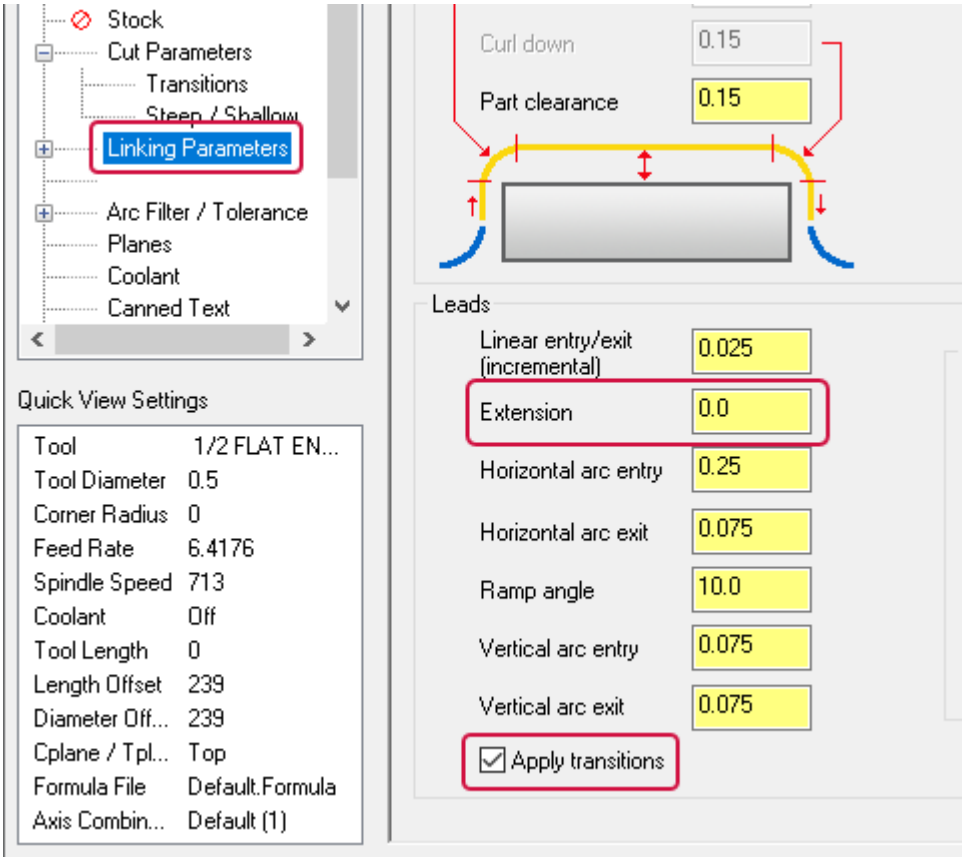
Two new methods are now available in the **Cut method** drop-down on the **Cut Parameters** page.



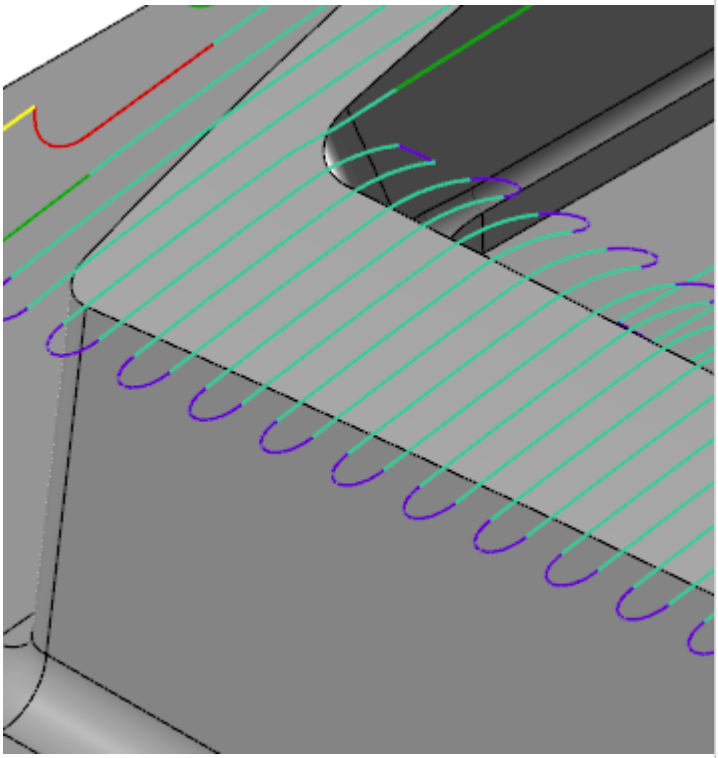
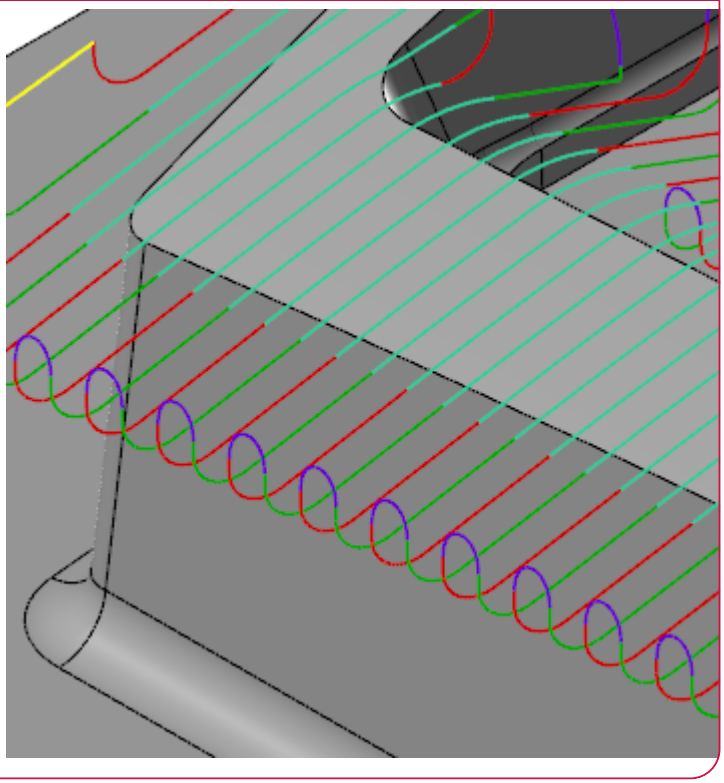
- **From center**: Cuts one direction, steps over to the next cut, and cuts in the opposite direction starting from the center and working its way outside while maintaining a climb cut.
- **To center**: Cuts one direction, steps over to the next cut, and cuts in the opposite direction starting from the outside and working its way to the center while maintaining a climb cut.

Enhancing Leads and Transition Functionality

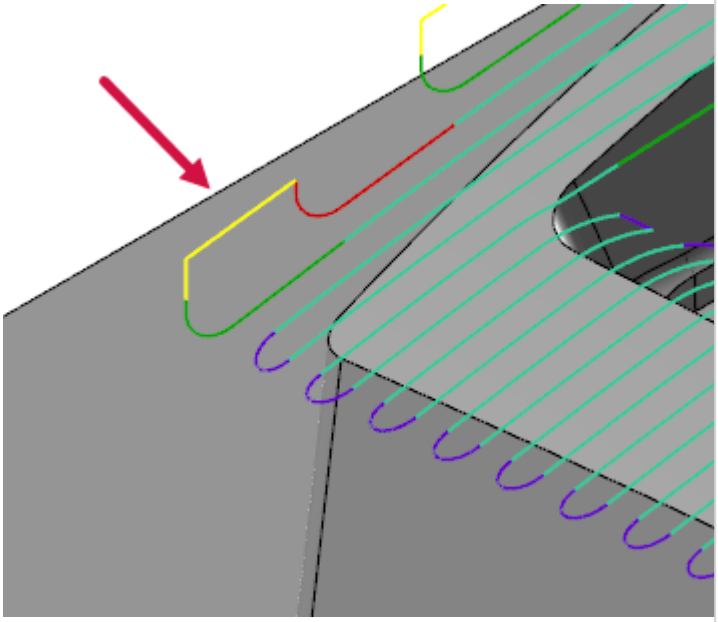
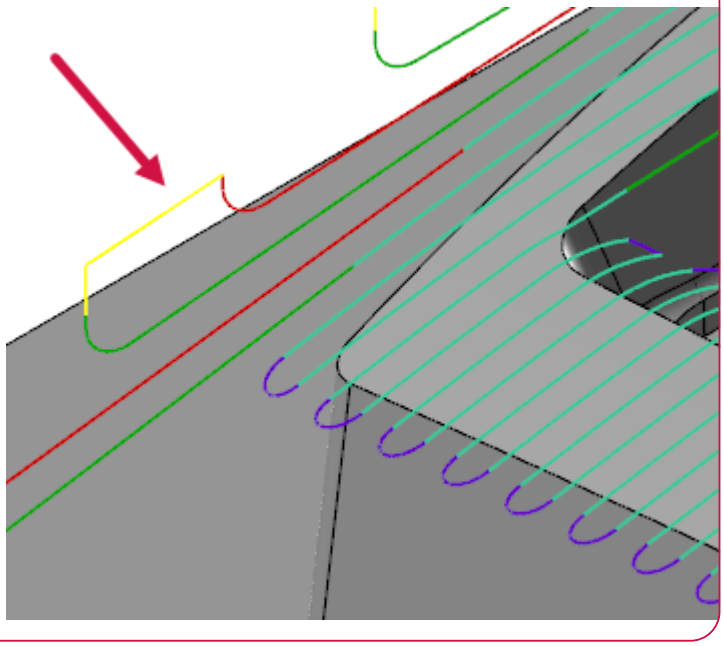
The **Linking Parameters** page for 3D toolpaths now includes two new options, **Apply transitions** and **Extension**.



Apply transitions, when selected, sets retract motions to be transitions motions. The new transition motion uses the values set on the **Transitions** page. **Apply transitions** is not available for Horizontal Area, Area Roughing, and Dynamic OptiRough toolpaths.

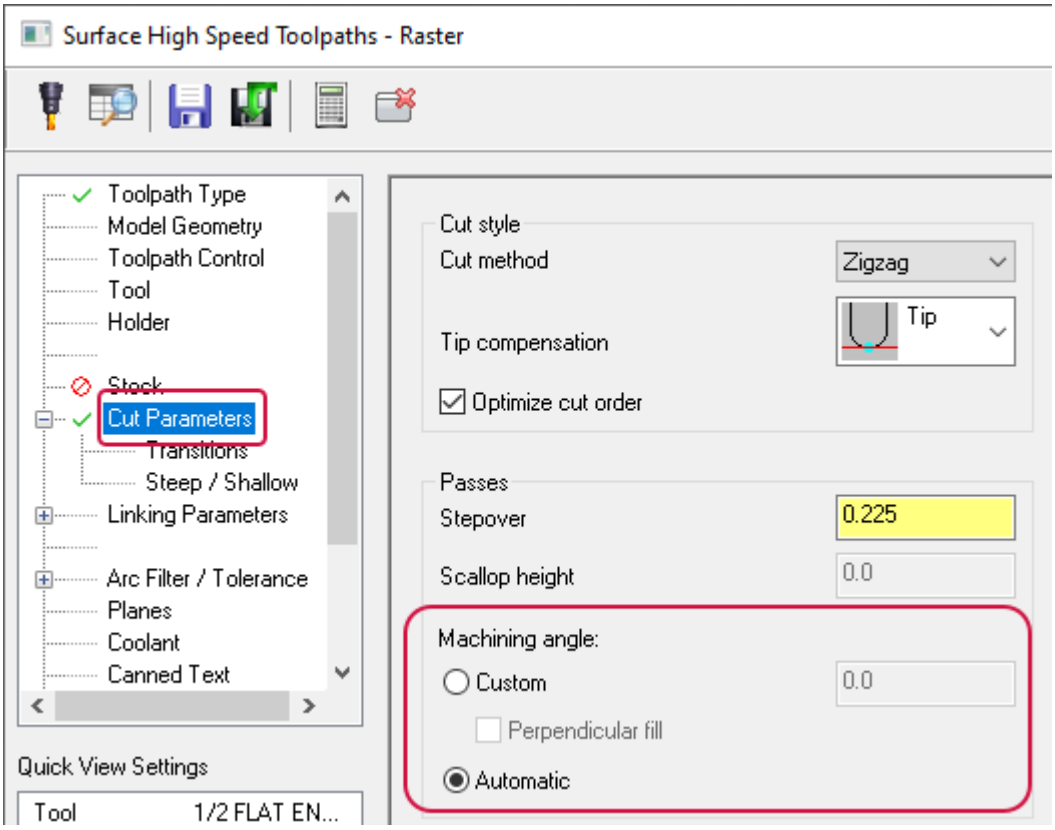
Apply transitions deselected	Apply transitions selected
	

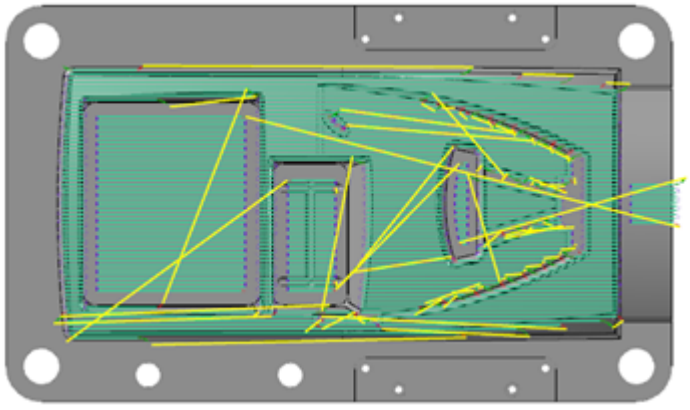
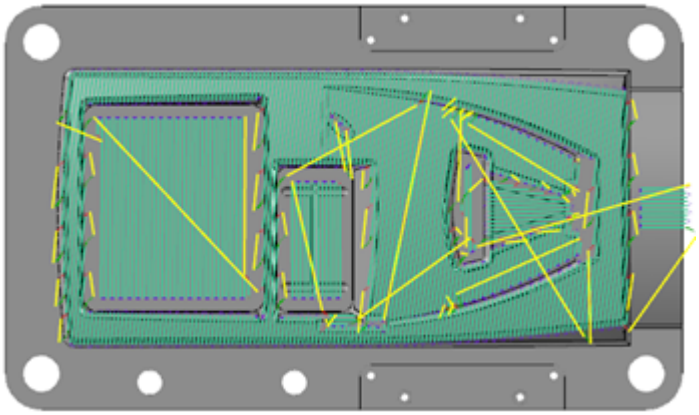
The **Extension** parameter extends the lead motion by the entered amount, using a tangential line.

Extension set to 0.25	Extension set to 1.0
	

Applying Automatic and Custom Angles to Raster Toolpaths

You now have an option to have Mastercam automatically set different angles to maximize the length of the cut pattern and/or minimize the connecting moves of a Raster toolpath. This option is on the **Cut Parameters** page.



Machining angle (Custom, set to 0)	Machining angle (Automatic)
	

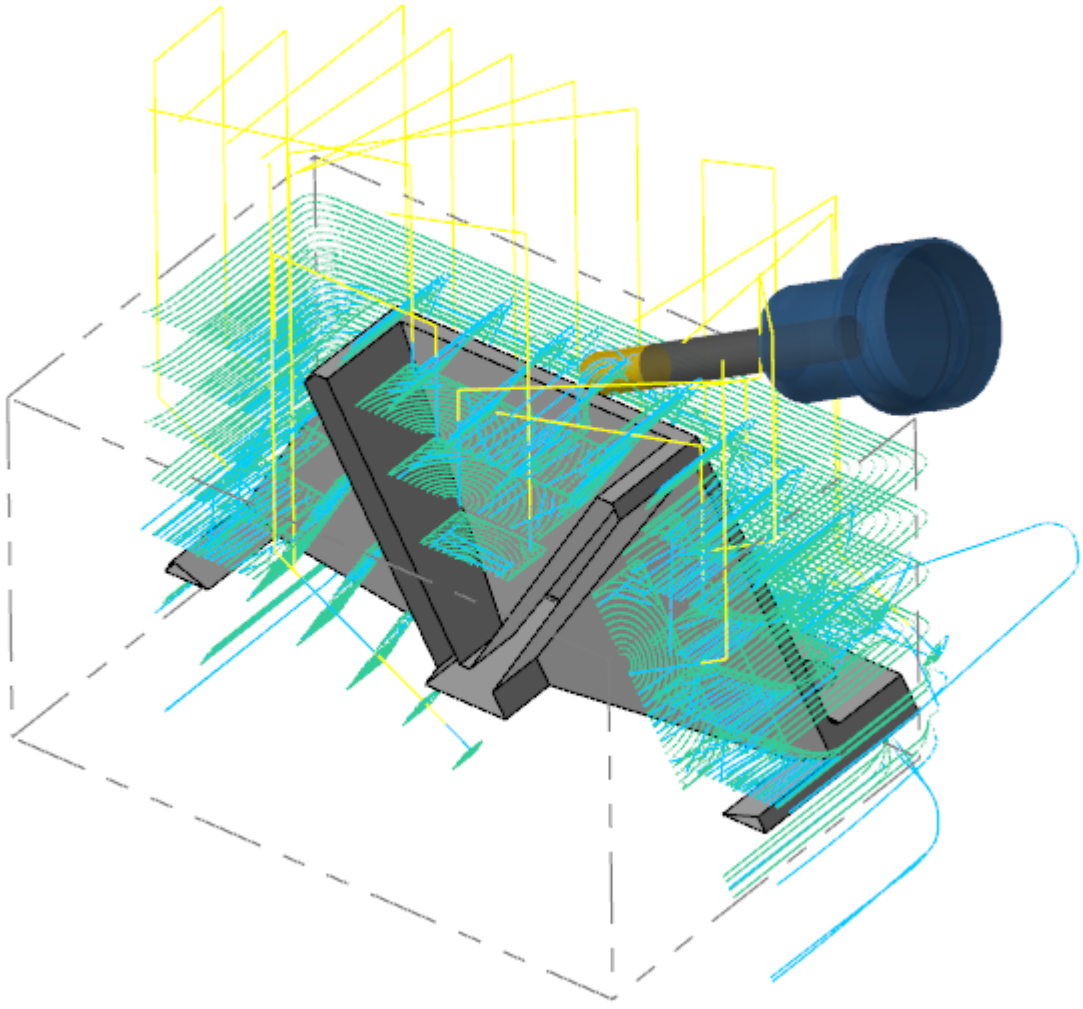
Multiaxis Enhancements

Listed below are enhancements made to Multiaxis toolpaths.

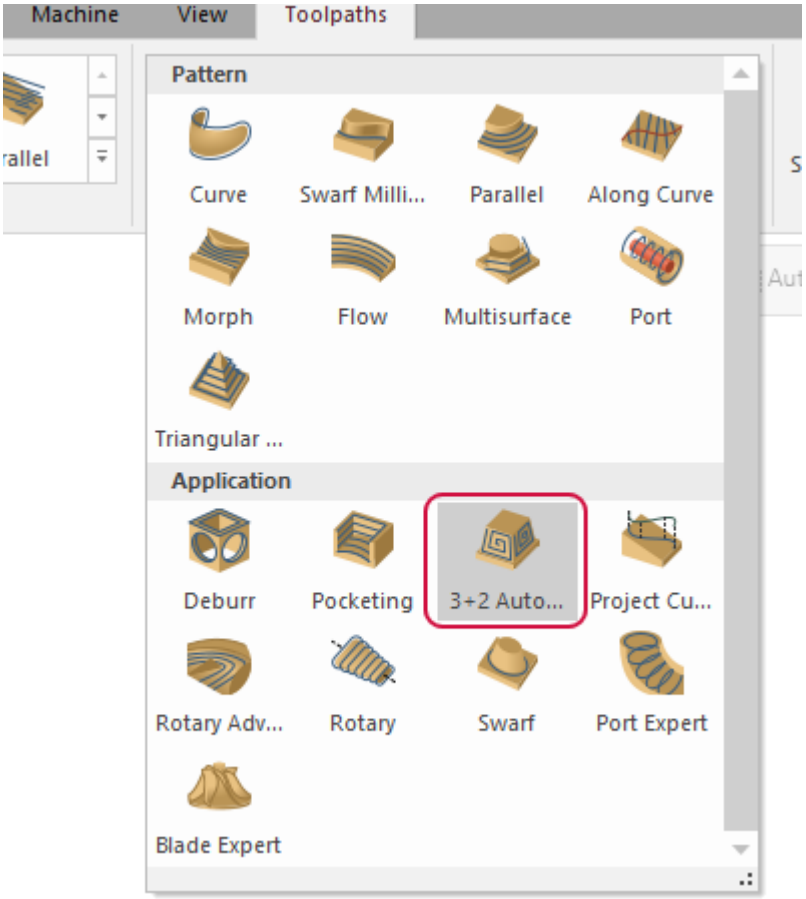
Machining with The New 3+2 Automatic Roughing Toolpath

The new 3+2 Automatic Roughing toolpath automatically makes multiplane 3-axis toolpaths for roughing. Mastercam analyzes the model and stock, and then creates a roughing toolpath. Then Mastercam calculates the remaining stock and computes a new toolpath. This continues until only a defined amount of stock remains. The results are all contained within the one toolpath.

The image below shows the new toolpath on a part with **Section view** enabled, allowing you to better see the toolpath.

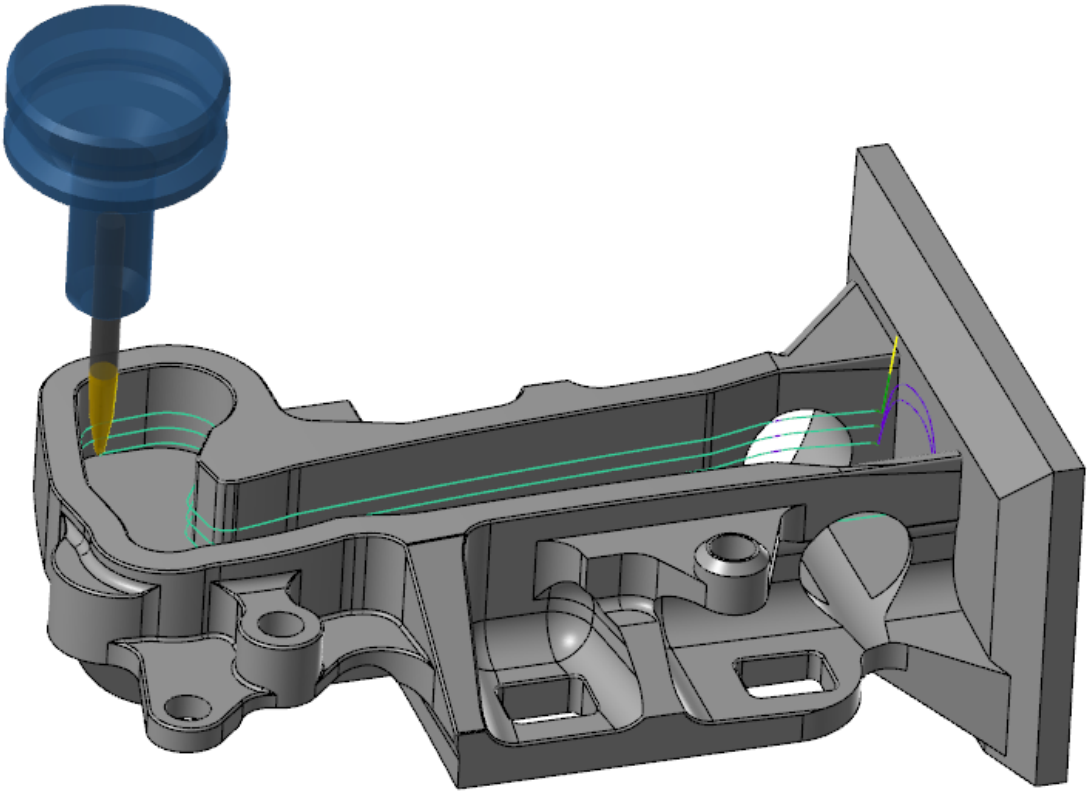


3+2 Automatic Roughing is located in the **Multiaxis** gallery on the **Mill Toolpaths** contextual tab.

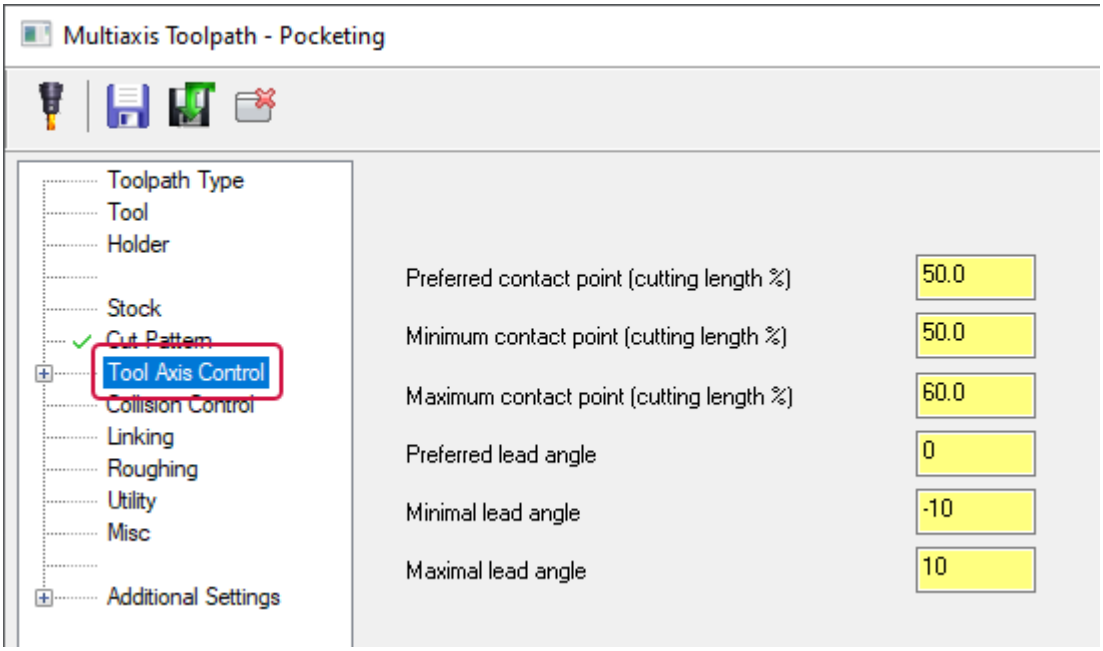


Machining with the Enhanced and Renamed Multiaxis Roughing Toolpath

The Multiaxis Roughing toolpath has been enhanced and renamed to Multiaxis Pocketing. This revised toolpath includes new options such as undercut roughing, wall finishing, and floor finishing.

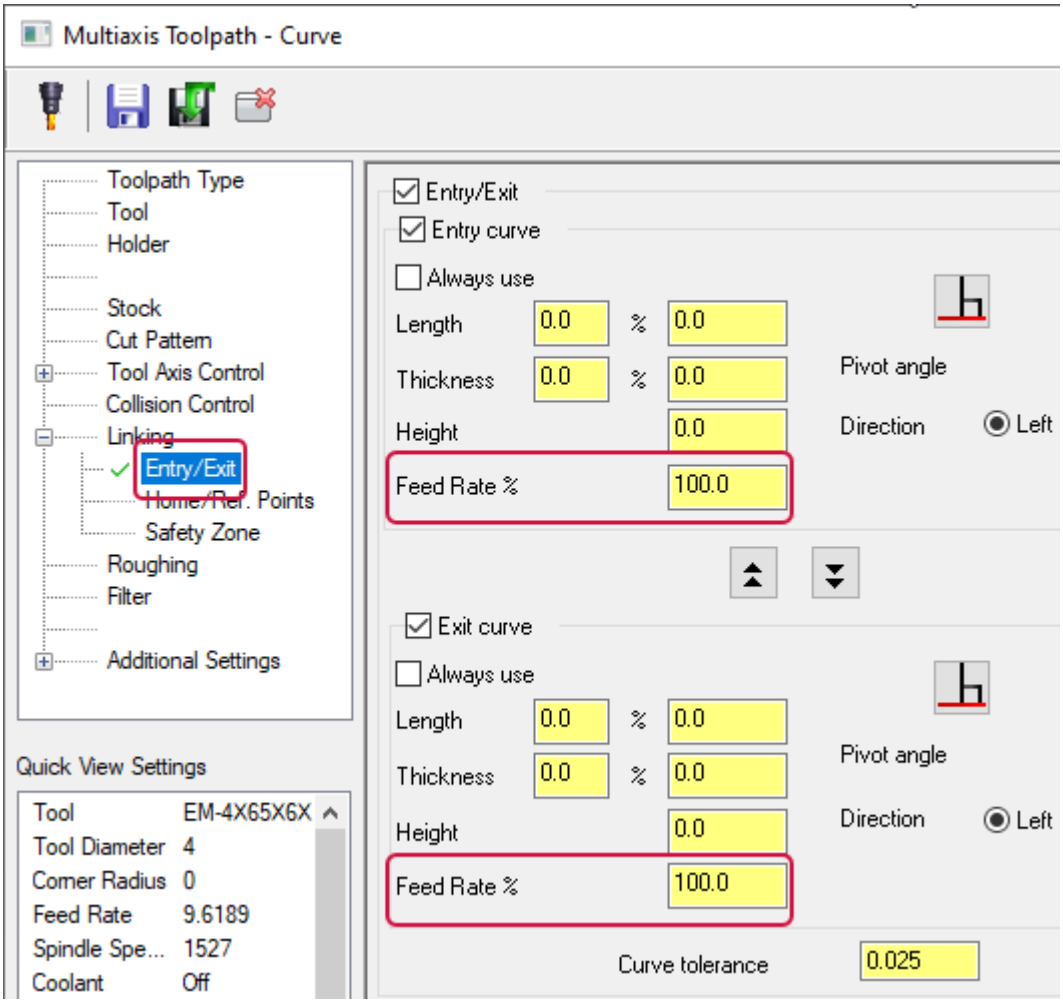


Multiaxis Pocketing provides greater control of the tool contact point and machining angle when using Accelerated Finishing™ tools. These options ensure that you get the best possible motion from these tools.



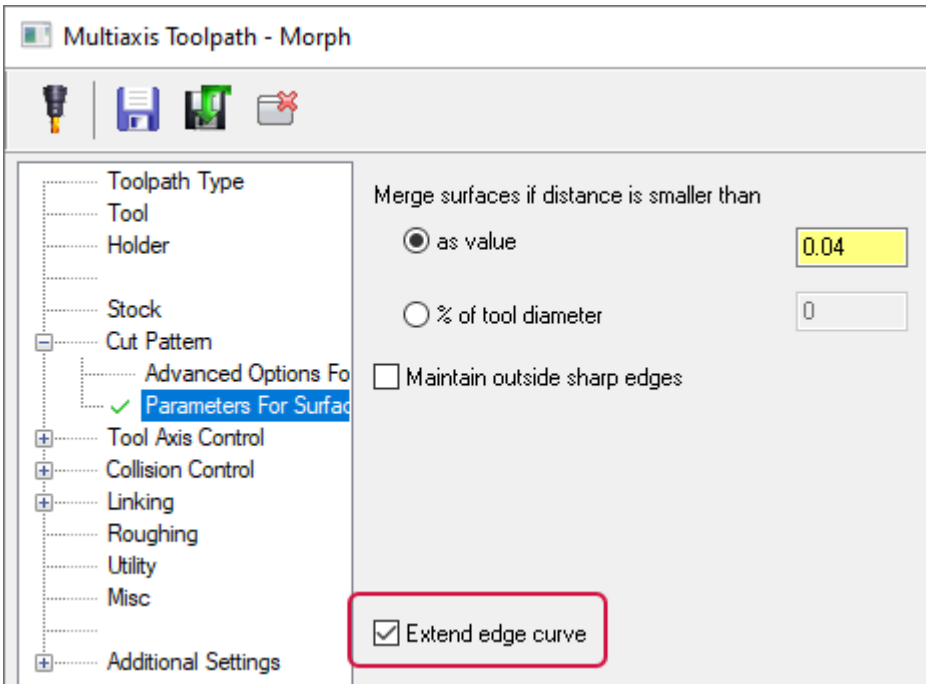
Setting Separate Feed Rates for Entry and Exit Linking Moves

The Curve, Flow, Multisurface, Swarf, and Port toolpaths now have separate feed rate control options for entry and exit linking moves. These controls are on the **Entry/Exit** page, under the **Linking** page.

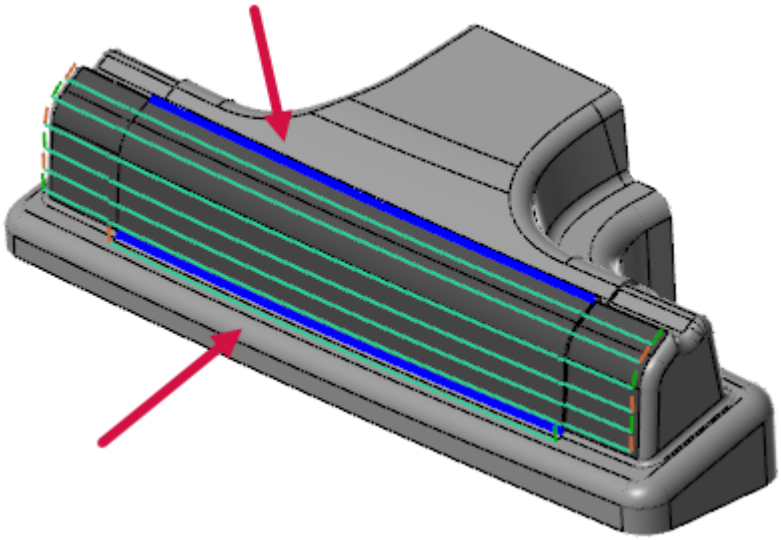


Extending Edge Curves for Morph Toolpaths

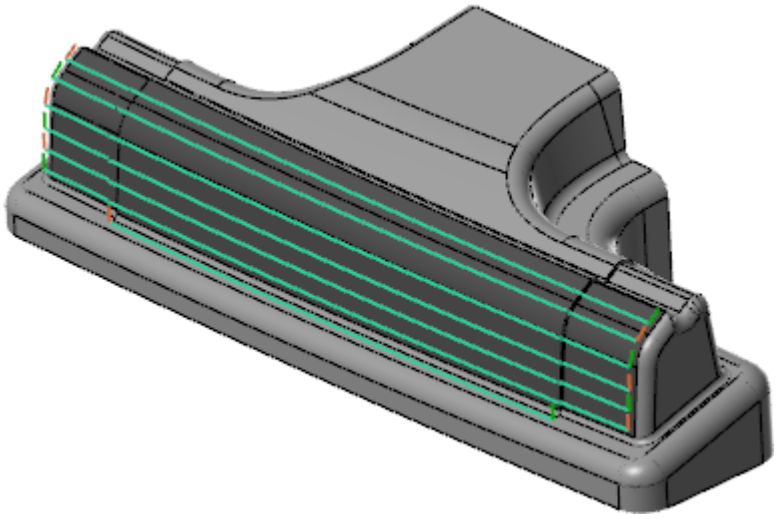
We have added the **Extend edge curve** option to the Morph toolpath to give you better control over your toolpath results. By using this option, you can get a different toolpath without drawing additional geometry. The **Extend edge curve** option is on the **Parameters for Surface Edge Handling** page.



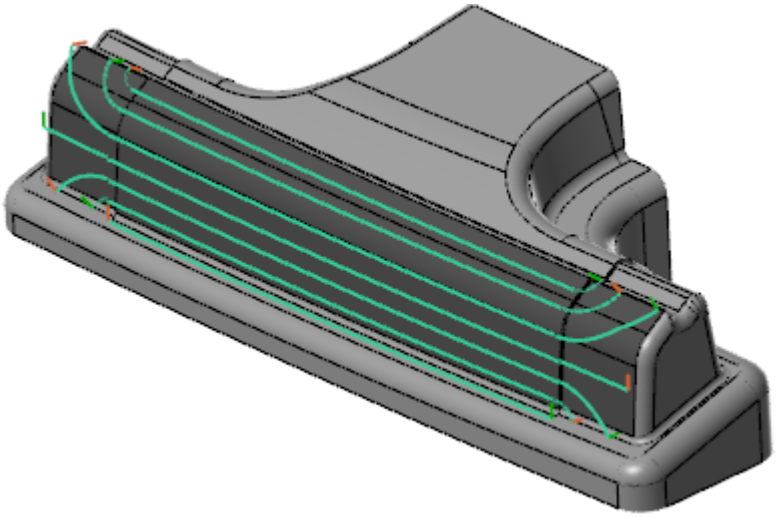
In previous versions of Mastercam, you needed to add wireframe and use it to generate the Morph toolpath.



In Mastercam 2021, you no longer have to add wireframe. Selecting **Extend edge curves** creates the following toolpath.



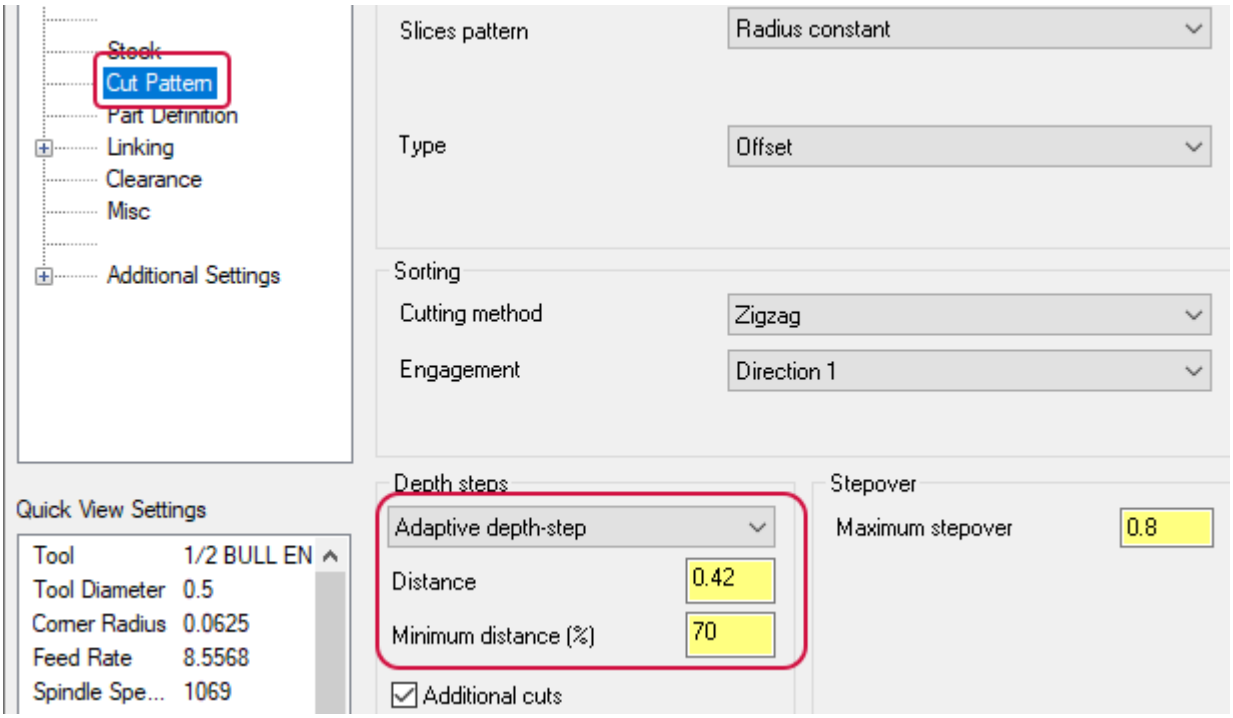
When deselected, the toolpath is created as shown below.



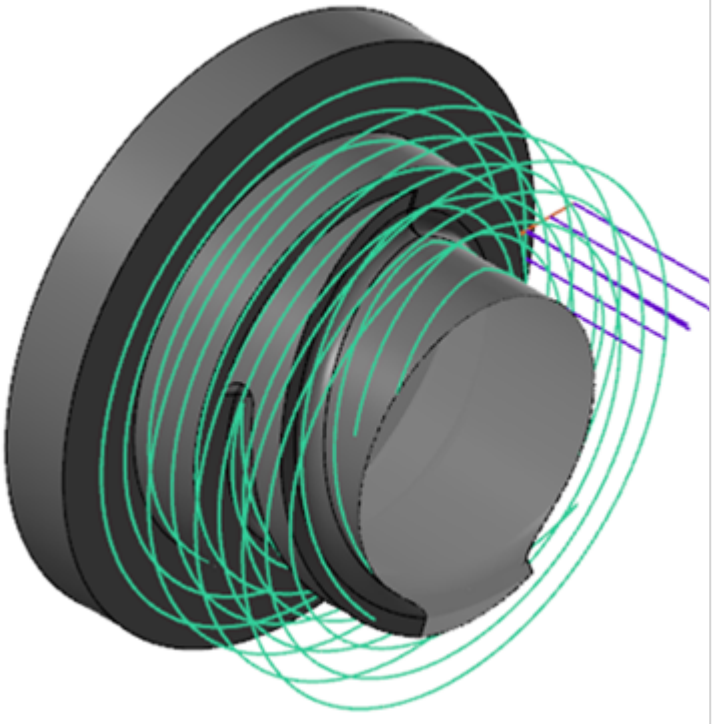
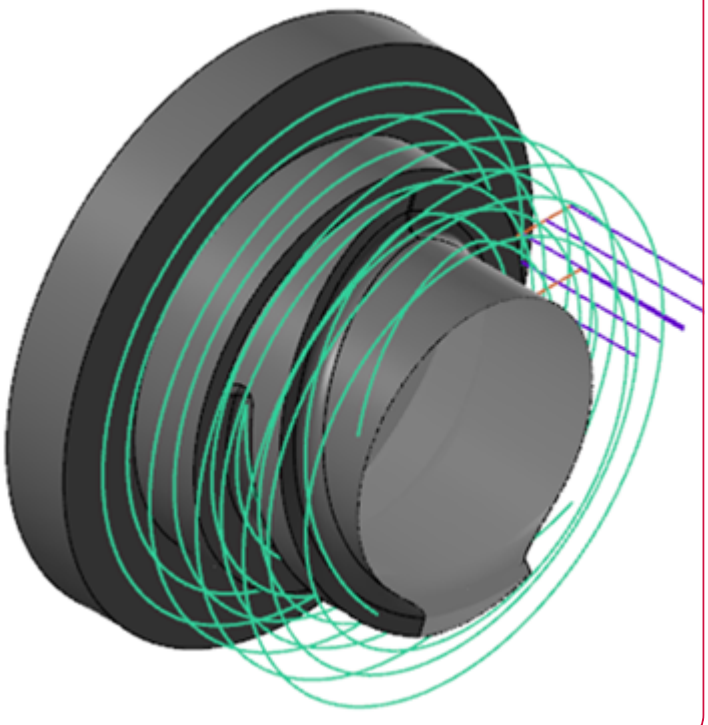
Rotary Advanced Enhancements

Creating Constant Z Cuts

A new option, **Adaptive depth-step**, has been added to the **Cut Pattern** page for the Rotary Advanced toolpath.

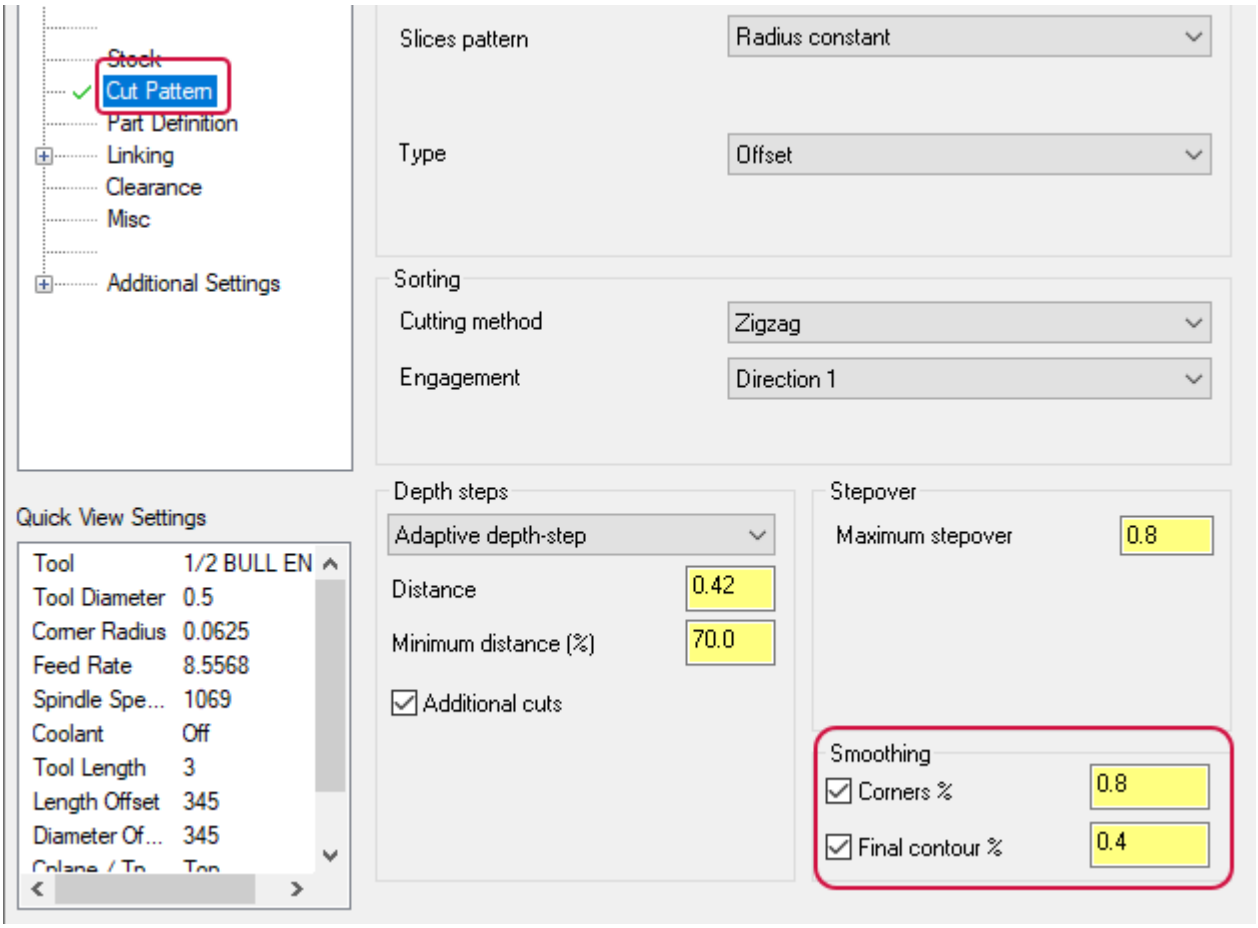


Adaptive depth-step creates constant Z cuts within the defined **Distance** and **Minimum distance**. **Constant depth-step** creates constant Z cuts.

Adaptive depth step	Constant depth step
	

Filleting Sharp Corners

The **Cut Pattern** page now includes smoothing options. Use the **Corners percentage** option to fillet in the sharp corners of inner contours, specifying the radius of the fillet as a percentage of the stepover distance. The **Final contour percentage** option does the same for sharper corners of the outer contours.



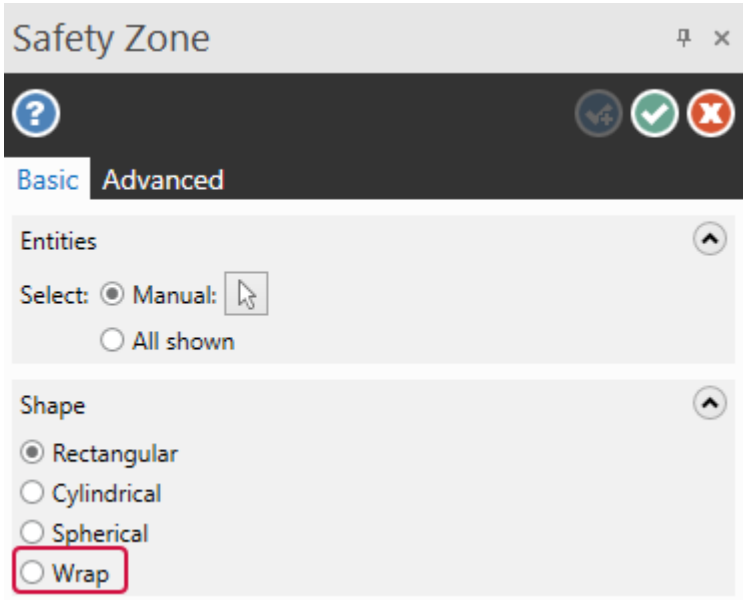
Improved Toolpath Processing Time and Surface Accuracy

In Mastercam 2021, the Morph, Parallel, Along Curve, and Project Curve toolpaths now run smoother and faster. Core functionality of these toolpaths was overhauled to hold better surface accuracy and normal resolution, which translates into more stable vectors coming from the toolpath engine.

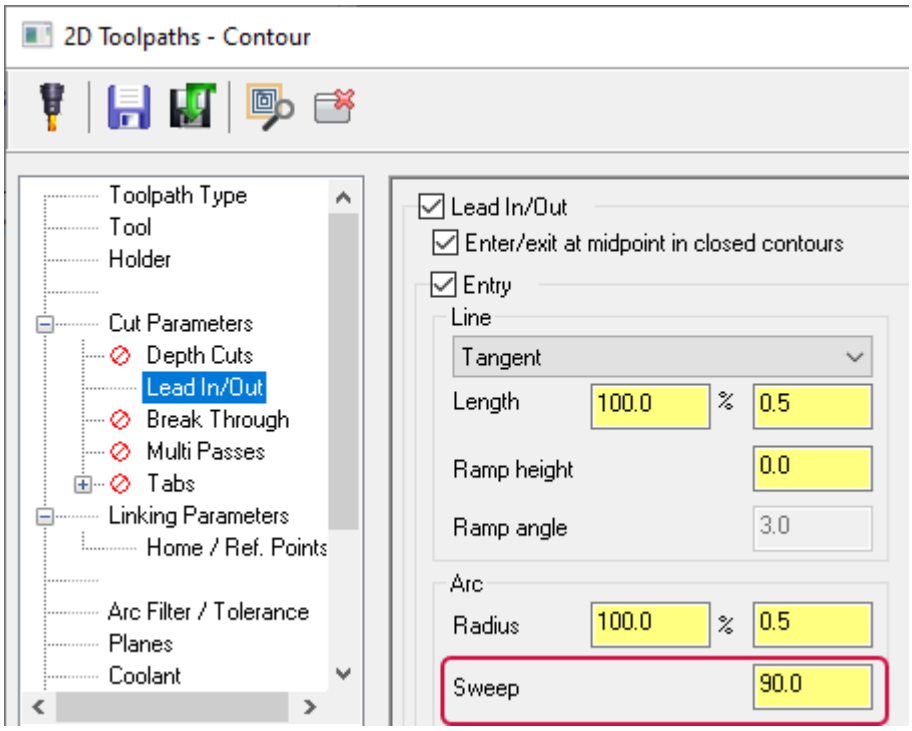
There have been improvements of up to 40% better cut times on some machines, with an average of about 20% on part testing. This should yield better surface finishes across the board, in addition to cut time improvements. There is no toolpath processing time increase or other negative changes associated with this improvement. You do not need to make changes manually. Simply regenerate the toolpath in Mastercam 2021.

Miscellaneous Enhancements: Mill

- Improved performance for the Area Roughing toolpath.
- Safety Zone also includes the **Wrap** option, which creates an irregularly shaped boundary that closely fits the selected entities. For further information, refer to "Wrapping Entities with Bounding Box" on page 100.



- The **Sweep** parameter on the **Lead In/Lead Out** page for 2D toolpaths now accepts negative values.



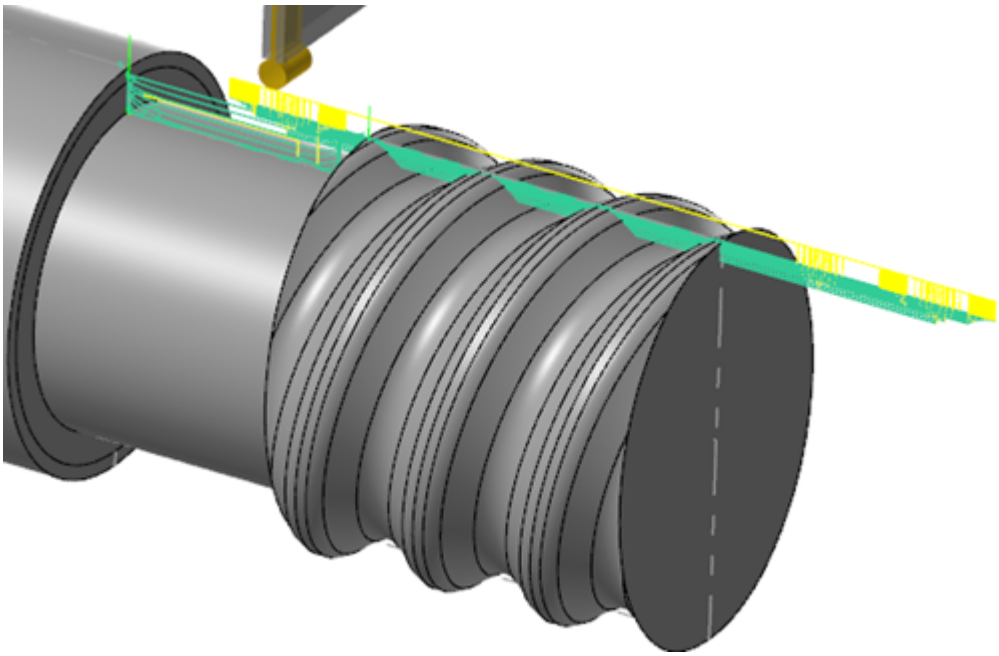
TURNING

Listed below are major enhancements made to the Lathe and Mill-Turn products.

> **NOTE**
> Unless otherwise stated, the new features and functionality listed in this section apply for both Lathe and Mill-Turn licenses.

Machining Custom Thread Forms

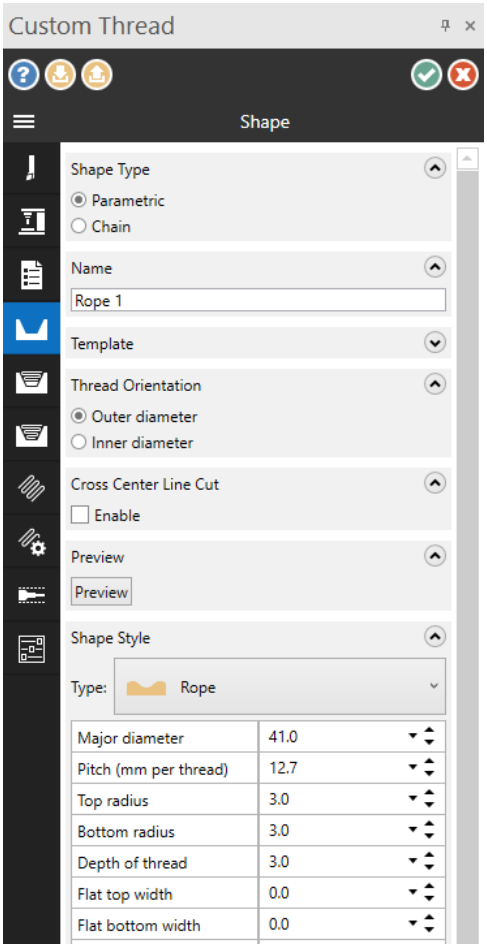
Mastercam 2021 introduces the new Lathe Custom Thread toolpath to support custom thread forms.



The new toolpath also takes advantage of Mastercam's next-generation panel interface to simplify and streamline your programming workflow. You can select chained geometry to represent the thread cross-section or select one of four parametric definitions:

- Rope
- Buttress
- Square
- Trapezoidal

Parametric definitions let you define the profile by entering dimensions directly in a properties grid, eliminating the need to chain geometry.



Mastercam lets you define roughing and finishing passes with separate cutting parameters.

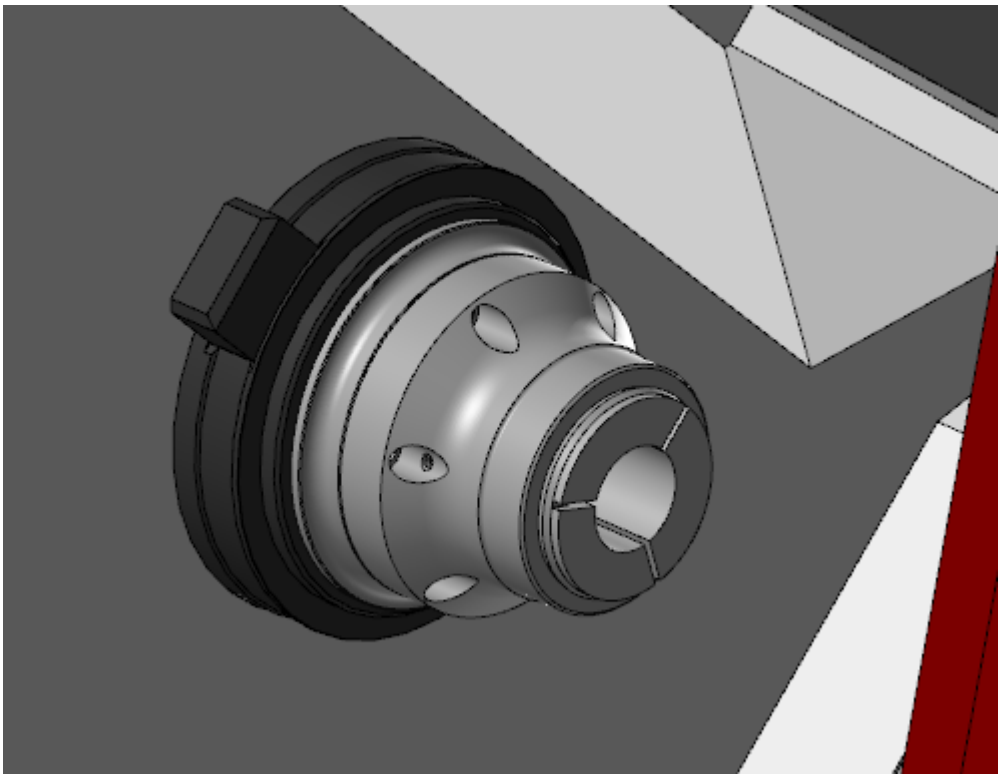
The new custom thread module also supports Mastercam's other advanced threading and toolpath features, such as multi-start threads, tool inspection, ID/OD threads, allowances for mating parts, and cross-centerline turning. You can even name and save custom thread forms so you can easily retrieve and reuse them in future operations.

Work-Holding Component Enhancements

Listed below are enhancements made to the work-holding components.

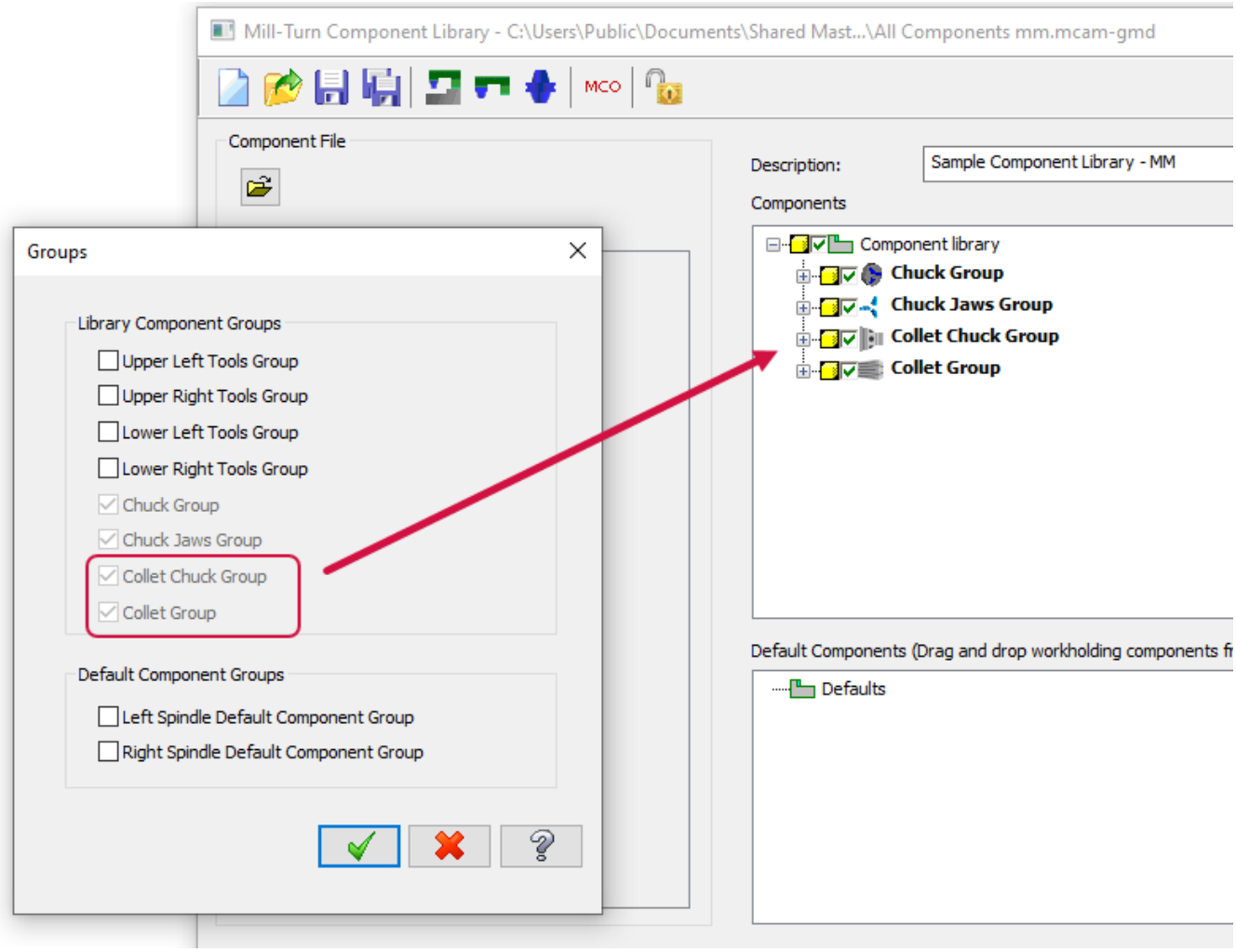
Support for Collets and Collet Chucks

Mastercam Mill-Turn now supports collet chucks and collets as individual component types. This greatly expands the range of machines that can be directly supported by Mill-Turn. The picture below shows an example of the new components.

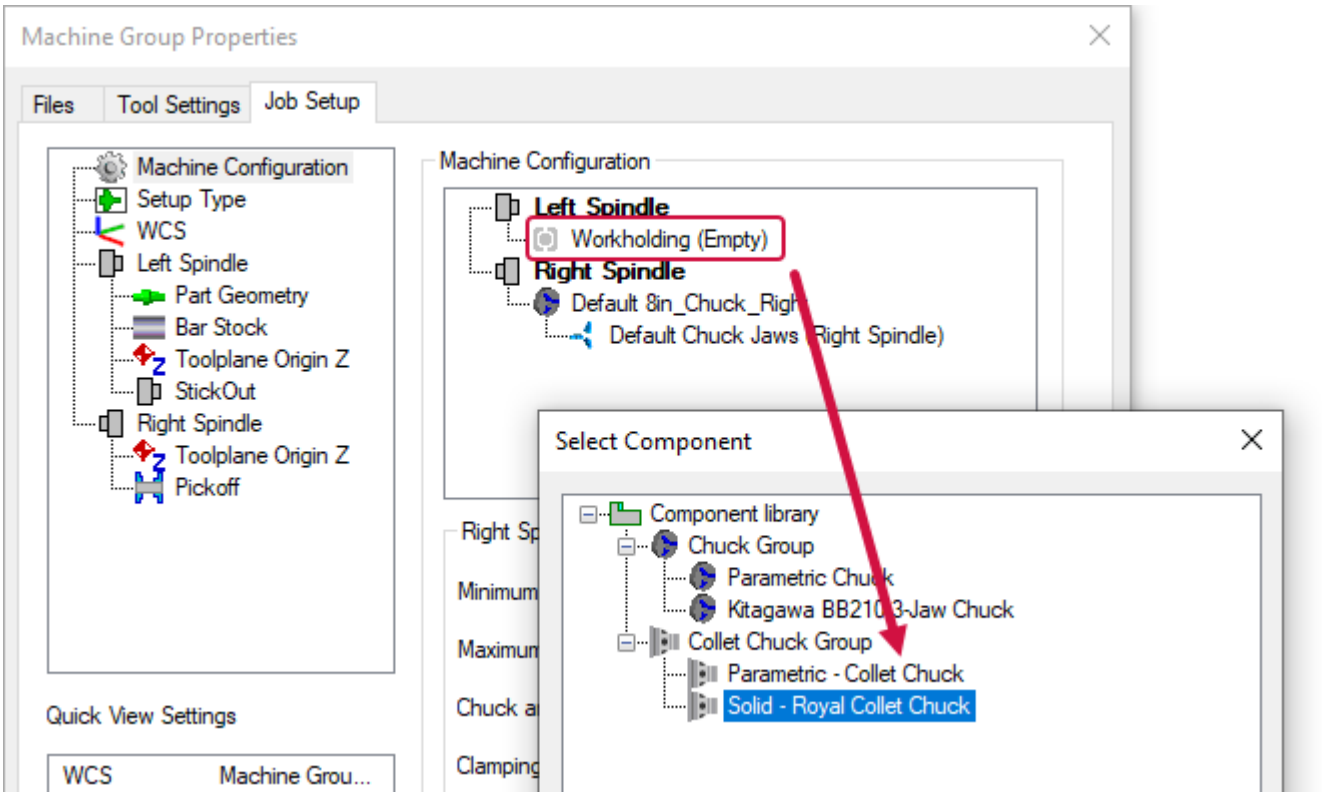


Collets and collet chucks are fully supported by Machine Simulation. They can be modeled either parametrically or you can choose a solid model of the component.

This functionality is available to all Mill-Turn users, not just machine developers. Use Mastercam's Component Library to create collets and collet chucks and organize them in libraries with your other chucks and chuck jaws. Then once you load your machine in Mastercam, simply select the desired components as part of Mill-Turn's Job Setup.



After you load your machine in Mastercam, use the **Machine Configuration** page in Job Setup to mount a collet chuck and collet on your machine. If chucks and jaws are already mounted by default, remove them before selecting a collet chuck and collet.



Improved Modeling for Chucks and Jaws

Mastercam 2021 expands the support for modeling chucks and chuck jaws. You now can define chucks and chuck jaws by selecting a solid model. Additionally, Mastercam 2020 introduced the ability to define a chuck with a chained profile. This functionality has been extended to include chuck jaws.

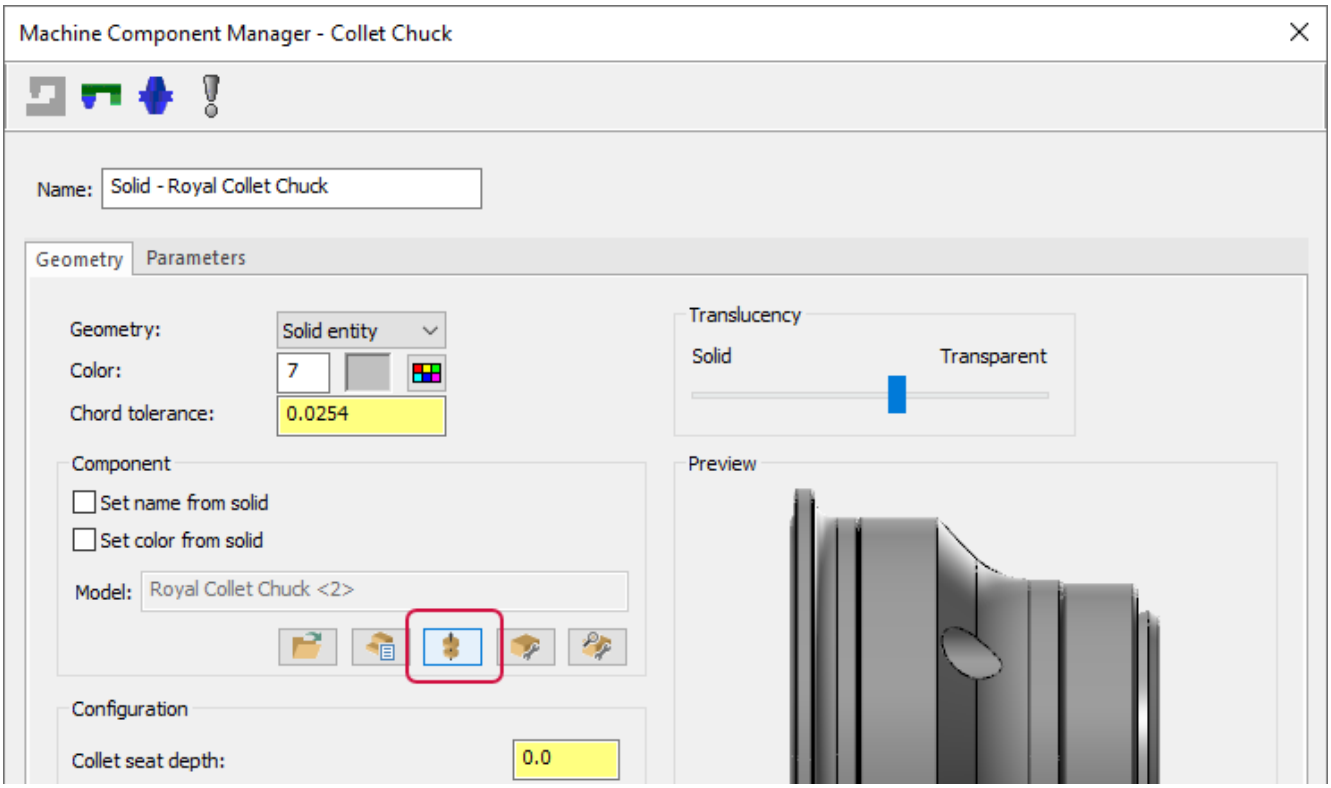
This functionality is available to both Lathe and Mill-Turn users.

- Mastercam Lathe uses the associated geometry to generate the component boundaries for chucks and jaws.
- Mastercam Mill-Turn's Machine Simulation and collision detection functions fully support the new geometry options.

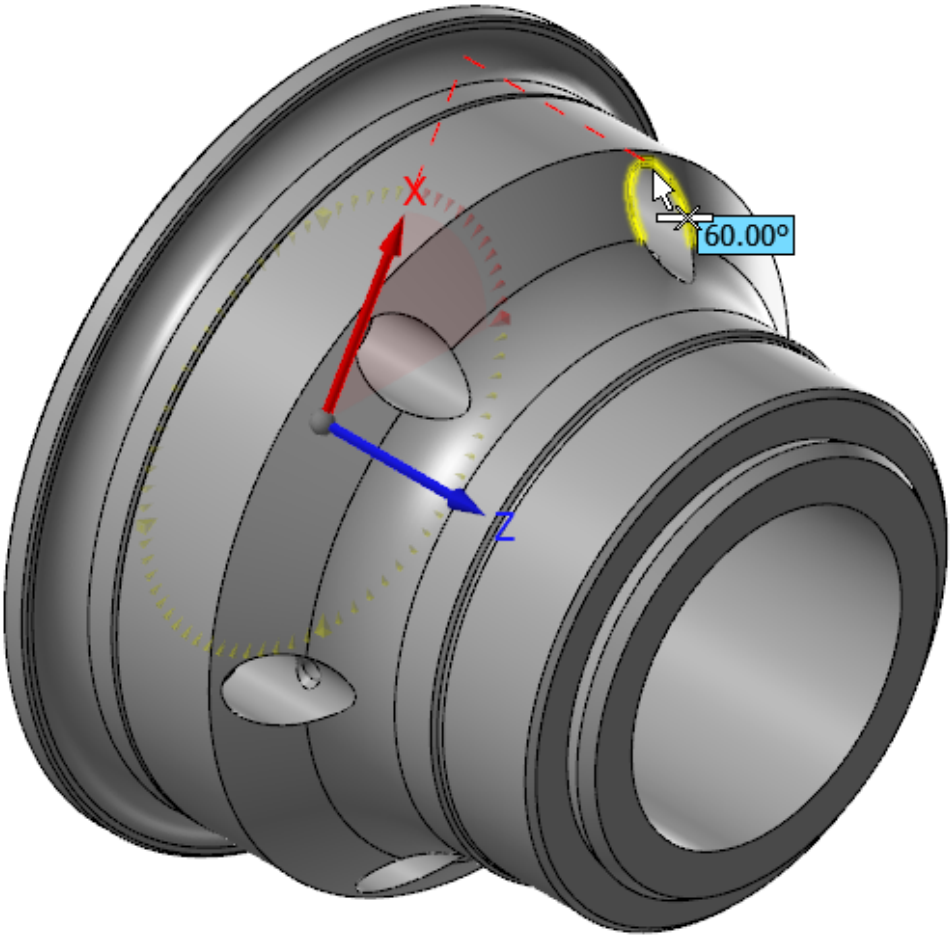
For chuck jaws that are defined by chained geometry, you can position the reference position (or clamping position) anywhere along the chain. The geometry can be drawn anywhere, it does not need to be drawn in the proper position. In addition, Mastercam will automatically flip the profile as needed so that the jaws can be used on either spindle.

The interface and workflow for working with chuck and jaws components have also been redesigned. These changes also apply to the new collet and collet chuck components.

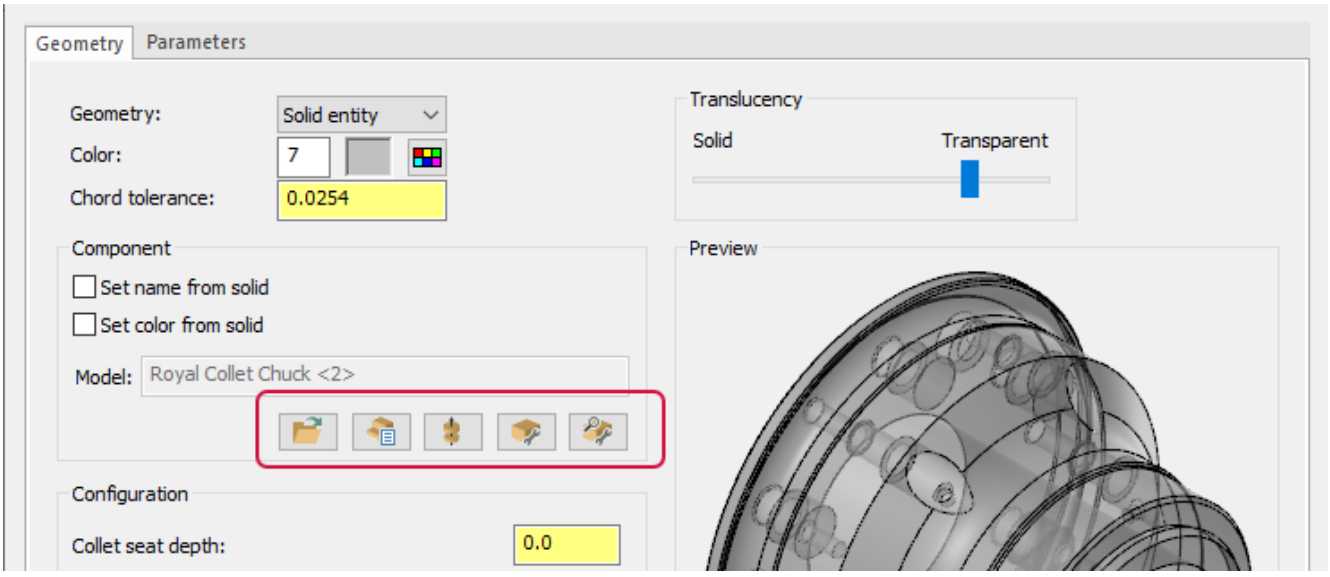
When you select a solid model, new function panels let you align and reposition the model. This means that the component model does not need to be drawn in a specific orientation or location. Select **Align component** on the **Geometry** tab. The new **Align Collet Chuck** function panel displays.



You can make the necessary changes as part of the component definition workflow. For example, in this picture, the model is rotated 60-degrees around the Z axis after it has been selected.

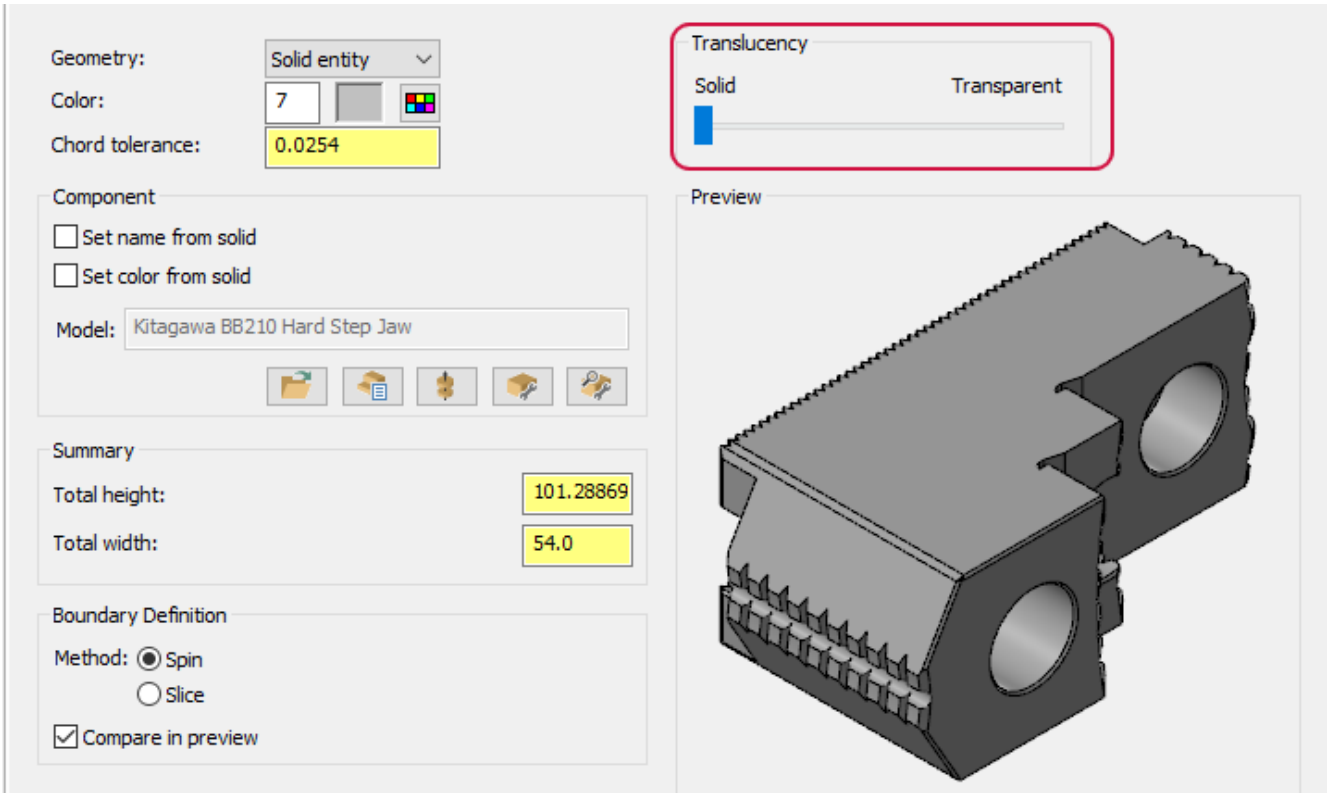


You can select a solid model from an external file or a level in your part. Additional controls let you optimize or repair the model, or return to the **Align** function panel to make changes. These new buttons are on the **Geometry** tab.

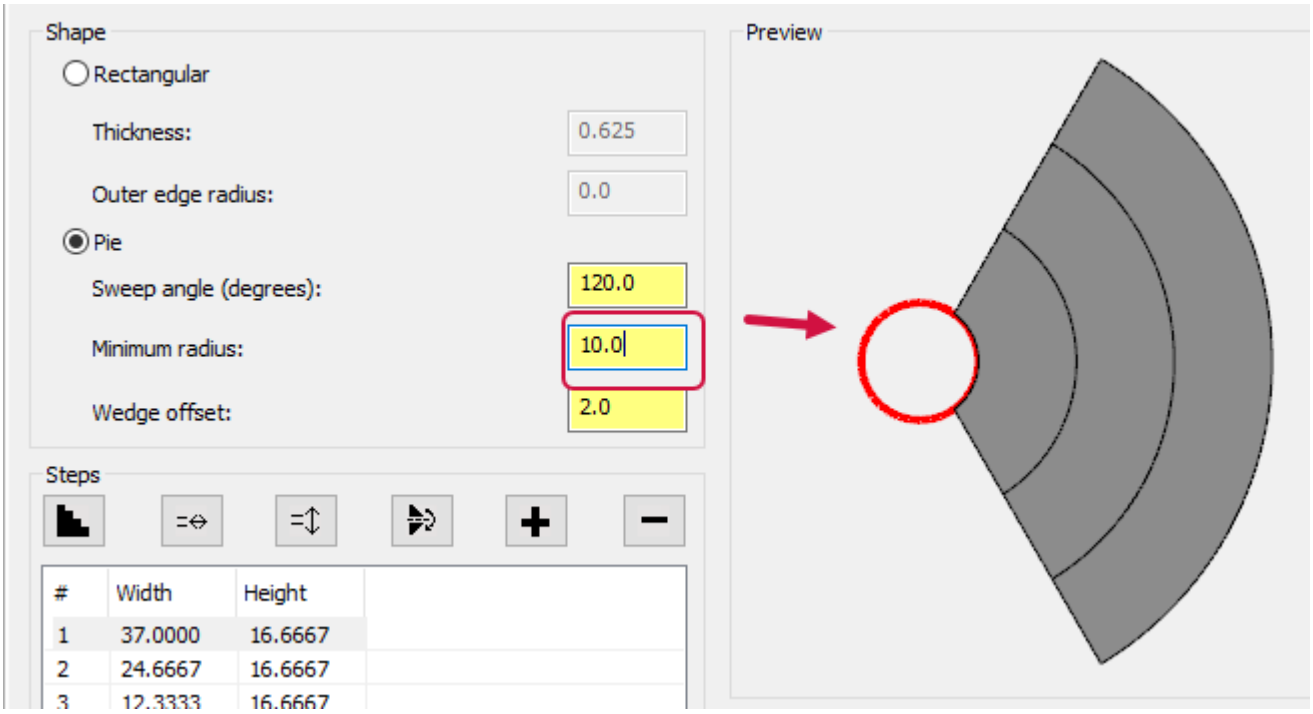


The **Preview** window has been significantly enhanced.

- Right-click in the window to rotate the component or view it from different angles.
- Use the **Translucency** slider to see details on hidden faces of the component.

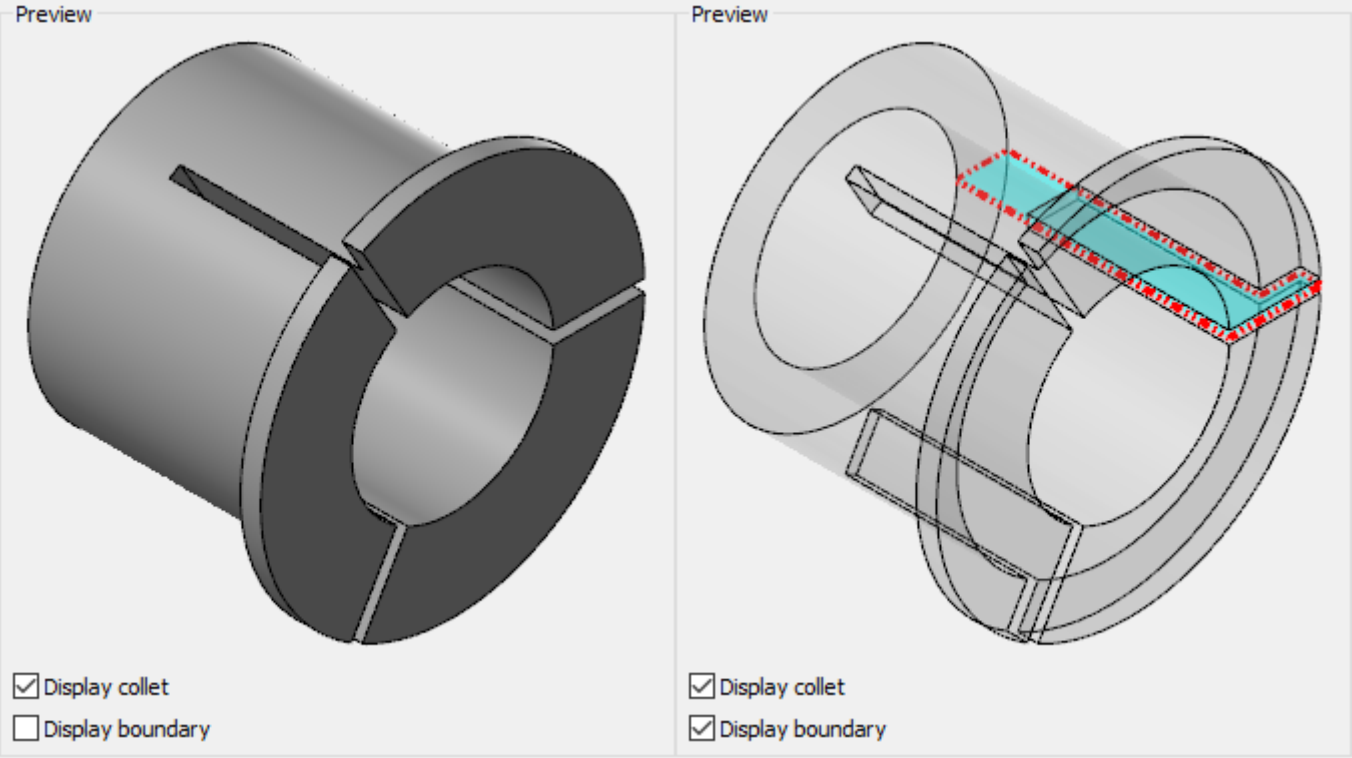


- The enhanced preview is also available for components that are defined parametrically or with chained geometry.
- Mastercam highlights the dimension or field that you are working on.



- The channel depth for chucks is previewed accurately.

- Additional options are available for specific components. For example, you can display the collet, its 2D boundary, or both.



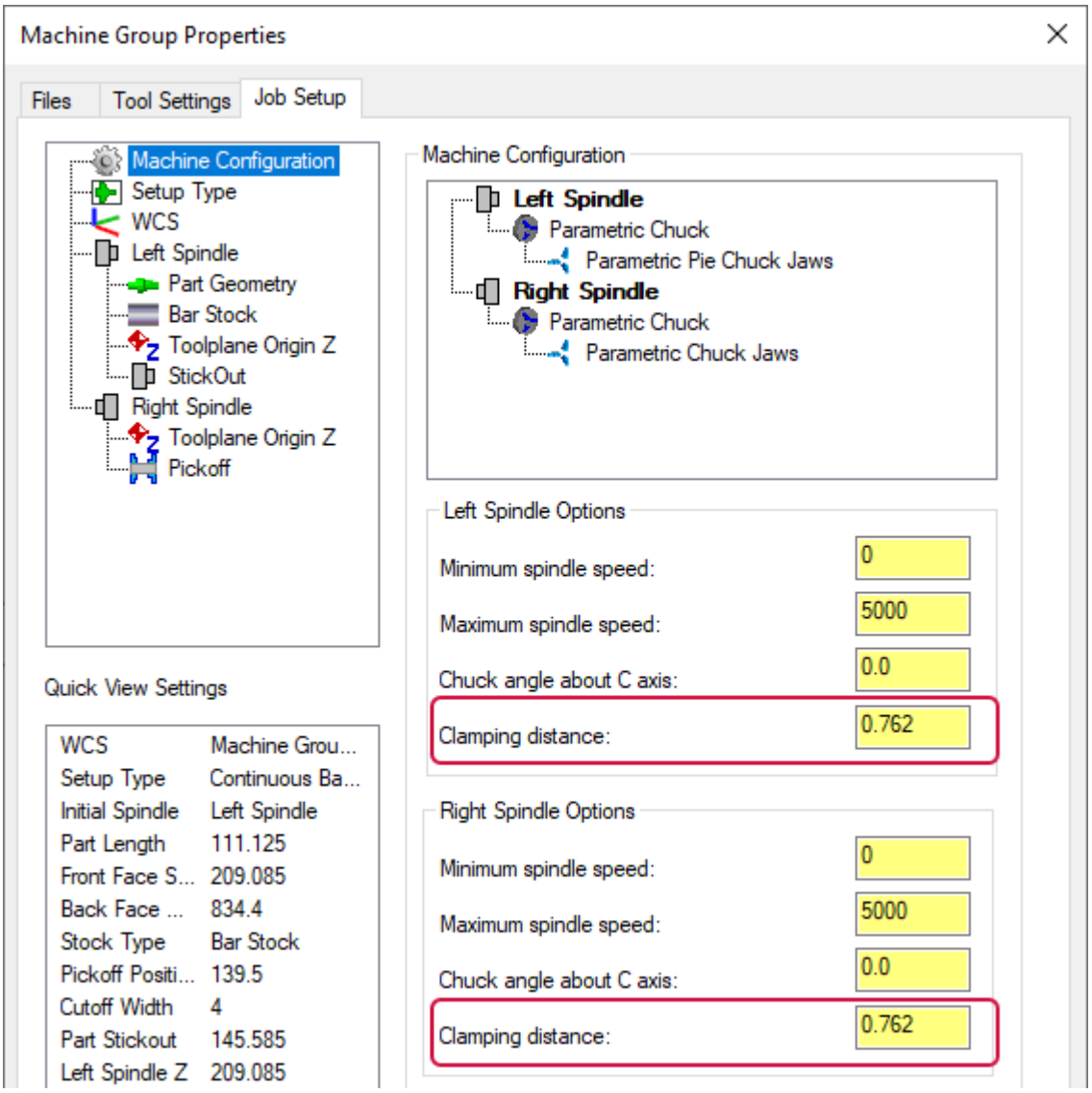
Improved Support for Jaw Movement

In earlier versions of Mastercam, the distance that the chuck jaws moved when clamping the part was defined as a clearance distance for the chuck.

This was misleading and confusing to many users. In Mastercam 2021 these settings have been more clearly labeled as **Clamping Distance** on the **Parameters** tab and incorporated into the general redesign of chuck and jaws components.

- New options let you enter the clamping distance manually or let Mastercam calculate it for you.
- Mastercam uses intelligent algorithms to calculate different values for jaws and collet components. For example, the range of motion for a collet is typically much smaller than for chuck jaws.

The redesign carries through to the Job Setup, where users can adjust the values as needed for each spindle. The fields are only available when the **Clamping Distance** has been set to **Manual**.



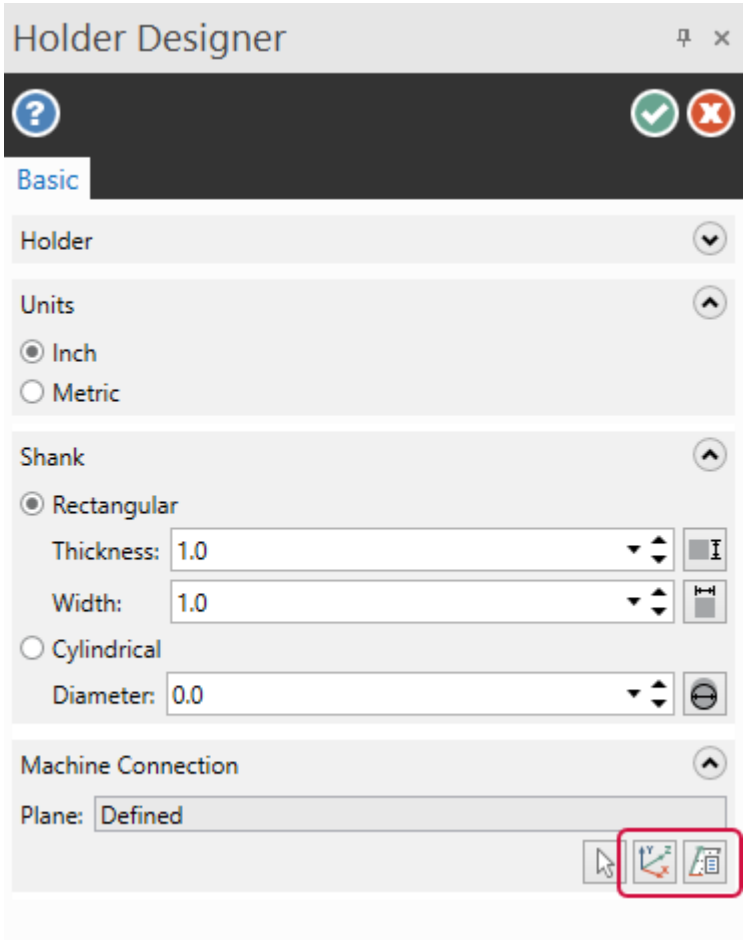
3D Tooling Enhancements

Listed below are enhancements made to the Holder Designer and 3D Tool Designer. Additionally, view manipulations in the Insert Designer and the Holder Designer are now consistent with Tool Designer, providing a smooth user experience with reduced mouse clicks.

Improving Machine-Connection Planes for Holder Designer

The following improvements have been made to the Holder Designer:

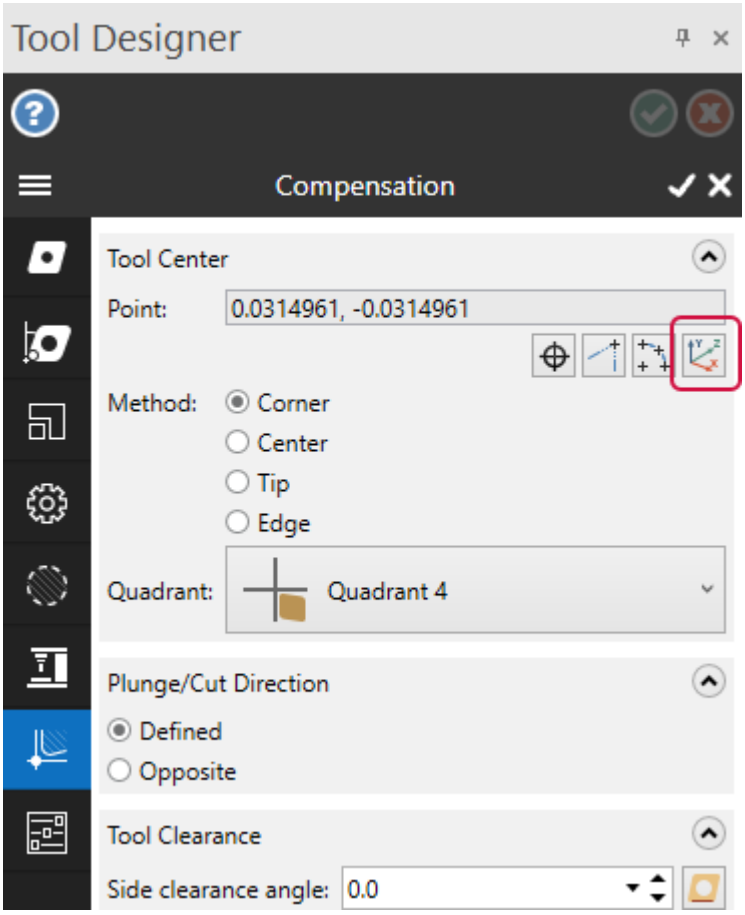
- The **Re-select location** button has been added to reposition the machine-connection plane center.
- The **Select named plane** button has been added to choose a named plane for the machine-connection plane.



Tool Designer Enhancements

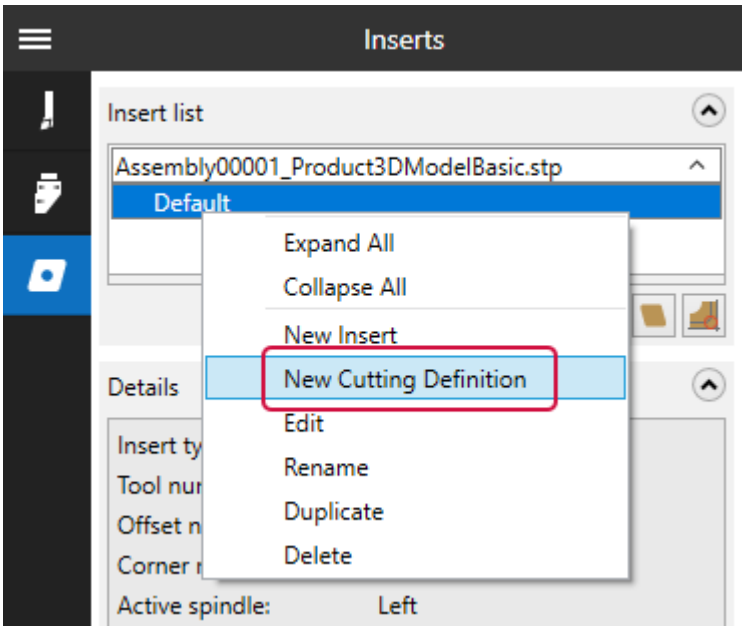
Adjusting a 3D Tool's Compensation Point

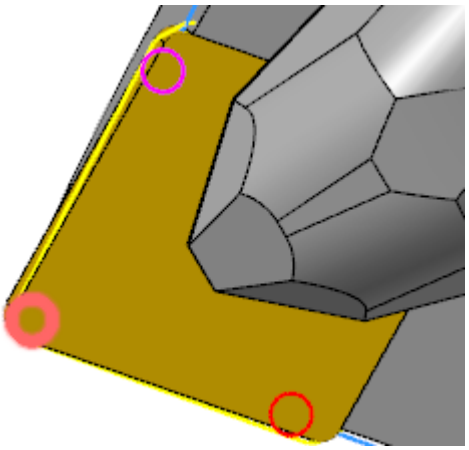
To help create an accurate 3D-tool definition with manufacturer-provided models of varying quality and accuracy, you can now make fine adjustments to the tool's compensation point. Click the **Fine Adjustment** button on the **Compensation** page to enter the function. The tool's solid models disappear, and the dynamic gnomon displays at the origin. You can then choose to move the geometry or reposition the gnomon.



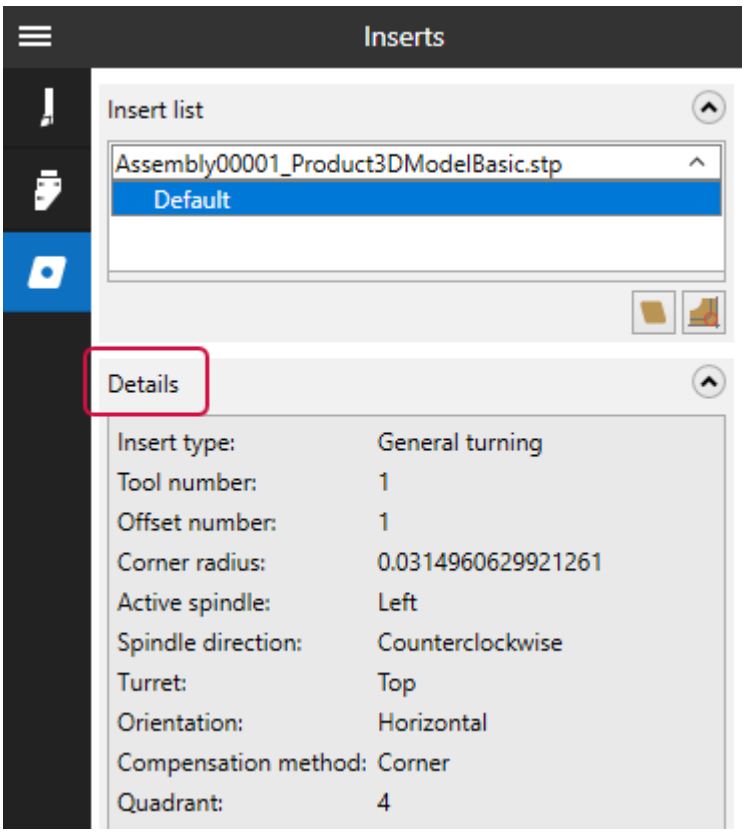
Creating Inserts with Multiple Definitions

Inserts in 3D tools can now support multiple definitions, which let you create versions of an insert with differing compensation methods, second-corner definitions, cut patterns, and so on. On the **Inserts** page of the Tool Designer, right-click in the insert list, and choose **New Cutting Definition** from the pop-up menu. Mastercam then displays the **Inserts** page which contains a group of options for defining an insert definition.



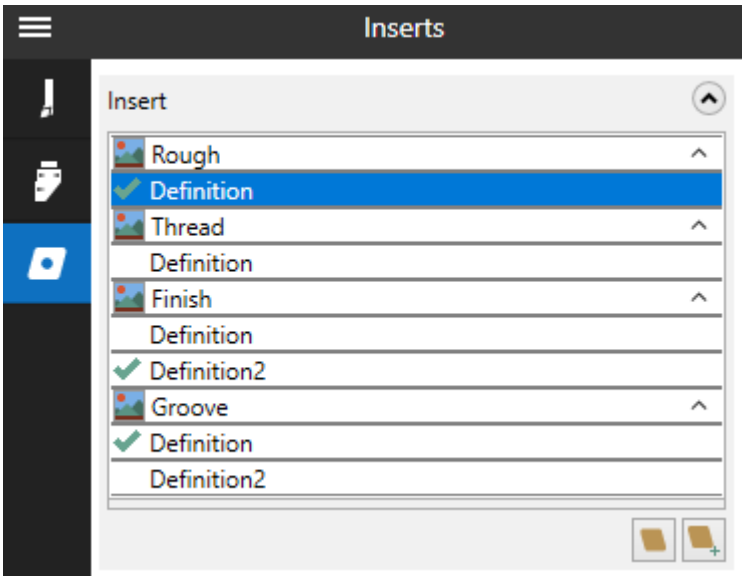


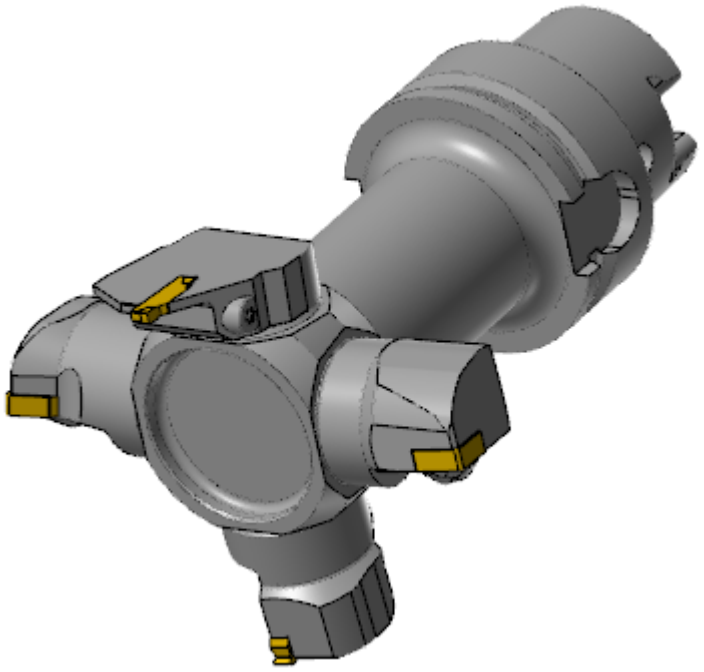
Additionally, there is a **Details** section that displays relevant information for each insert definition.



Creating 3D Tools with Multiple Inserts

Mastercam's 3D tools now support multiple inserts, with this new functionality located on the **Inserts** page of the Tool Designer panel. You define multiple inserts much as you did with single-insert models. Right-click in the insert list, and choose **New Insert** from the pop-up menu. Mastercam then displays the **Insert** page where you define the new insert like you did with previous 3D tools. Each insert you define is added to the insert list.

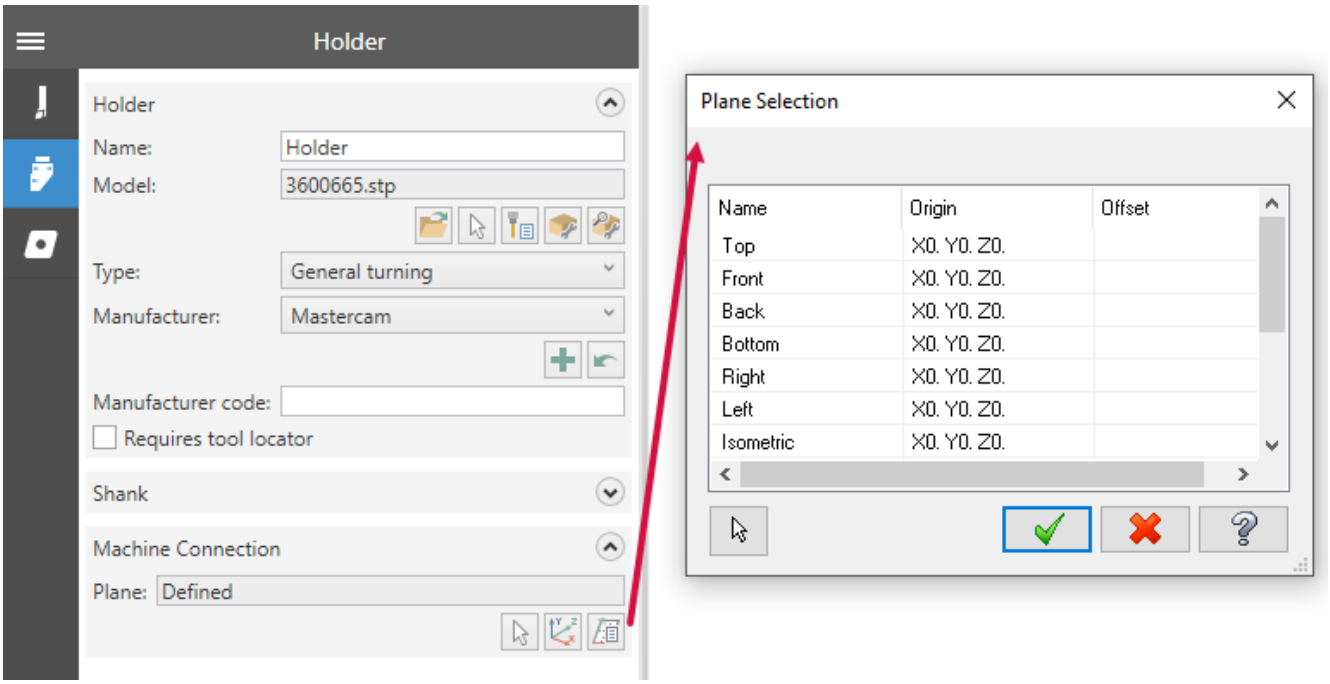




Resetting the Machine-Connection Plane

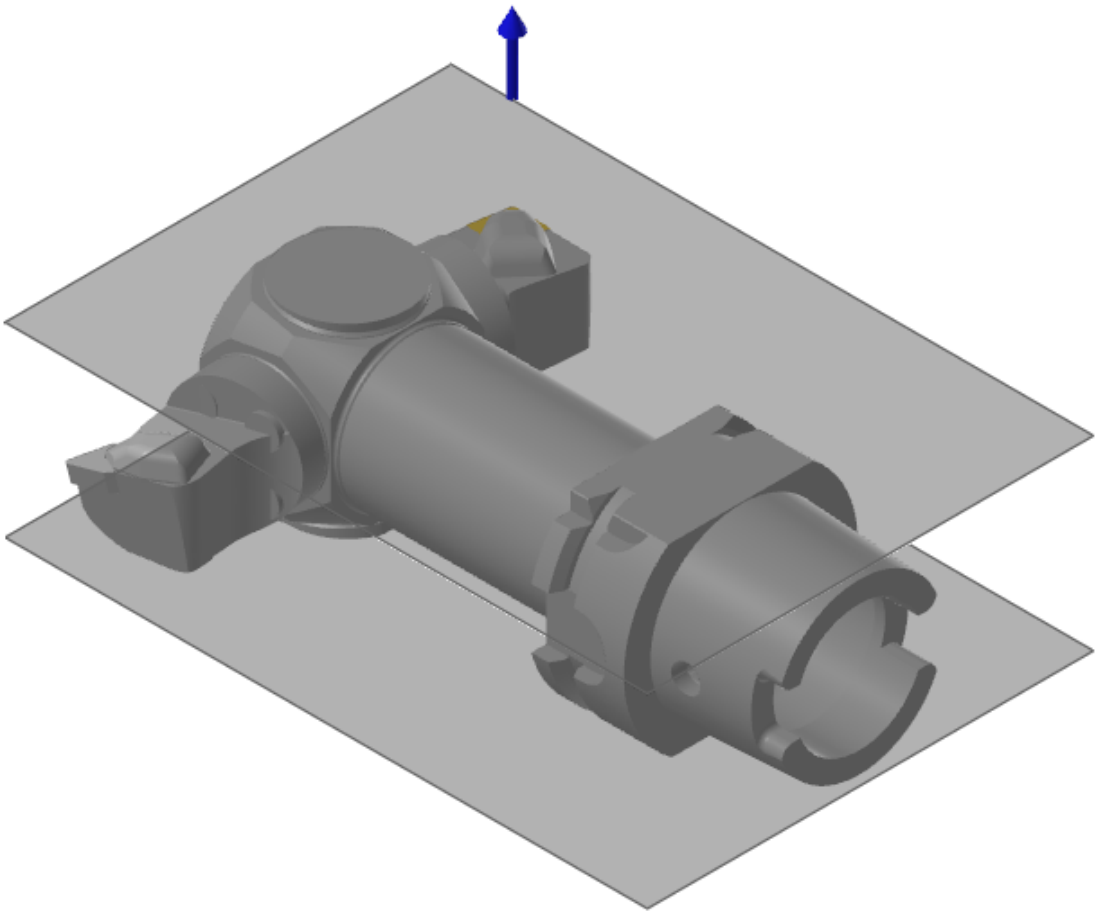
Previously, when replacing the holder model in Tool Designer, the machine-connection plane stayed populated. Because it is unlikely that the machine-connection remains correct after replacing the holder, the machine-connection plane now resets.

Additionally, you can now choose a named plane for the machine-connection plane. Click the **Select named plane** button on the **Holder** page to display the **Plane Selection** dialog box. After you select the plane, the re-position function launches so that you can choose the correct center position.



Improved Visual Feedback for Adjusting Boundaries

When adjusting the tool boundary in the **Boundary** page of Tool Designer, Mastercam now displays the location of the adjusted plane as a translucent rectangle.



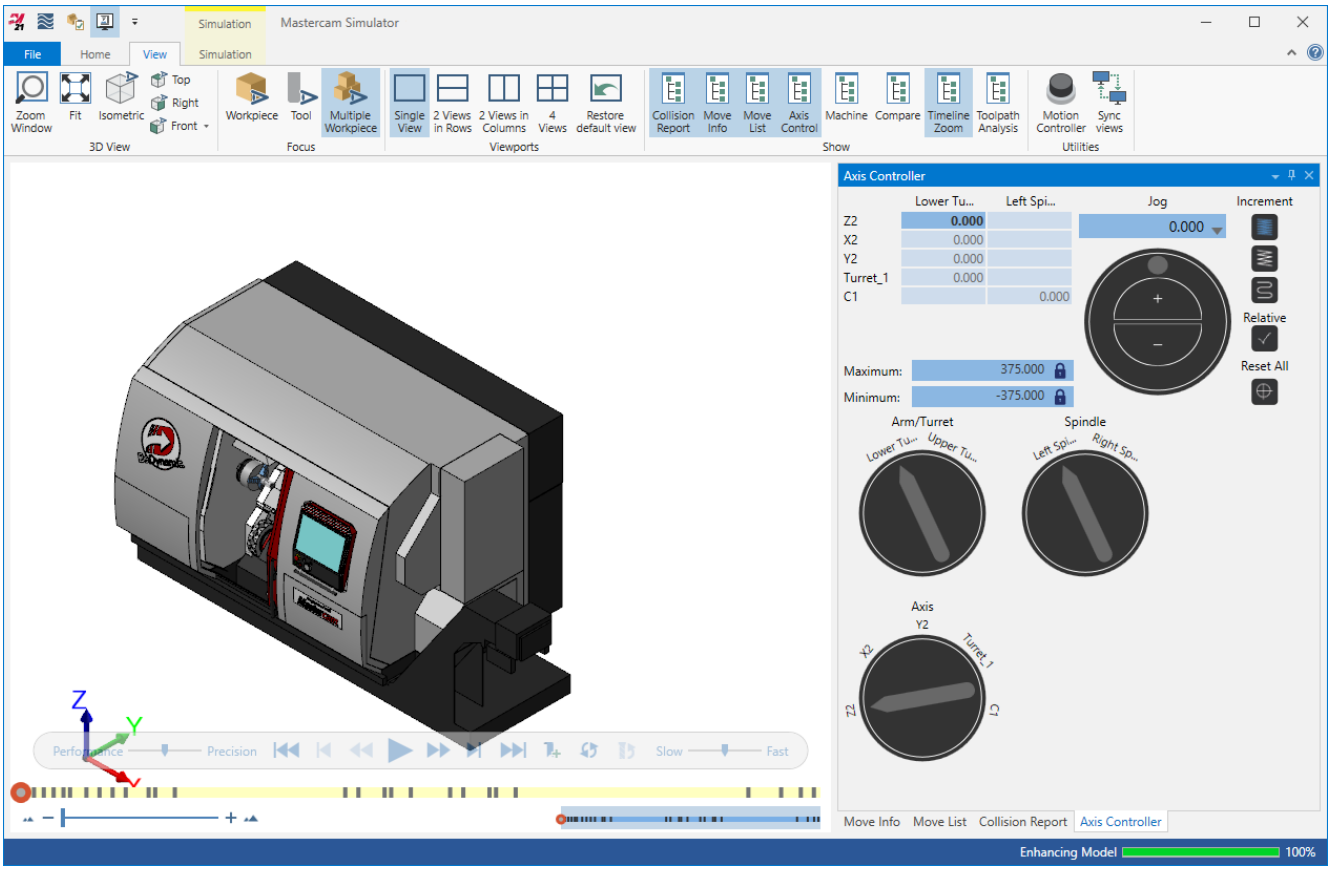
Adding 3D Tool Information to Setup Sheet

3D tool information has been added to setup sheet. The images and tags that have been added include:

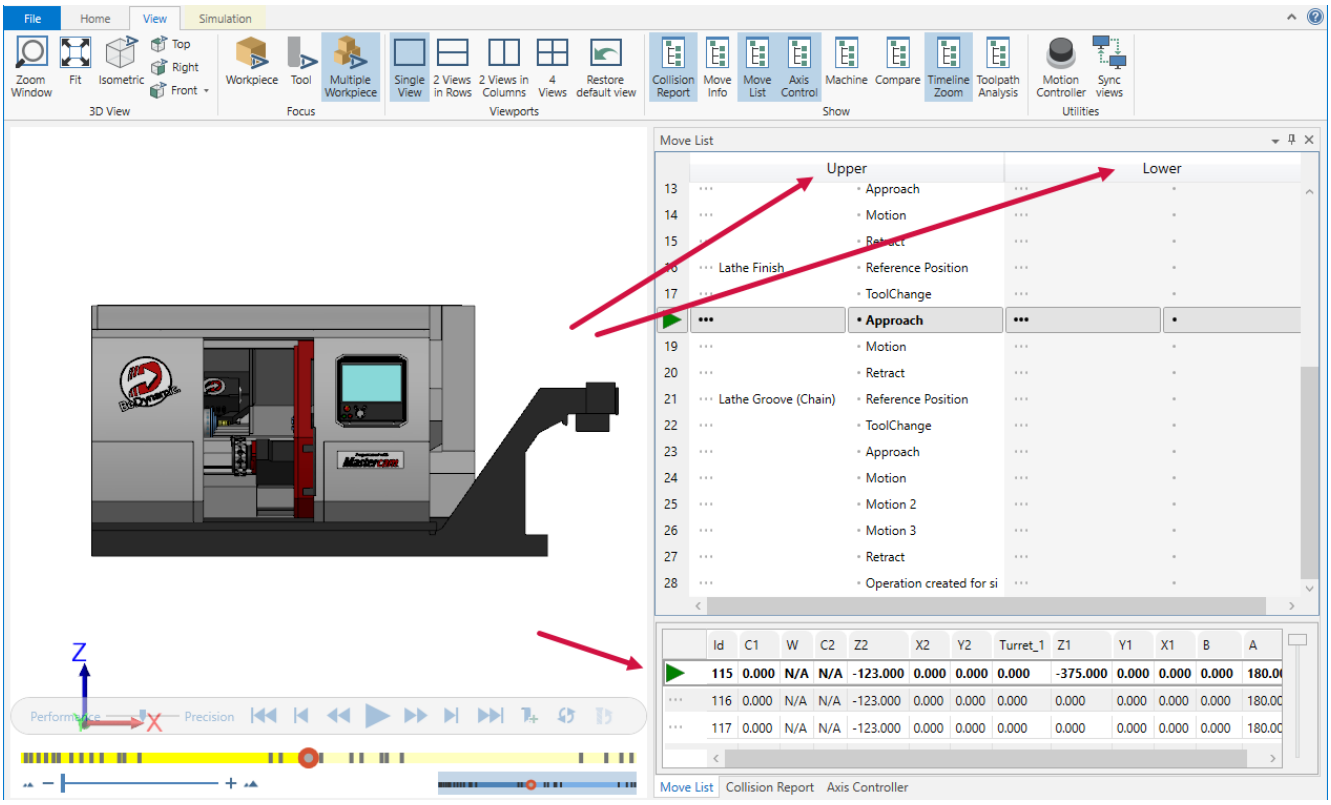
- 3D images for tools, holders, and inserts
- Images for multiple inserts on a tool
- MACHINE-TOOL-LOCATOR-NAME (for multi-station tool locators)
- MACHINE-TOOL-LOCATOR-STATION-NAME (for locators that have multiple stations)
- NAME (the insert name is filled in)
- INSERT-SLOT (added for debugging)

Mill-Turn Simulation with Mastercam Simulator

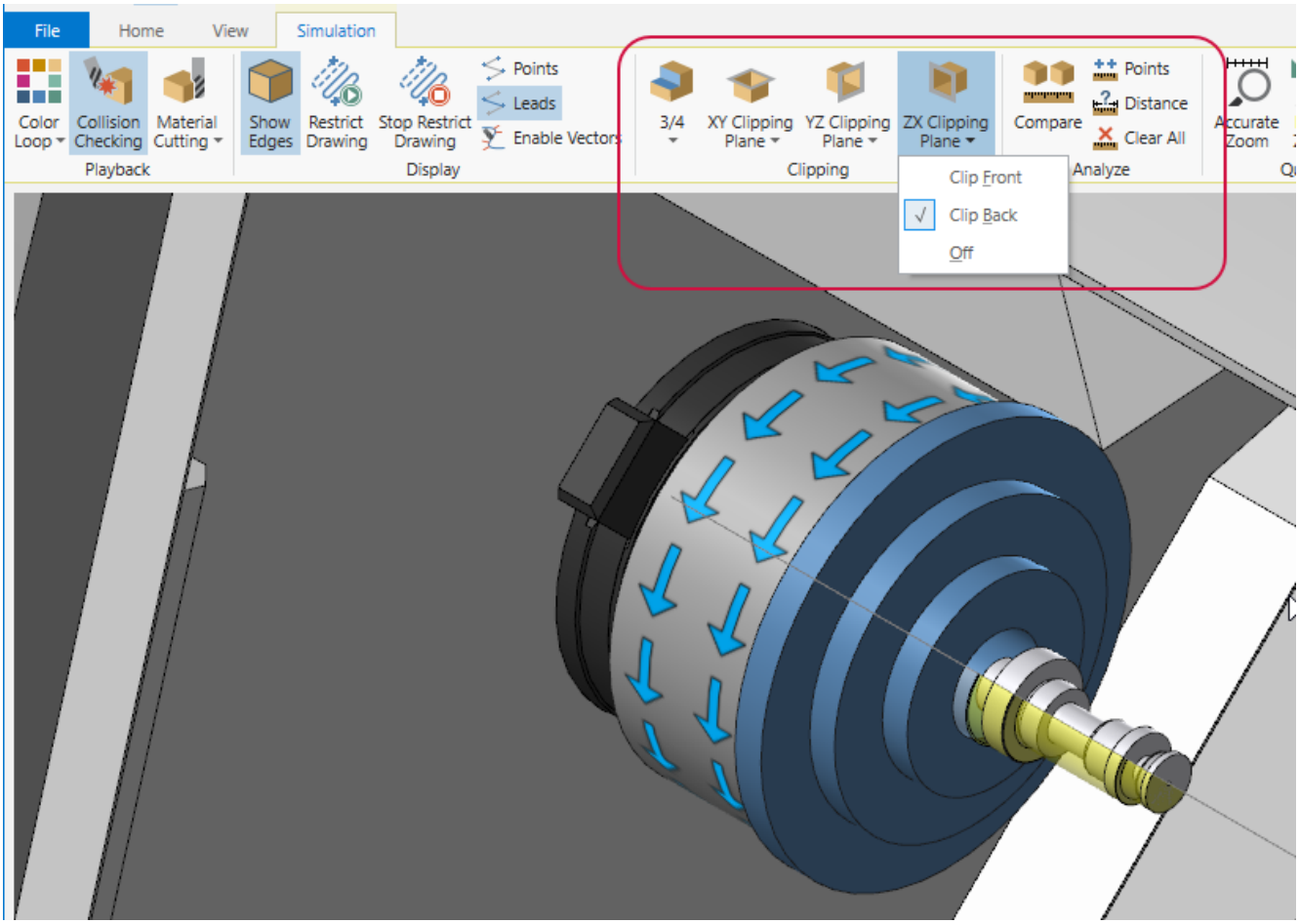
For Mastercam 2021, Mill-Turn Simulation has been added to the Mastercam Simulator interface. You can experience Mill-Turn simulation with the same interface tools as Mastercam Simulator.



The new interface includes an easier-to-use and more intuitive Machine Axis Controller, as well as other benefits. The method to launch simulation from the Sync Manager has not changed. The newly designed **Move List** allows you to see multiple streams of data at one time, as well as the status of every axis at each position in the toolpath sequence.



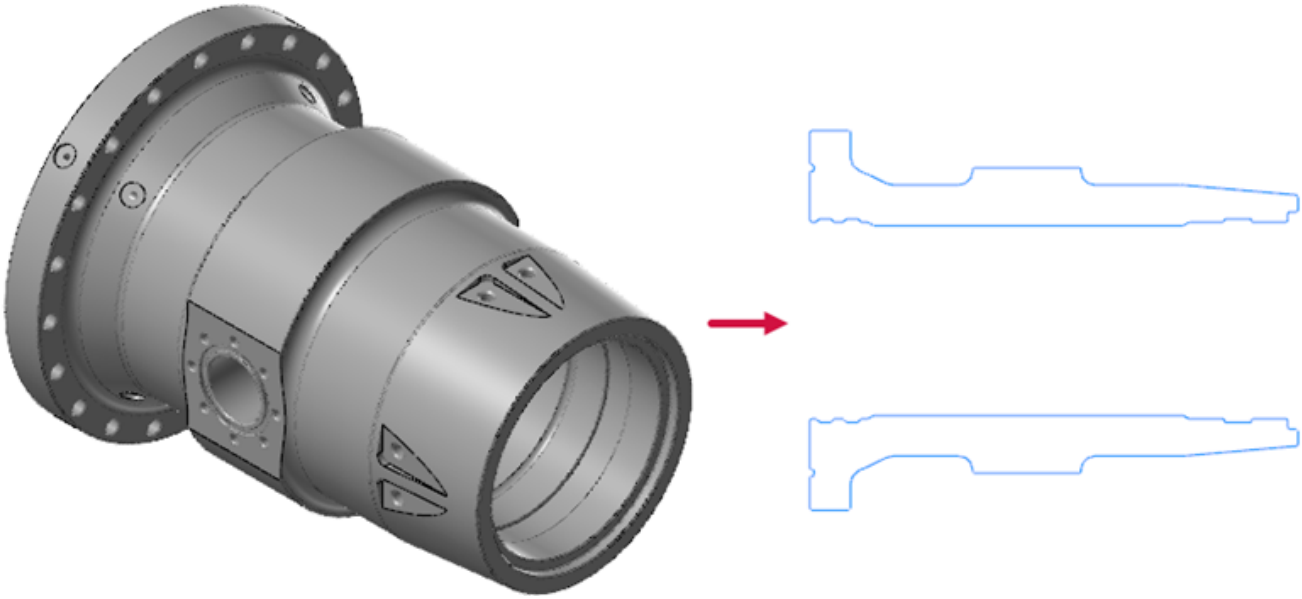
Familiar tools for stock comparison, stock clipping, and measuring are all available in Mastercam Simulator.



Lathe Solid Chaining Enhancements

Using a Spun Profile for Chaining

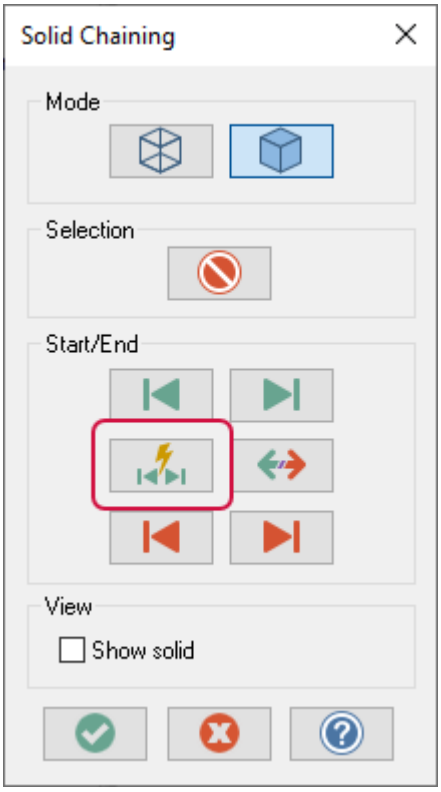
Previously, lathe solid chaining used a slice profile for chaining. This was problematic when the profile of the part was not cylindrical. In Mastercam 2021, lathe chaining now uses a spin profile which creates profile geometry by virtually spinning the geometry about the selected axis and generating a close approximation of the actual profile. This results in a more accurate profile without the need to add temporary geometry.



When using the spin profile, both the upper and lower profiles are displayed initially. Once you chain the upper or lower profile, the unchained profile will no longer display in the graphics window.

Dynamically Adjust the Start and End Point of Chains

Lathe solid chaining now includes the **Dynamic** option that is available in standard solid chaining. Use Dynamic to move the start or end of a selected chain to any position along an edge.

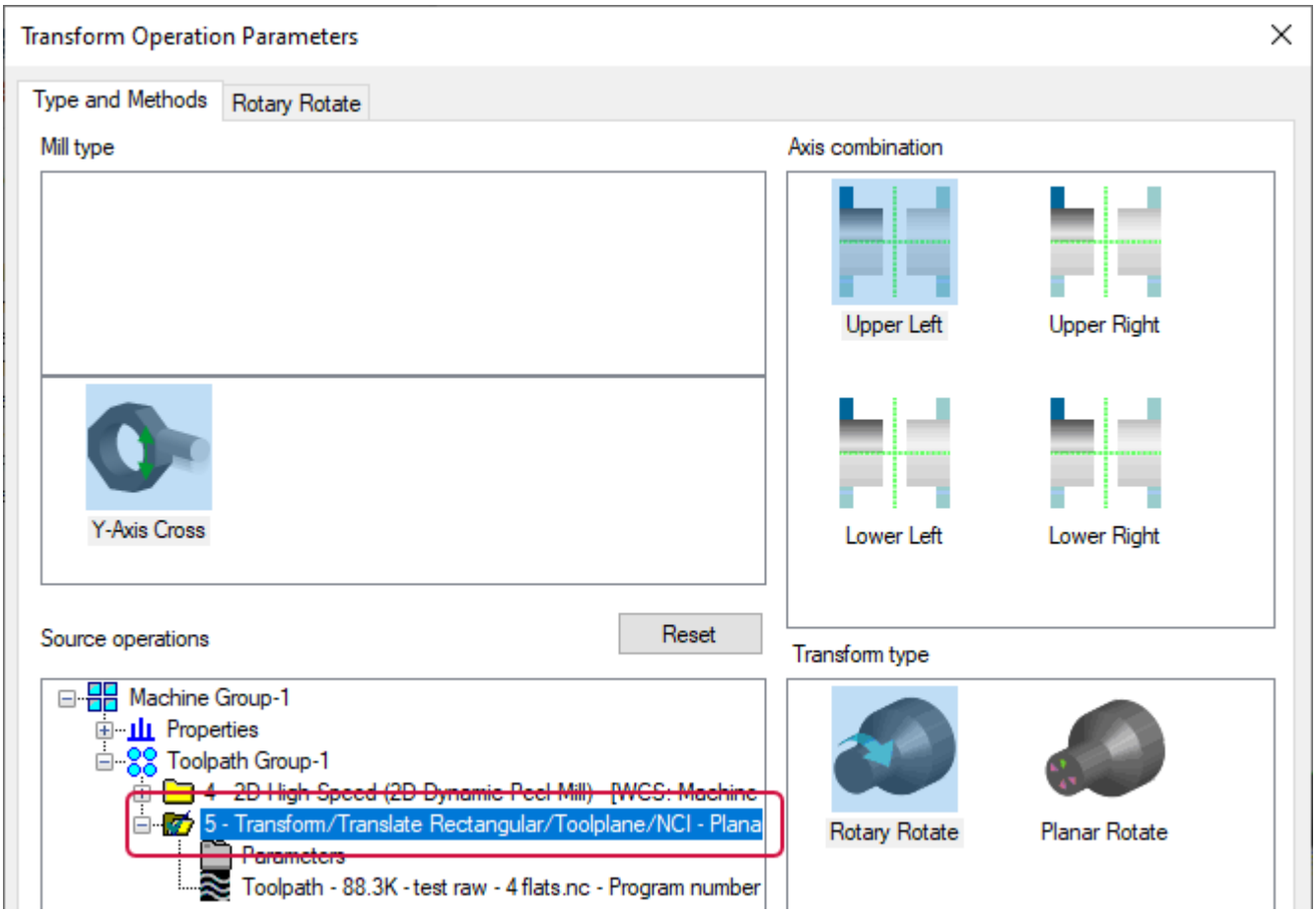


Transforming a Transform Operation for Mill-Turn

Mill-Turn users can now transform a transform operation, within certain limits. The most important limitation is that you cannot transform an existing transform operation if this would result in multiple tool planes being created.

For example, if you create a mill toolpath to machine a slot in Y-axis cross orientation, you can perform a rotary rotation transform to create multiple copies of the slot rotated about the Z-axis. If you wish, you can now transform that transformed operation with a planar translation transform—for example, to create additional sets of slots at different Z locations. However, Mastercam will not let you perform a planar rotation transform, because that would create additional multiple toolplanes that cannot be supported.

The **Transform Operation Parameters** dialog box includes the necessary logic to filter the lists of available operations or transform methods to ensure that you do not create unsupported transformation scenarios. For example, if you select a transform operation from the **Source operations** list, Mastercam filters the list of **Transform** types to display only supported methods. Similarly, if you select the **Transform type** first, Mastercam filters the list of available source operations. Mastercam also takes the **Mill type** and **Axis combination** into account when filtering the operations.



Improved Support for Swiss Machines with the Pickoff-Cutoff Toolpath

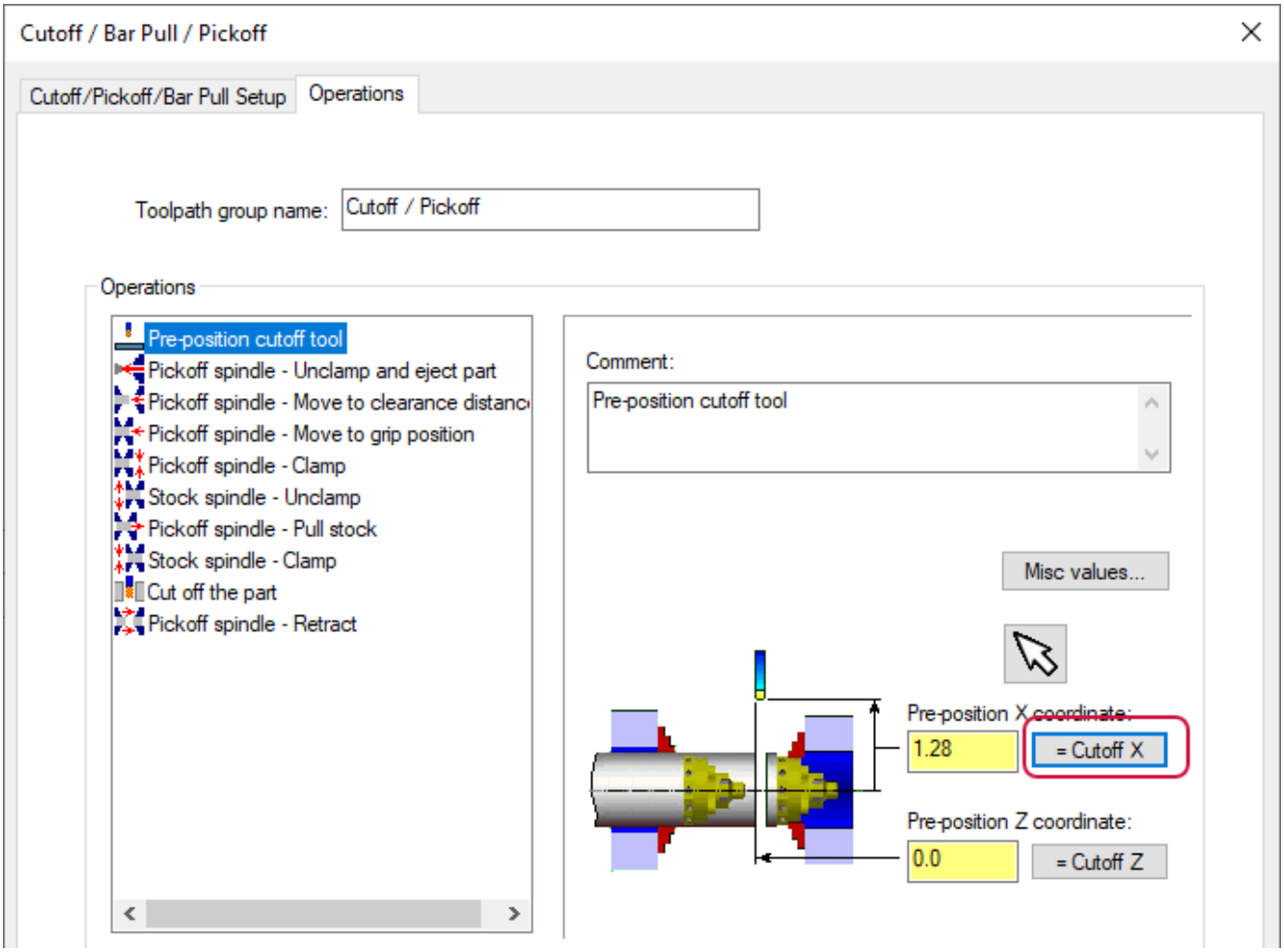
Mastercam's Pickoff-Cutoff toolpath has been enhanced to work better with Swiss-style lathes. The toolpath now works in two different modes:

- Lathe mode, which is similar to the way that it has always worked.
- Swiss mode, which is triggered by a switch in the post.

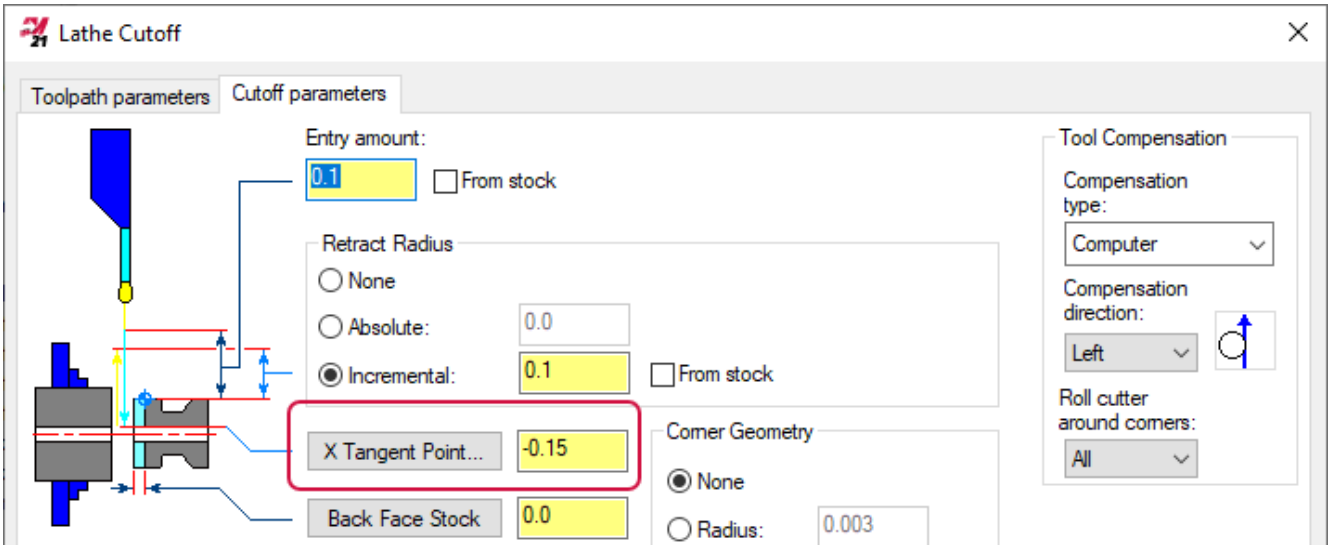
The new Swiss mode introduces the following differences.

On the **Setup** tab for the Pick-off Cut-off toolpath, Mastercam uses a different method to calculate the X coordinate of the cutoff move. When you click **From stock**, Mastercam adds the tool clearance distance from the **Machine Group Properties** dialog box to the stock dimension. This ensures that you will not get a collision warning when pre-positioning the cutoff tool.

For pre-positioning the cutoff tool, a new button named **= Cutoff X** has been added to the **Operations** tab. By selecting this button, the **Cutoff X** coordinate value copies to the pre-position page. This ensures that the cutoff tool will be pre-positioned at the proper point to begin the cutoff operation. In addition, the minimum X value for the pre-position point has been reduced from the top of the chuck to zero.



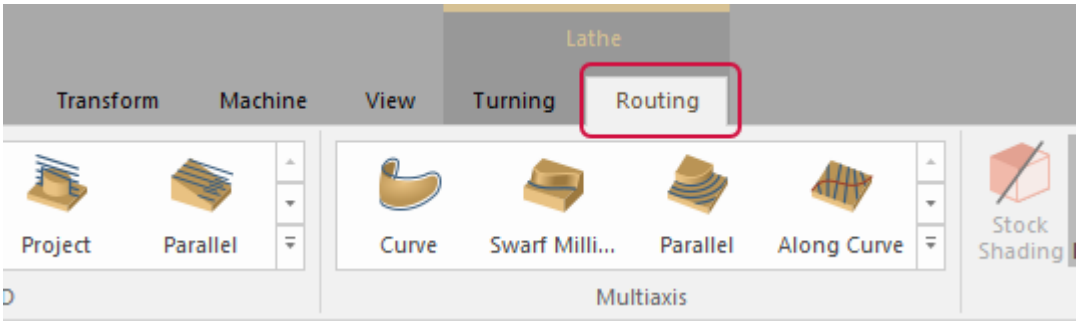
Mastercam also uses the **X Tangent Point** value from the Lathe Cutoff operation differently.



- In Lathe mode, **X Tangent Point** is interpreted as an absolute value.
- In Swiss mode, Mastercam calculates the **X Tangent Point** value from the stock model. The **X Tangent Point** value from the Cutoff parameters tab is instead added to the calculated value as an adjustment.

Creating Router Operations with Mastercam Lathe

Lathe users who also have a Router license are now able to access Router's milling operations without owning a separate Mill license.



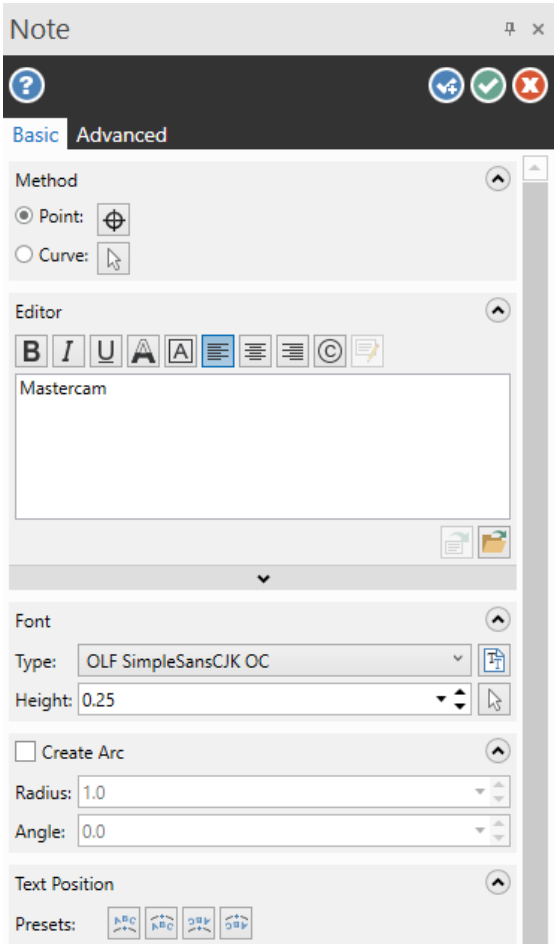
However, the Mill-Turn product still requires a Mill license in addition to Lathe if you want to create milling operations.

DESIGN

Listed below are major enhancements made to the Design product.

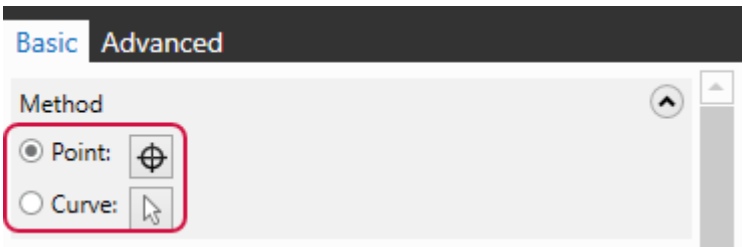
Note Enhancements

Mastercam makes it easier to create text for notes and labels by adding many of the capabilities of the Create Letters function into the Note function. Additionally, Note's re-designed function panel gives you improved usability, more options, and increased control. Note is the preferred method of creating text for notes, labels, and toolpath geometry.



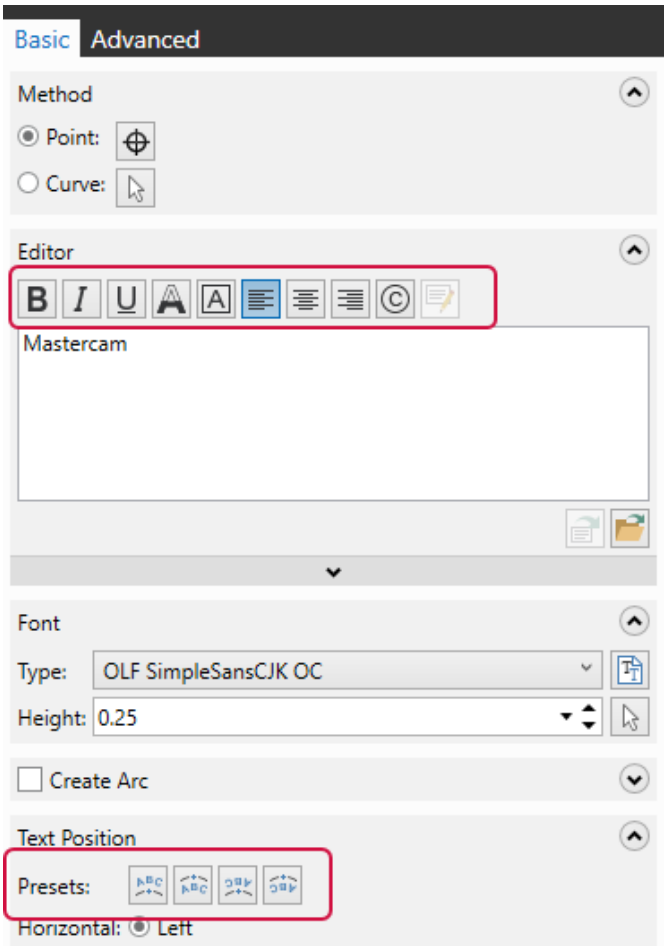
Redesigned Function Panel Interface

There are two new methods you can use to place your note in the graphics window.



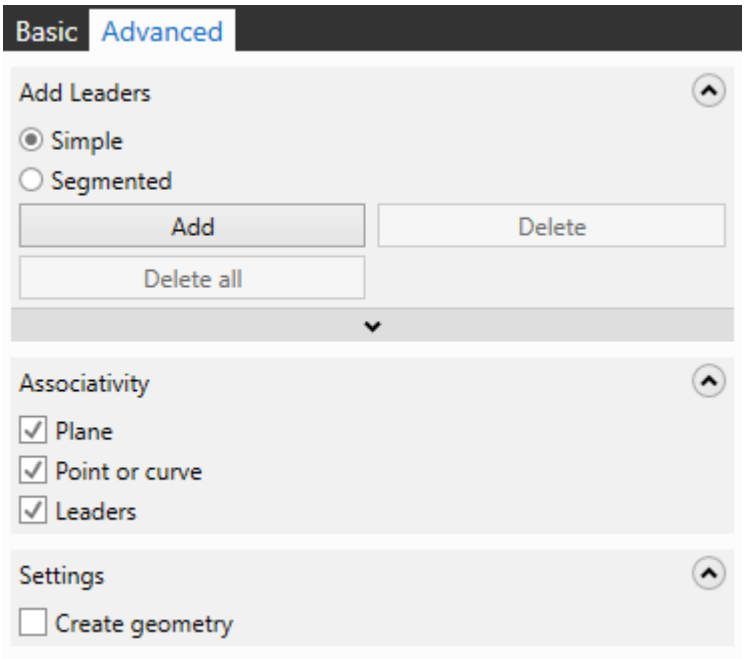
- **Point** positions the new note in a linear direction and enables the **Create Arc** functionality. **Create Arc** uses the selected position on the screen as the center point of the text as it rotates around a circle without requiring wireframe.
- **Curve** lets you position the new note on any selected line, arc, or spline. Use your cursor to reposition the note text by clicking and dragging it along the entity.

New options in the function panel are available so you can easily modify the formatting and alignment of your text. Some examples are the new editing buttons above the text editor that you can use to control the style of the text and the new **Presets** buttons that let you quickly position the text relative to the basepoint.



Just as you can in the Create Letters function, Note gives you the ability to position your text around an arc without the need of any pre-existing geometry. Reposition the note text by clicking and dragging it along the arc.

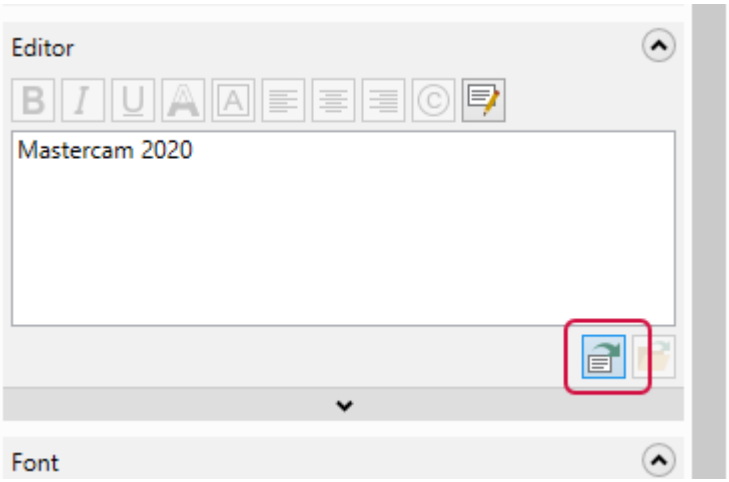
On the new **Advanced** tab, you will find options to add simple or segmented leader lines and to convert your text to chainable geometry when you exit the function. You can also associate your note to a **Plane**, **Point or curve**, and/or **Leaders**.



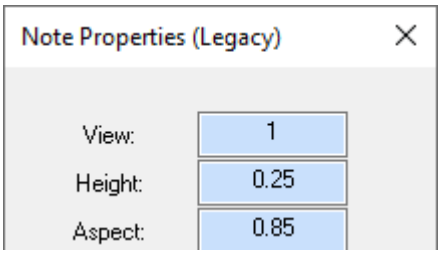
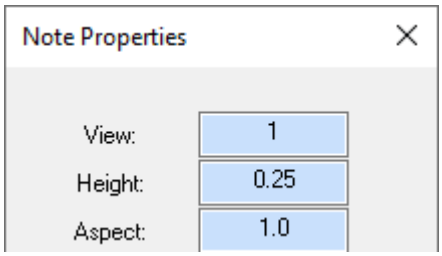
Updating Legacy Notes

The redesigned Note function supports notes from prior releases of Mastercam with some limitations. You can edit legacy notes in Mastercam 2021 with the same capabilities that were available in previous versions of Mastercam.

You cannot use the new formatting buttons on a legacy note. However, if you update a legacy note to a new note entity, all of Note's new editing capabilities are available. To convert a legacy note, select **Convert legacy note** in the function panel.



Mastercam's Analyze Entity function indicates if a note was created in a previous Mastercam version by updating the title of the dialog box with the word **(Legacy)**.

Legacy Note	Mastercam 2021 Note
	

Creating Text with TrueType Fonts

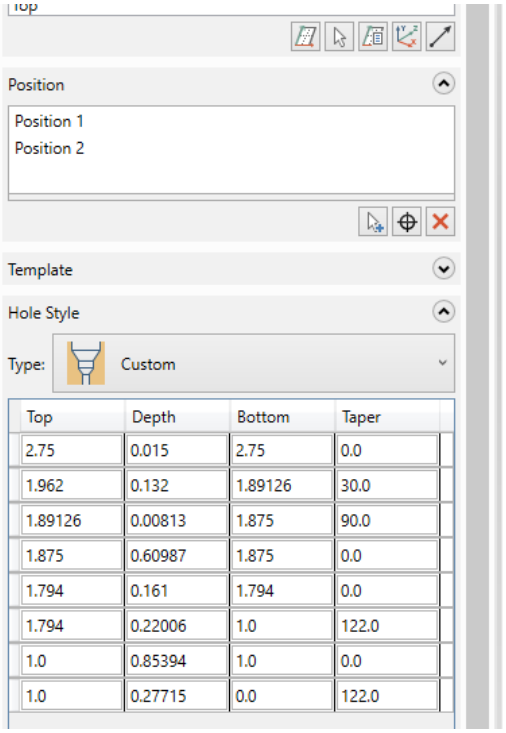
Any text created in Mastercam 2021 uses TrueType fonts, which includes our new, single-stroke fonts. You are able to create only alphanumeric characters using older-style fonts such as Stick and Box fonts with the Create Letters function.

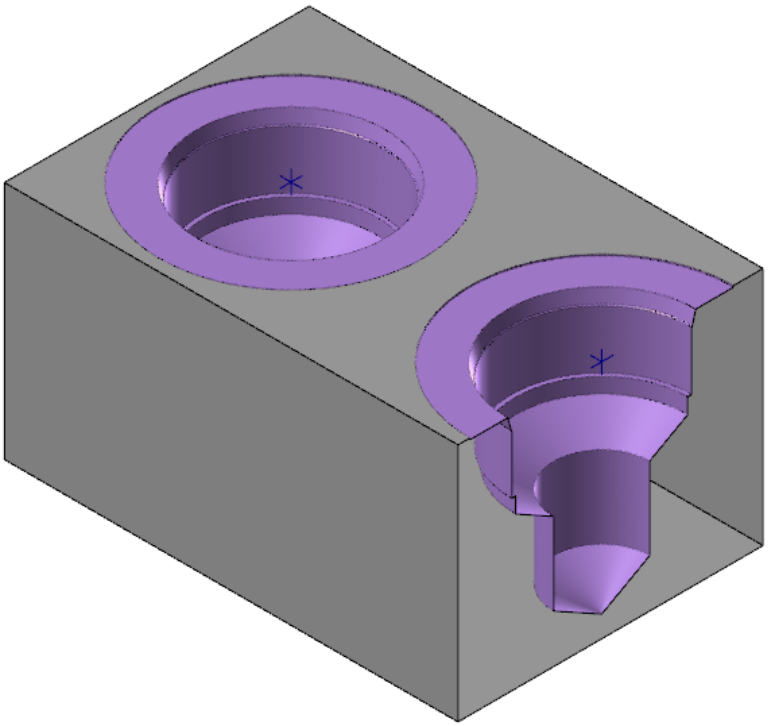
Solid Enhancements

Listed below are enhancements made to Solid functions.

Detecting and Creating Custom Holes

Mastercam 2021 has the power to recognize complex hole geometry—including intersecting and non-conventional holes—in solid bodies with no history. Once Mastercam discovers these holes, you can use the new, expanded options in the Hole function to save your custom holes to a template for future use.





Additionally, you are no longer limited to our default styles. You can create any style custom hole as needed and save it to a library for reuse in the future.

Detecting Holes with Add History

Use Mastercam's Add History function to detect complex holes in solid bodies. Simply select Hole operations to find all holes within the range of minimum and maximum radius values that you enter. Mastercam creates an operation in the Solids Manager for each unique detected hole style.

Occasionally, intersections, interruptions, and certain dimensional configurations will make it difficult for Mastercam to detect some holes. If the automatic detection does not return all the holes in the body, use **Manual** mode to select the faces that will define the hole. When you choose this mode, Mastercam disables the minimum and maximum radius values and accepts selections where the interior diameter of one section is larger than the sections that adjoin it. Mastercam will process all adjoining faces to create a hole operation. Single face and Window selection are available in this mode.

Saving and Editing Standard and Custom Hole Operations

In the Solids Manager, Mastercam creates a Custom hole operation for any hole that does not match an existing hole type already found in the part. You can edit this operation just as you can any other solid operation.

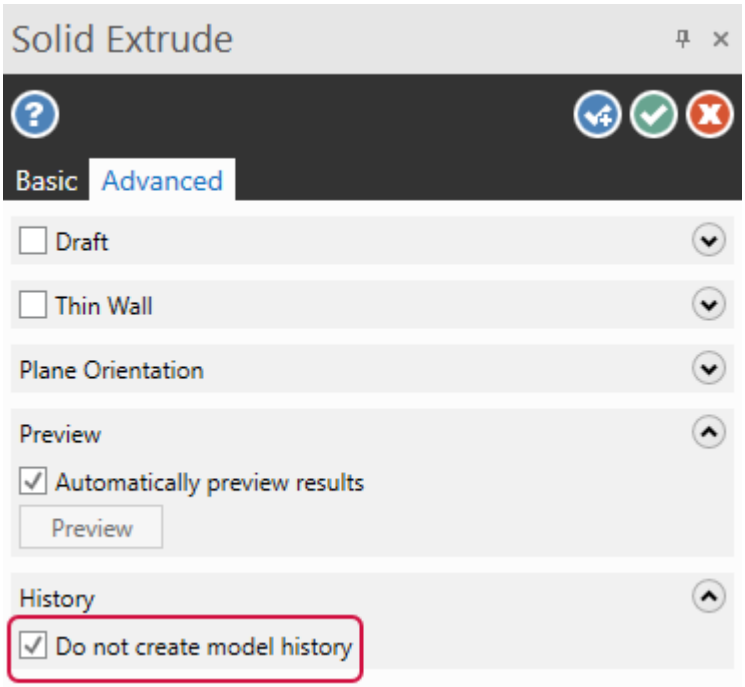
Double-click any Custom hole operation to open the **Hole** function panel. Each part of the hole can be modified by adjusting the dimensions in the **Hole Style** section of the panel. You can also save your custom hole dimensions to a library (previously called a template) so that it is available in future sessions of Mastercam.

You can even convert your standard holes into custom holes which populates all custom hole values with values from the standard hole. You can then add new segments to the converted standard hole as opposed to creating a hole from scratch.

Powerful new options let you modify custom holes after their initial creation. Use the new **Add Segment** option to add a new row or line segment to the bottom of the custom hole grid. Right-click on the first column of the grid and you can add segments and features such as chamfer.

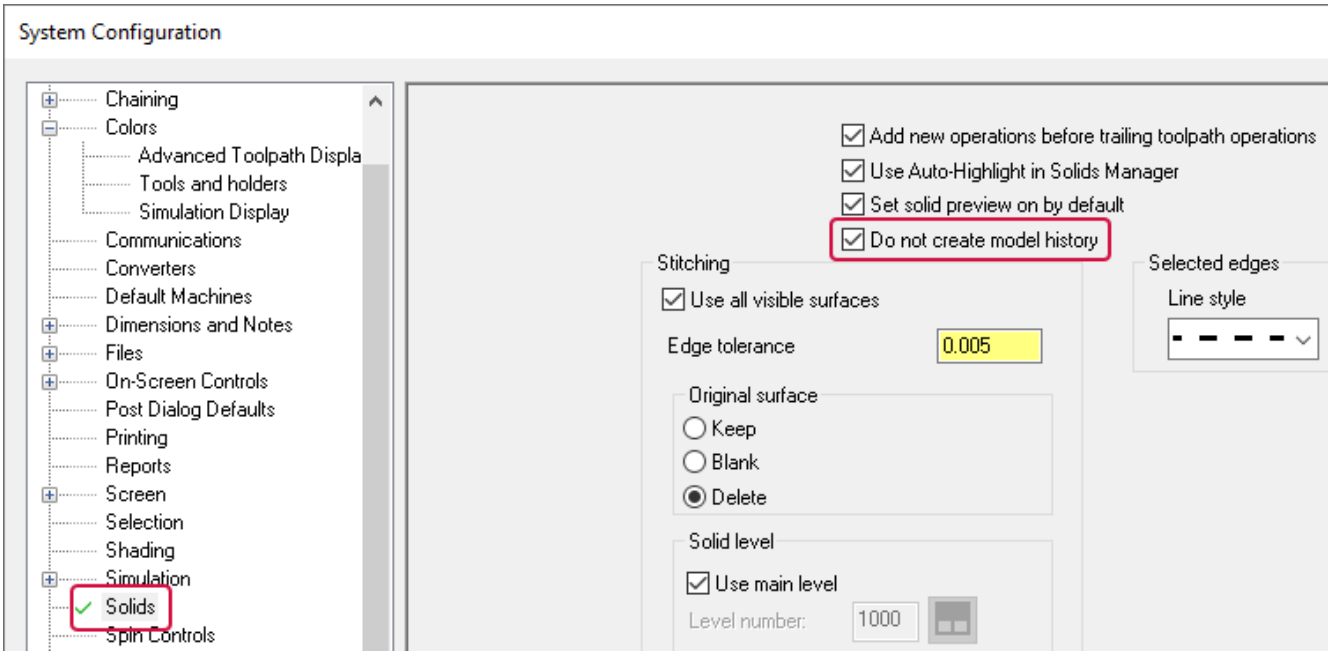
Using Solids Functions Without Adding History

In this version of Mastercam, you can now use any history-based Solid function without adding history to the file. Unlike solids that have histories, solids without histories do not have to be regenerated each time you edit them. To activate this mode, select the **Do not create model history** option on the **Advanced** tab of the history-based function you are using.



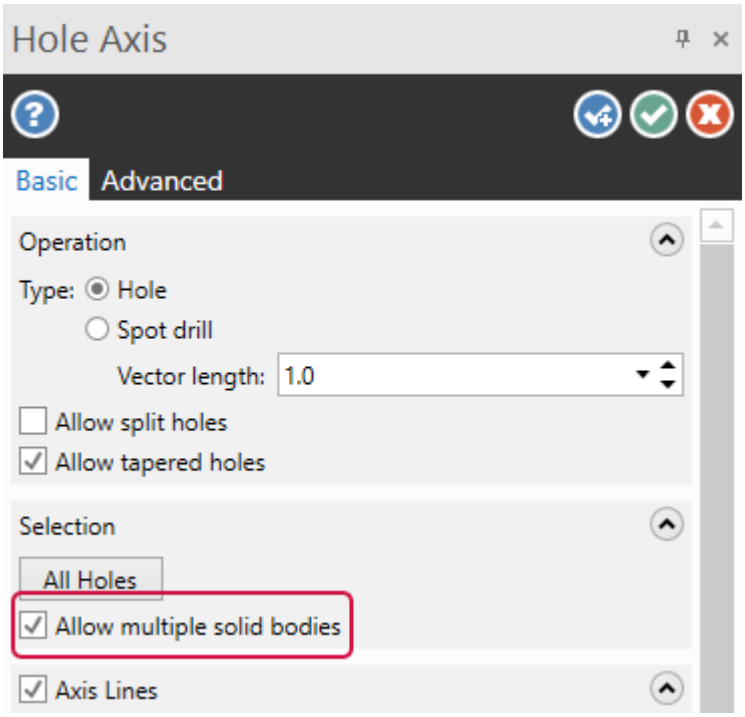
Once you activate the option, it stays selected in all history-based function panels. Just as if you were to apply Model Prep functions to a solid with history, if you modify a solid with history while you are in this mode, you will remove all that solid's history. The option is disabled in existing solid operations in the Solids Tree.

To change the default setting of this option, select **Do not create model history** on the **Solids** page of the **System Configuration** dialog box.



Selecting Multiple Holes Across Multiple Bodies

You can now select holes of the same diameter on multiple bodies with one click. Select the **Allow multiple solid bodies** option on the **Basic** tab of the **Hole Axis** function panel, and press the [**Ctrl**] key when you click the desired hole diameter. All holes that match the highlighted diameter in that body as well as in any other visible body will be selected.

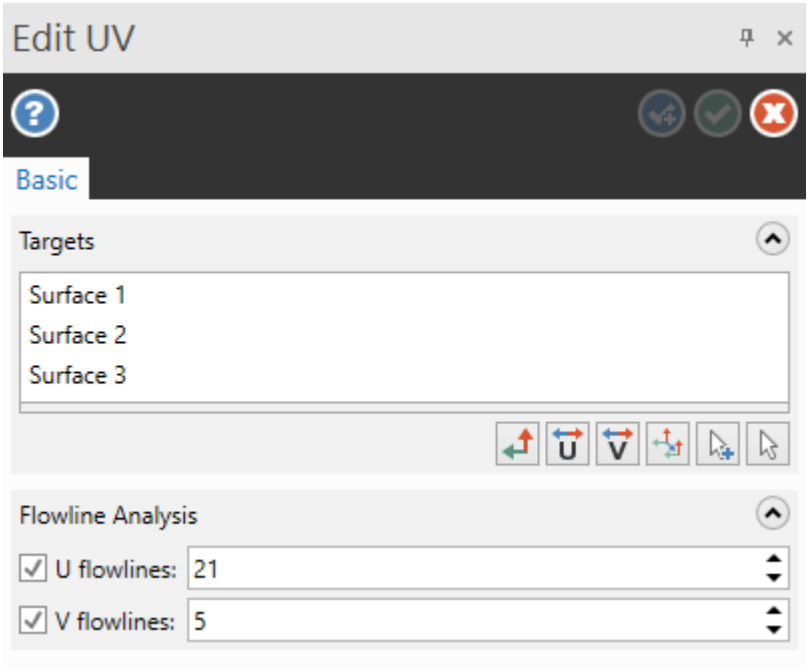


Analyzing and Modifying UV Directions on Surface Models

Mastercam 2021 offers two new flowline functions, Edit UV and Reflow UV, that you can use to analyze and modify the UV direction on surface models to more effectively utilize the flowline milling toolpaths.

Editing the U and V Directions of Surfaces

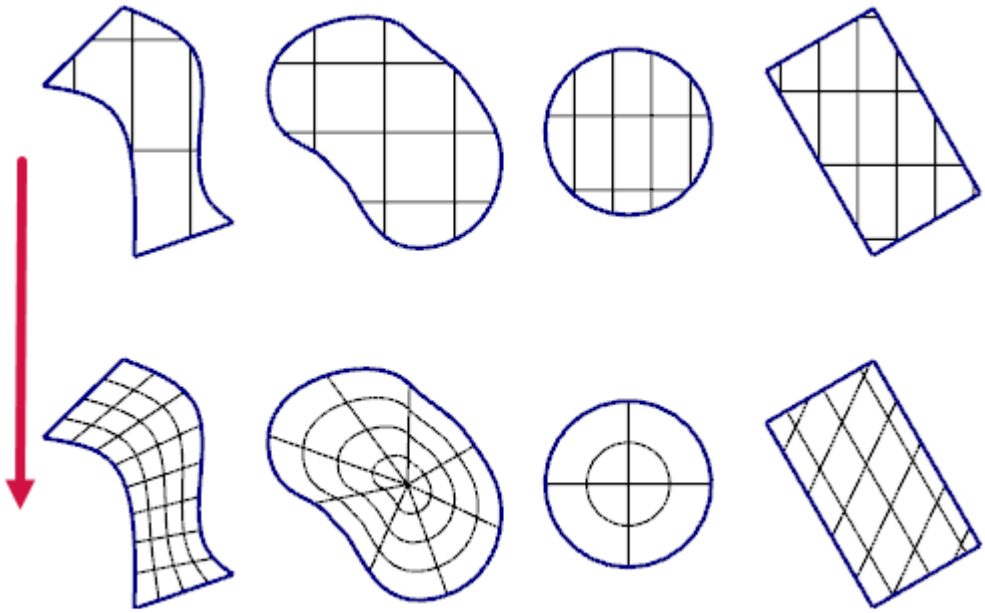
Edit UV allows you to switch U and V axes as well as to independently reverse U and V directions of flow on any number of surfaces.



After the surfaces are selected, a UV tangent arrow control will appear at the center point of each surface, and if enabled, temporary flowlines are drawn on the surfaces. You can also use the **Propagate** option to change all targeted surface U and V directions to match those of a "seed" surface. The targeted surfaces must touch.

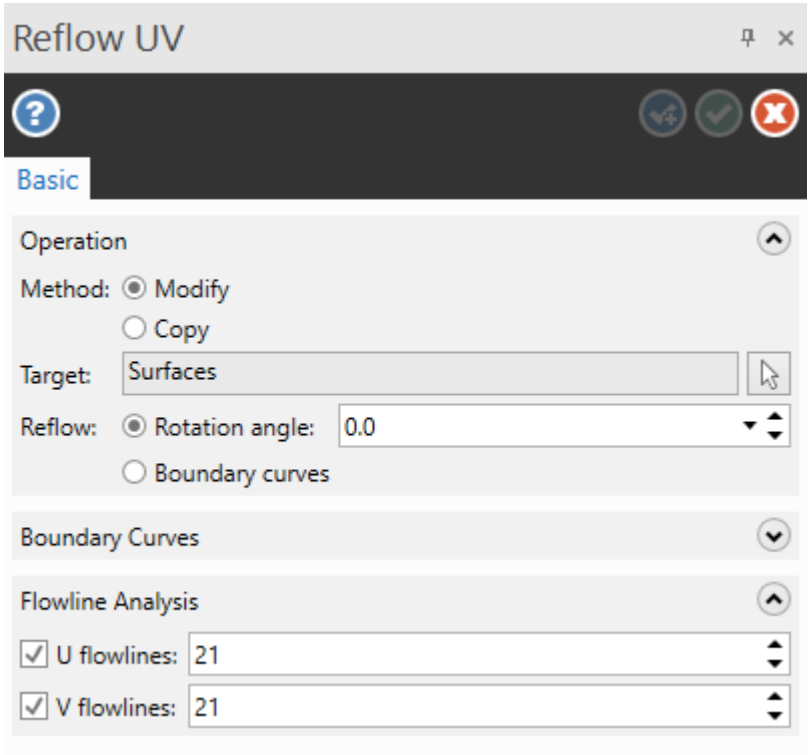
Redefining Surface Flow

Reflow UV allows you to redefine the flow by specifying a rotation angle or new boundary curves.



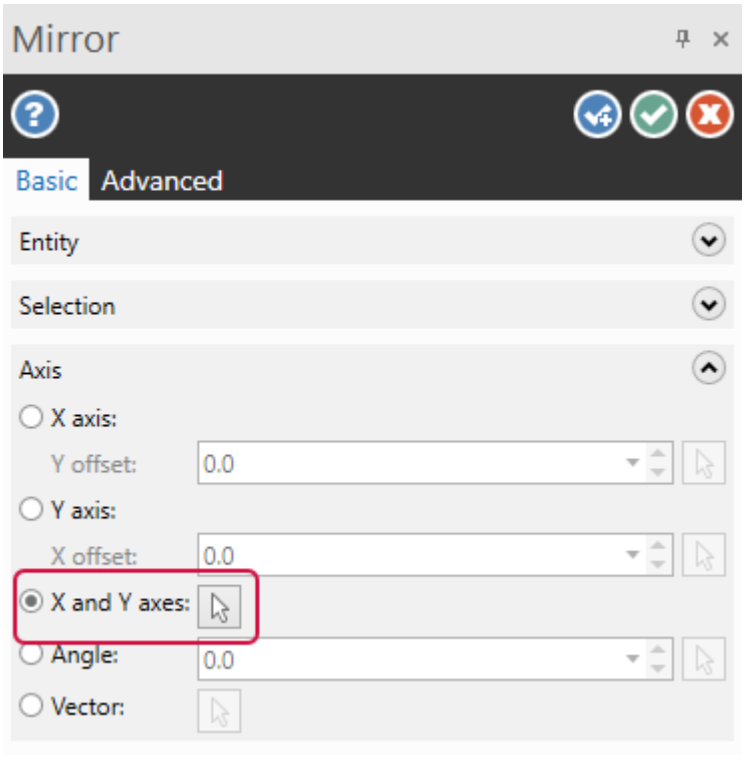
First select the surface you want to reflow. Then:

- Select **Rotation angle** to place a rotation control at the center of the parent surface. The flow is initially aligned with the U direction.
- Select **Boundary curves** to choose up to four curves to define the boundaries of a new surface to be formed over the existing surface. The flow is redefined by the shape of the new boundary curve.

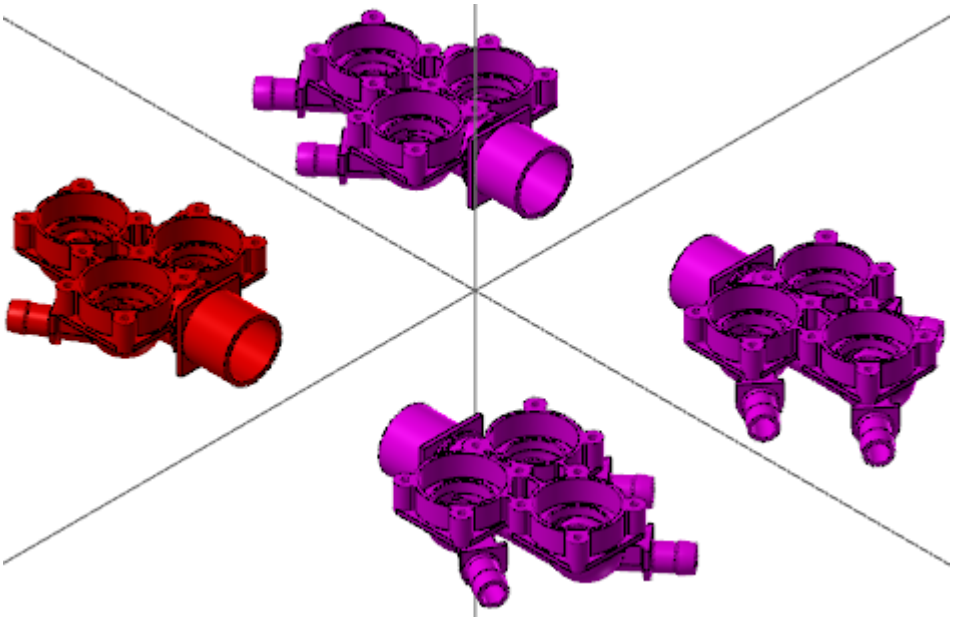


Mirroring About Both the X and Y Axes

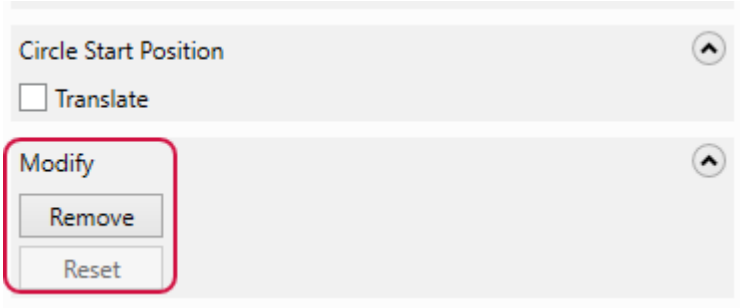
In previous versions of Mastercam, you needed two operations to mirror geometry about both the X and Y axes using the Mirror function. A new option in Mastercam 2021 allows you to mirror geometry about both the X and Y axes in one operation.



Not only does the **X and Y axes** option create three copies of the selected geometry immediately, it also allows you to define a new axis origin by selecting a position on the screen.



Additionally, we have added the modify commands, **Remove** and **Reset**, so that you can remove or restore any of the four instances of your transformed geometry.



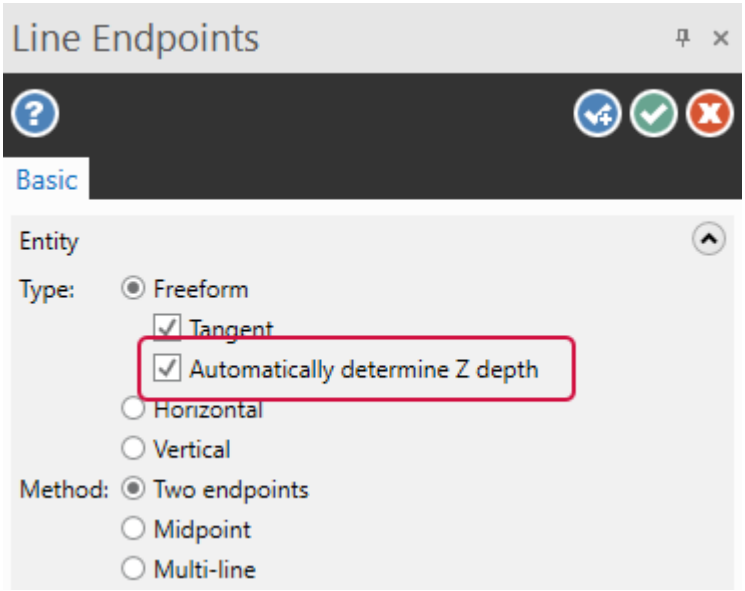
NOTE
The **Join** method is disabled when you select to mirror about the X and Y axes.

Wireframe Enhancements

Listed below are enhancements made to Wireframe functions.

Automatically Determine Z Depth

When you created geometry in previous releases, using Line Endpoints, Line Perpendicular, Arc Three Points, or Arc Endpoints, Mastercam projected the second endpoint to the system's Z depth unless you snapped it to an existing AutoCursor point. The new **Automatically determine Z depth** checkbox keeps any new, non-AutoCursor points at your first endpoint's AutoCursor depth. **Automatically determine Z depth** is not available when you work in 2D mode.



If you use the **Multi-line** method to create lines, Mastercam will keep the new point at the depth of the previous AutoCursor position until you select another AutoCursor position.

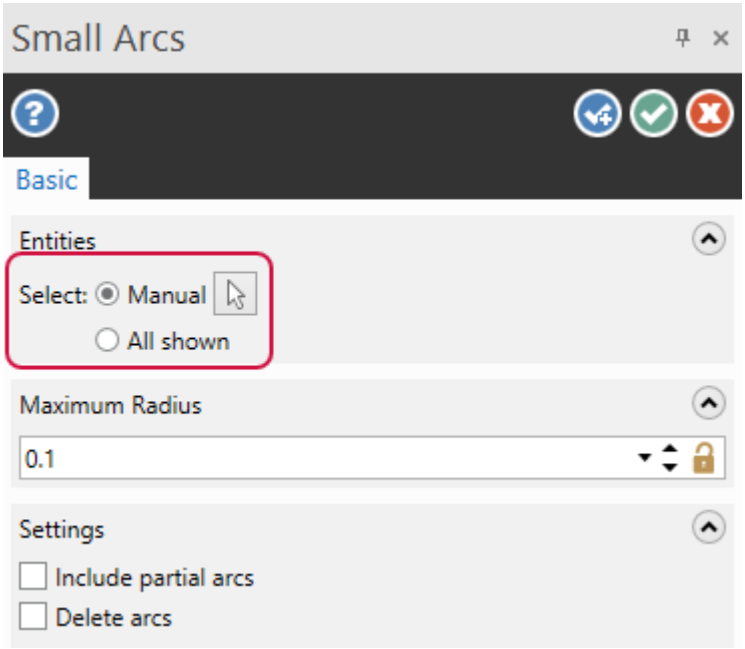
Modifying Geometry by Dragging Your Mouse

The "drag mode" functionality that was previously introduced to the Divide function is now available for Trim to Entities (for all **Methods** except **Auto**), Fillet Entities, and Chamfer Entities functions. As you drag the mouse, Mastercam performs the function on the wireframe entities your mouse encounters. Change the settings in the function panels to modify each action.

If you complete a sequence (by releasing the mouse button), you still have the option to use the controls on the Quick Access Toolbar, or [**Ctrl+Z**] and [**Ctrl+Y**] to undo or redo your actions.

Incorporating DrillPt Functionality into Small Arcs

The **Small Arcs** function panel now incorporates the functionality of the DrillPt Add-in. The DrillPt Add-In has been removed from Mastercam 2021.



Point Small Arcs is located under the **Point Position** drop-down on the **Wireframe** tab.

Miscellaneous Enhancements: Design

- You can draw a window to select multiple faces when you want to change the color of a set of solid faces.
- The **Clear all** function on the **Model Prep** tab now supports preselection, so you no longer need to reselect solid bodies when you reset faces and features to their original colors.
- Mastercam now applies the result color to entities that you modify with the following functions. This display acts as a visual confirmation of your modifications.
    - Simplify Spline
    - Break at Intersection
    - Break Circle
    - Close Arc
    - Spline - Convert to NURBS

- You can now create point segments on multiple entities with the Point Segment function by using the new **Chain** option.

GENERAL

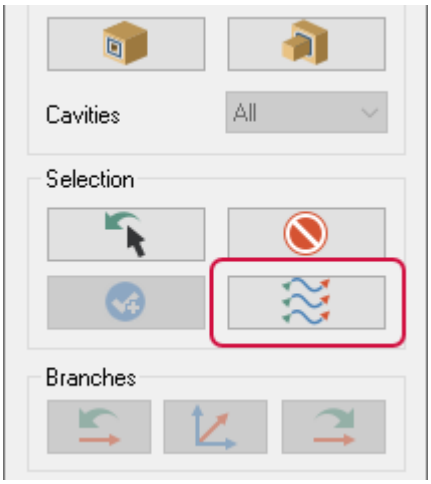
Listed below are general enhancements made to Mastercam. This includes improvements to selection and other functions not specific to one product line.

Chaining Enhancements

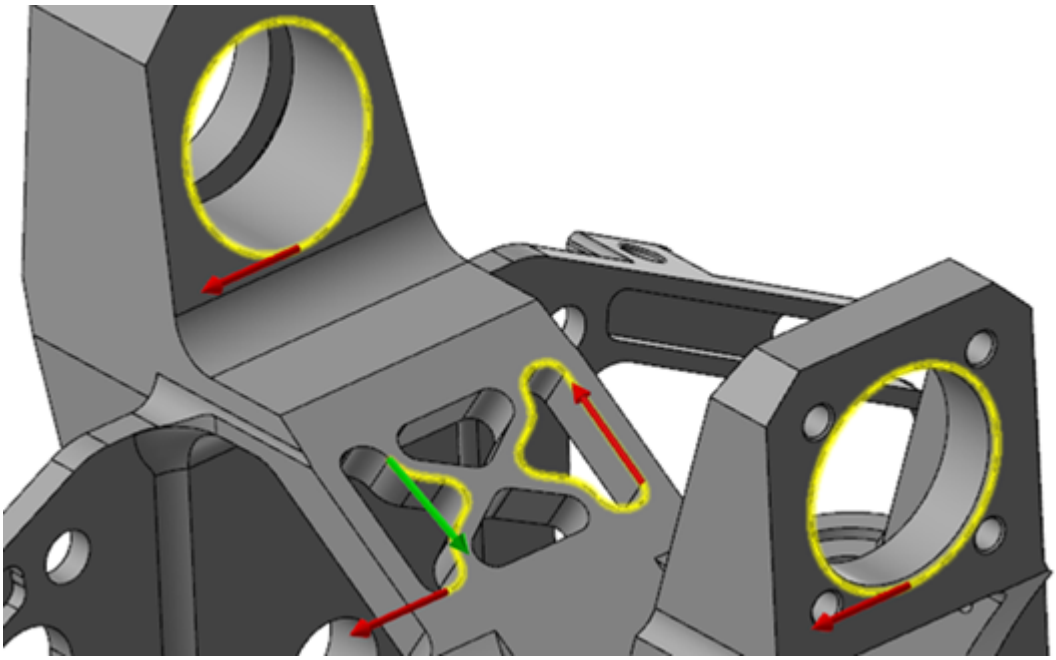
Listed below are enhancements made to solid chaining and wireframe chaining. Additionally, there are enhancements that affect both chaining modes.

Display All Selection Arrows

The **Display all selection arrows** button in the **Wireframe Chaining** and **Solid Chaining** dialog boxes have replaced the **Unselect all** button.

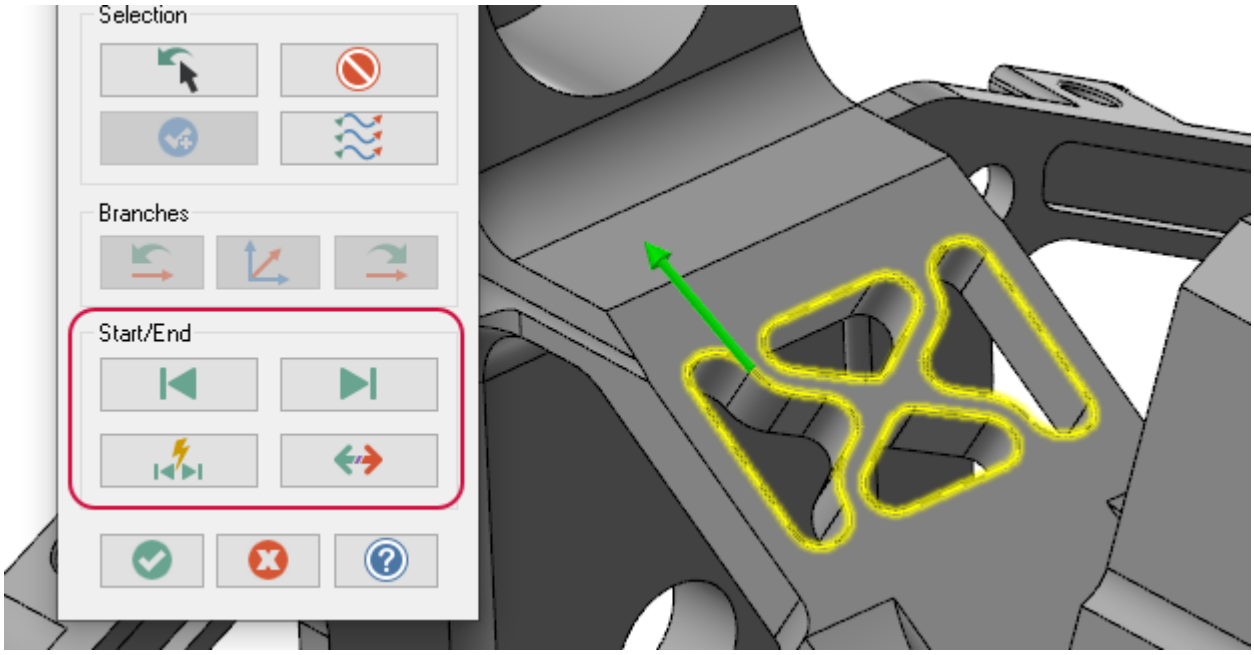


The new function is available only when you have created multiple chains. Clicking this function displays all chain selection arrows. Full loops display only one chain direction arrow; partial loops display both the start and end arrows.

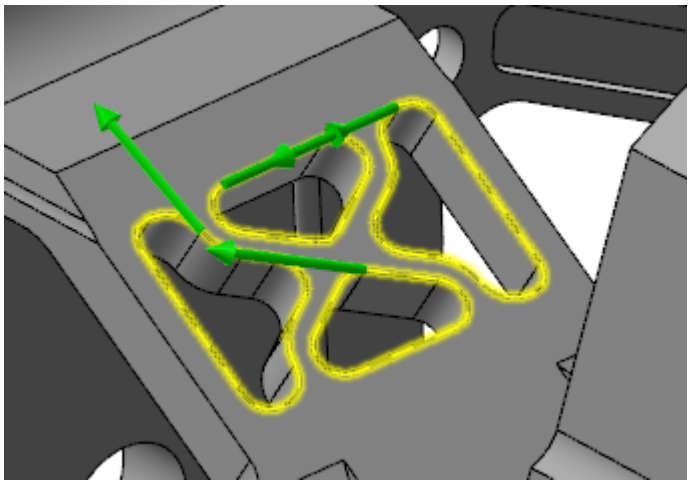


When selection arrows display, you can edit chains without leaving the **Chaining** dialog box. Select the start or end arrow in the graphics window to activate a particular chain.

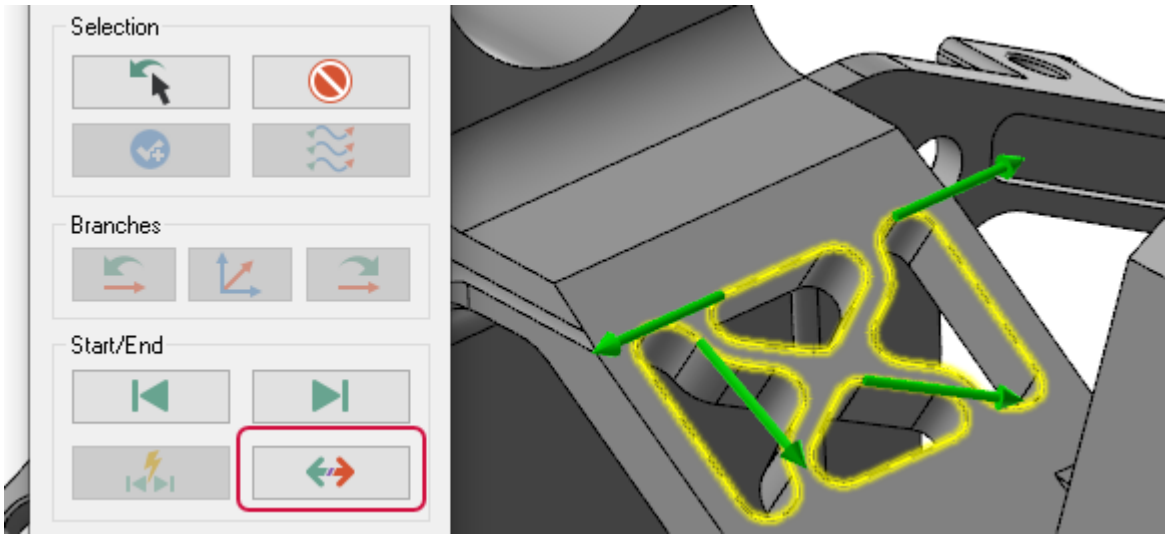
- Click an active chain to enable the **Start/End** controls in the **Wireframe Chaining** and **Solid Chaining** dialog boxes.



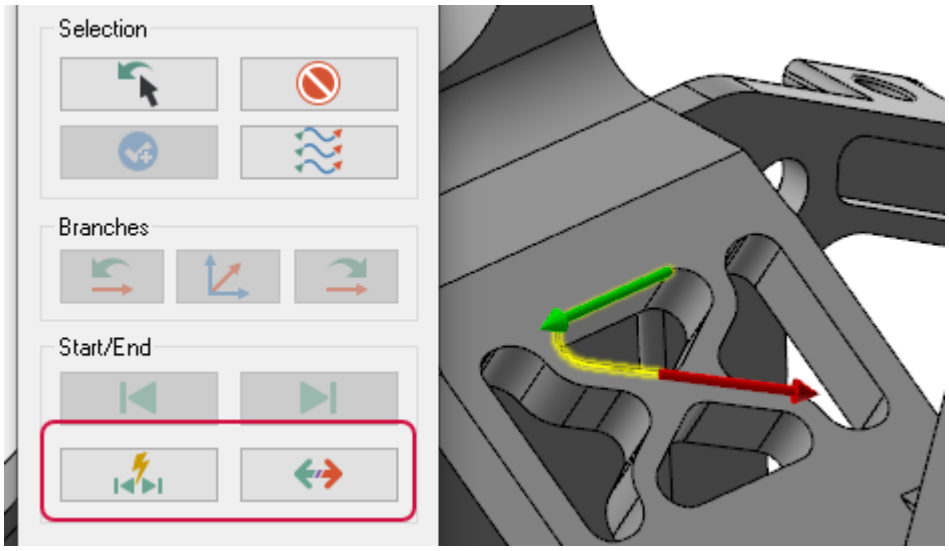
- After creating multiple chains, select **Display all selection arrows** to display all chain selection arrows.



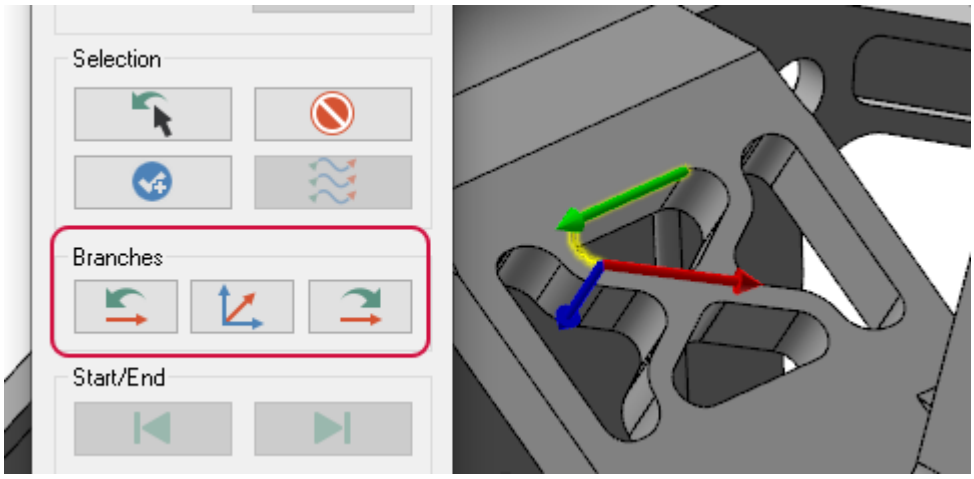
Then select **Reverse** to switch the direction of all active chains.



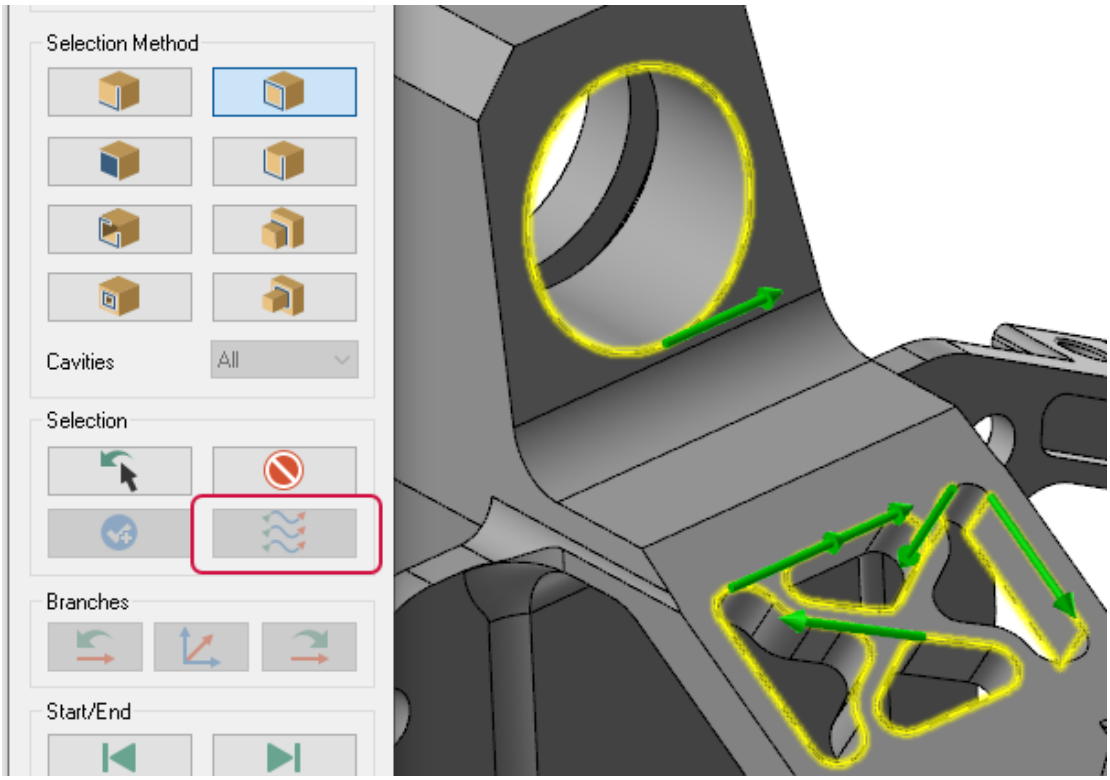
- Select a chain in the graphics window and select **Reverse** to switch the direction of the one chain you selected or select **Dynamic** to position the start or end of the chain at a location you choose in the graphics window.

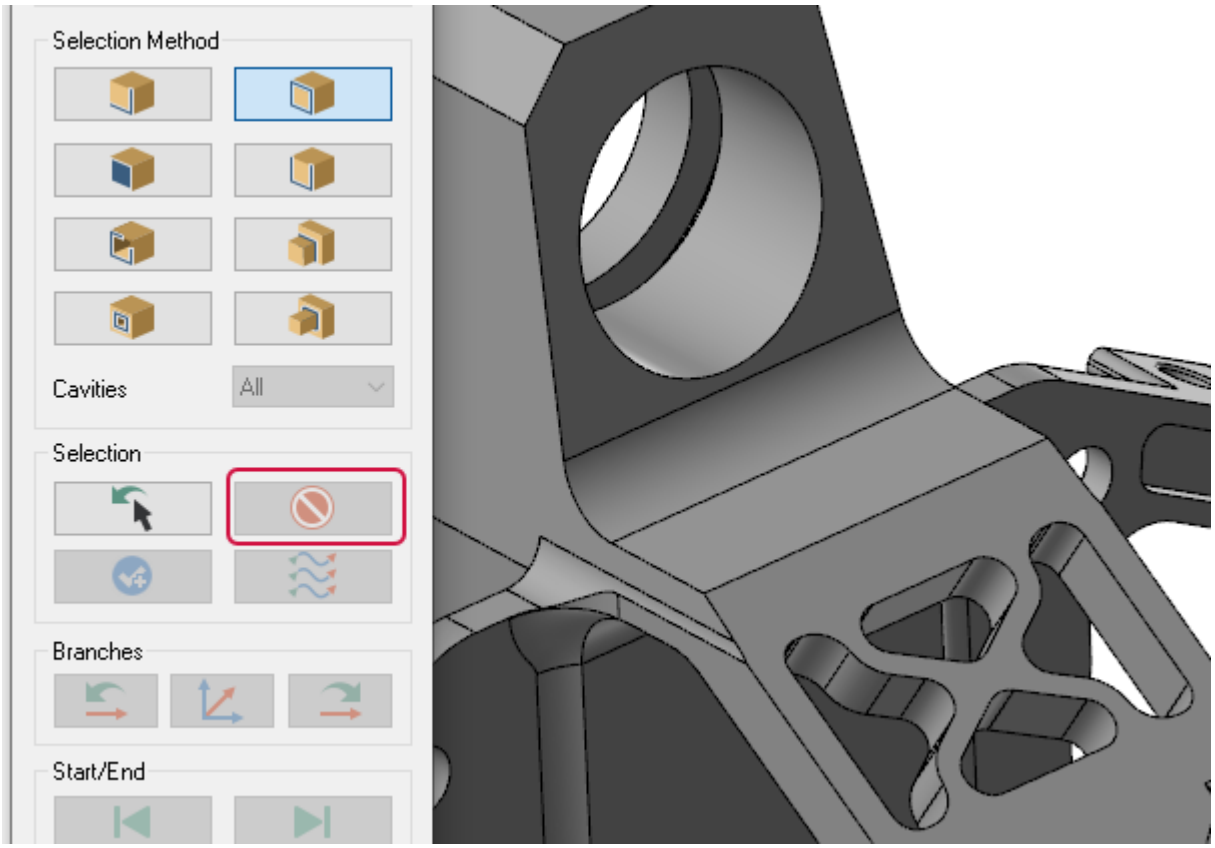


- Use the **Branches** buttons to guide the path of a chain that you select.

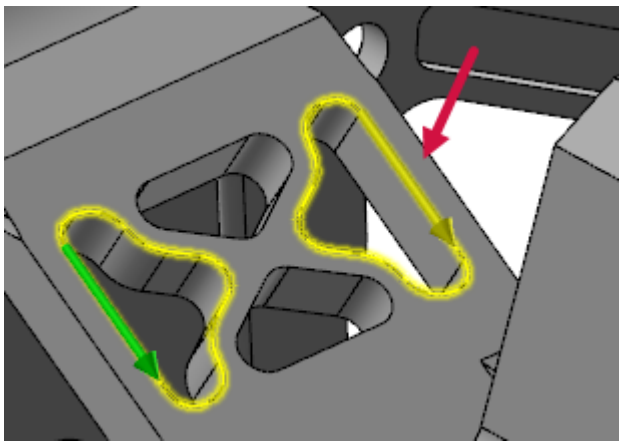


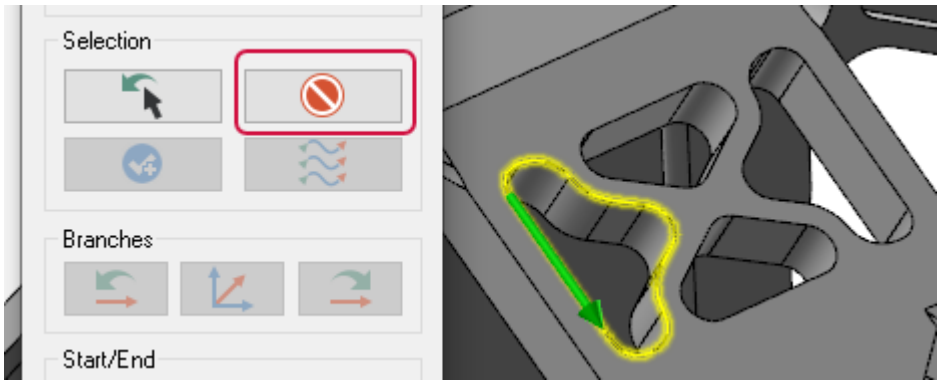
- Use **Display all selection arrows** in conjunction with **Unselect** to clear the selection of all active chains.





- Select one of the active chains and then **Unselect** to clear the selection of only that chain. This is helpful when you use window selection (or another selection method) to create multiple chains and need to remove one or more of them from the selection.





Chaining Modes Persist

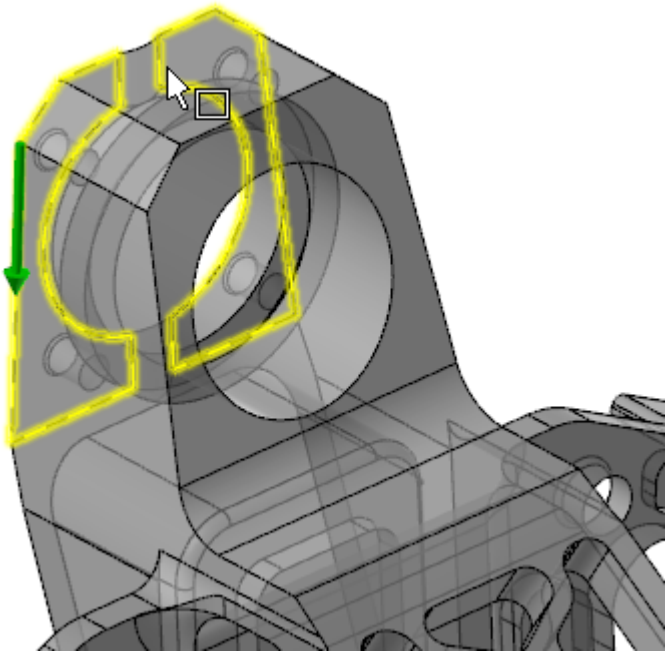
The last used settings for **Wireframe Chaining** and **Solid Chaining** dialog boxes now persist across Mastercam sessions. You no longer set the default chaining modes in the **System Configuration** dialog box. The **Default chaining mode** parameters have been removed from the **Chaining** page.

Solid Chaining Enhancements

Listed below are enhancements made to chaining when in **Solid** mode.

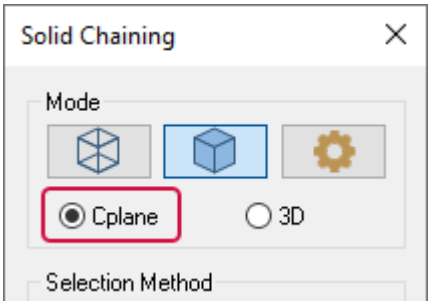
Back Side Loop Support

When selecting chains in translucent mode, Mastercam displays a loop associated with the face even if the face is on the back side of a part.



Selecting Chains Parallel to the Cplane

The new **Cplane** mode has replaced the **Face** mode in the **Solid Chaining** dialog box.

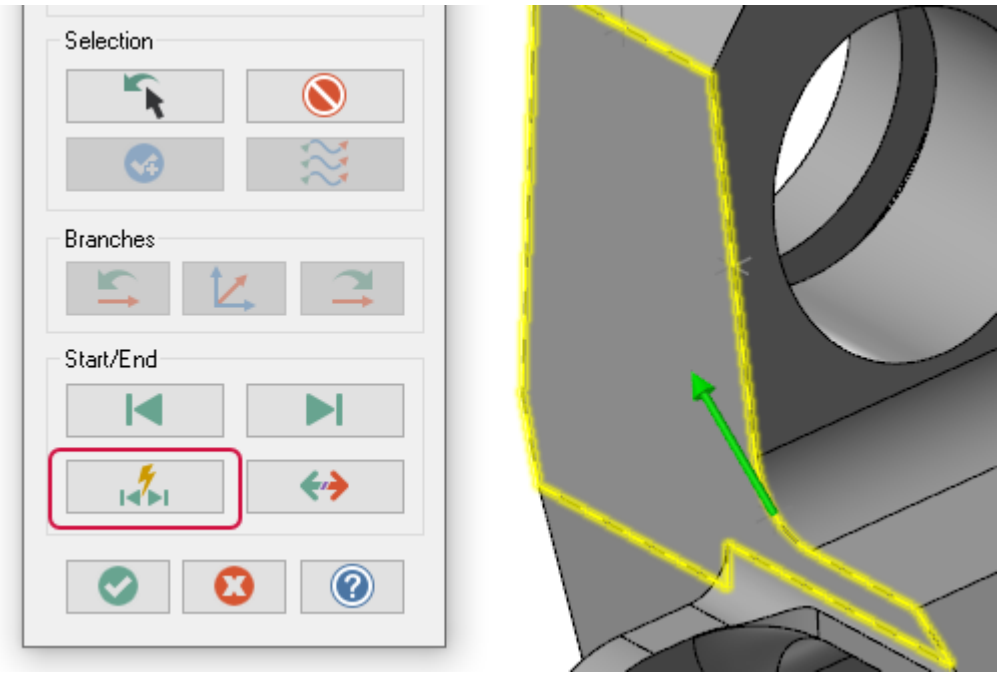


The functionality varies depending on the chaining method you use with the **Cplane** mode selected.

- When you select **Cplane** and use the **Edges** method of chaining, only edges parallel to the active **Cplane** are chainable.
- When you select **Cplane** and use the **Loop** method of chaining, only loops parallel to the **Cplane** are chainable.
- When you select **Cplane** and use the **Face** method of chaining, only faces parallel to the active **Cplane** are chainable. When enhanced selection, like double-click or triple-click, is used, then all faces at the same Z depth as the selected face are chained. When using window select, faces at the current Z depth are chained.

Dynamically Adjust the Start and End Point of Chains

The **Dynamic Start Point** option that is available for wireframe chaining is now available for solid chaining. Dynamic Start Point allows you to specify a new start point of a chain without first creating wireframe geometry. Dynamic Start Point maintains associativity with the solid.



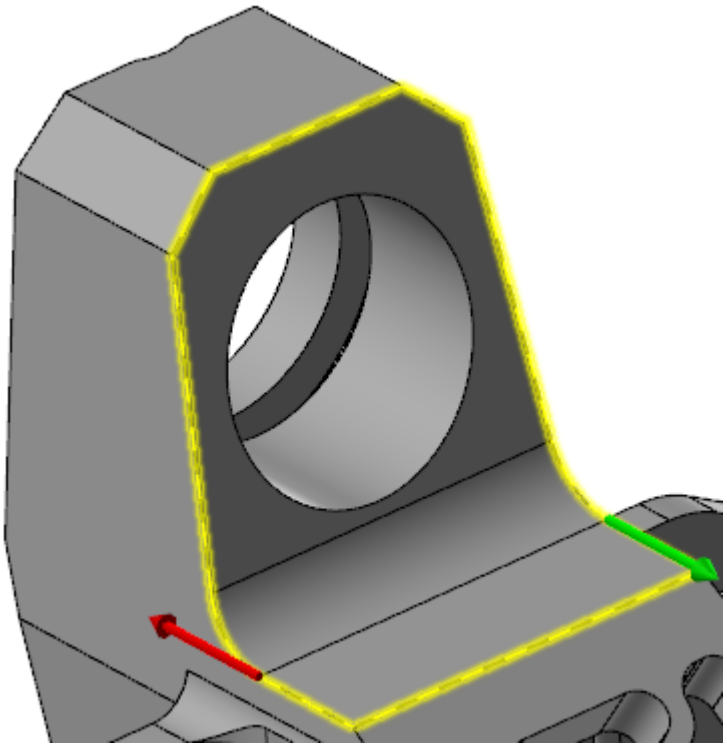
Lathe Chaining

Lathe solid chaining has been enhanced when chaining for a Lathe toolpath. Refer to "Lathe Solid Chaining Enhancements" on page 64 for more information.

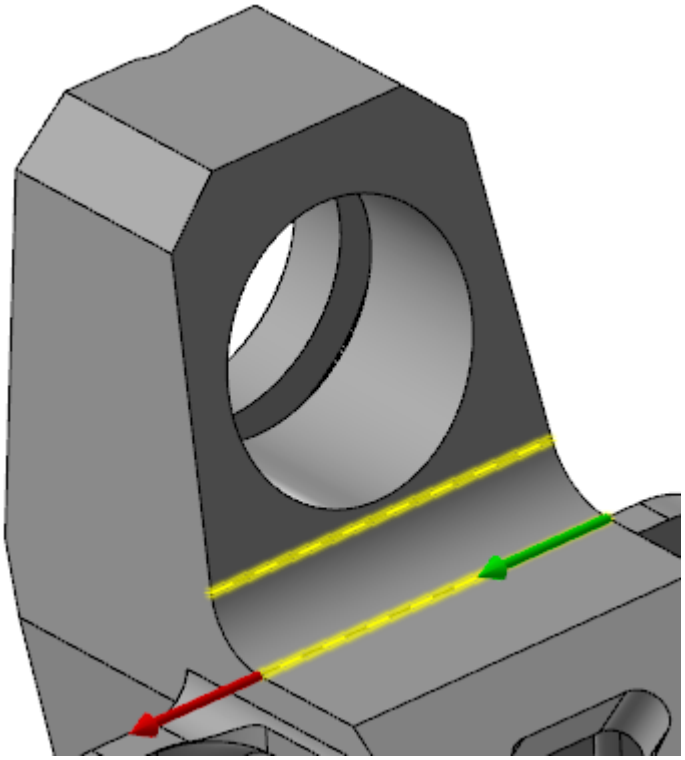
Selecting Outer Open and Shared Edges

The solid chaining modes, **Open Edges** and **Shared Edges**, were added to Mastercam 2020. In Mastercam 2021, these modes have been enhanced and renamed to **Outer open edges** and **Outer shared edges**.

When in **Outer open edges** mode, select a solid face to chain the open edges of the face in a clockwise direction.



When in **Outer shared edges** mode, select a solid face to chain the outer shared edges of the face in a counterclockwise direction.



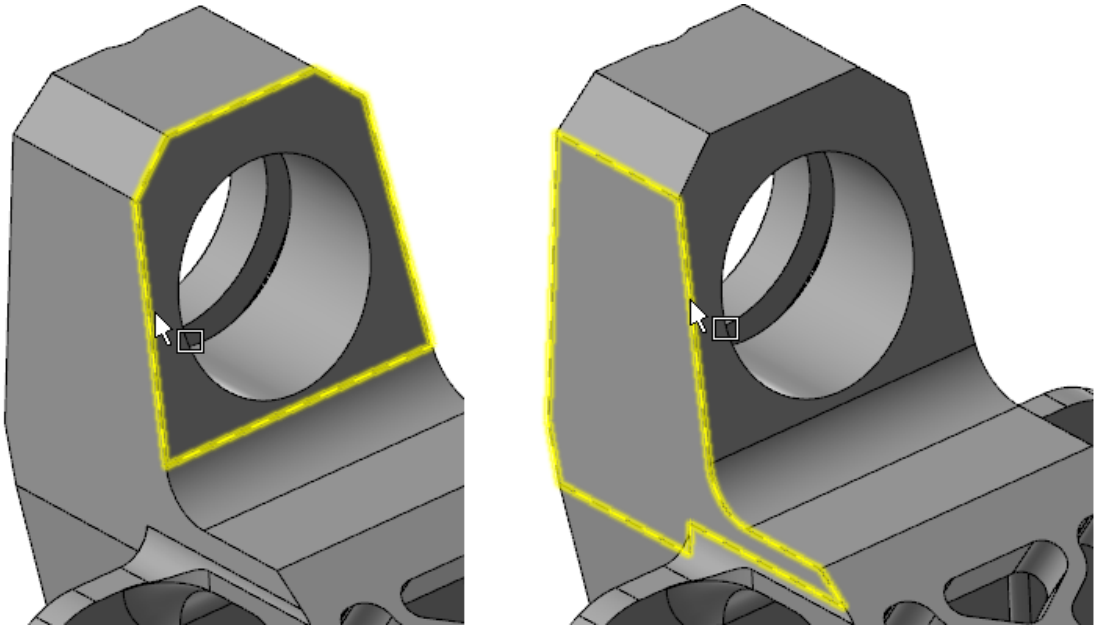
Multiple Selection Methods Added for Loop

Additional selection methods are now available when using **Loop** mode:

- [**Shift+click**] to select tangent faces.
- [**Alt+click**] to vector select.
- [**Ctrl+click**] to select matching fillets and holes.
- Window-select to select multiple faces.
- [**Ctrl+Shift+click**] to select similar faces.
- Double-click to select a feature.
- [**Ctrl+Shift+double-click**] to select similar features.
- Triple-click to select the solid body.

Improved Visual Support when Selecting a Shared Edge

The solid chaining workflow has been enhanced when using the **Loop** or **Partial Loop** methods. Mastercam no longer presents the **Pick a Reference Face** dialog box that allows you to cycle through the possible reference faces to choose the correct face to chain. In Mastercam 2021, you hover over the geometry until the correct loop displays.



Wireframe Chaining Enhancements

Listed below are enhancements made to chaining when in **Wireframe** mode.

Indicating Chain Direction for Selection Methods

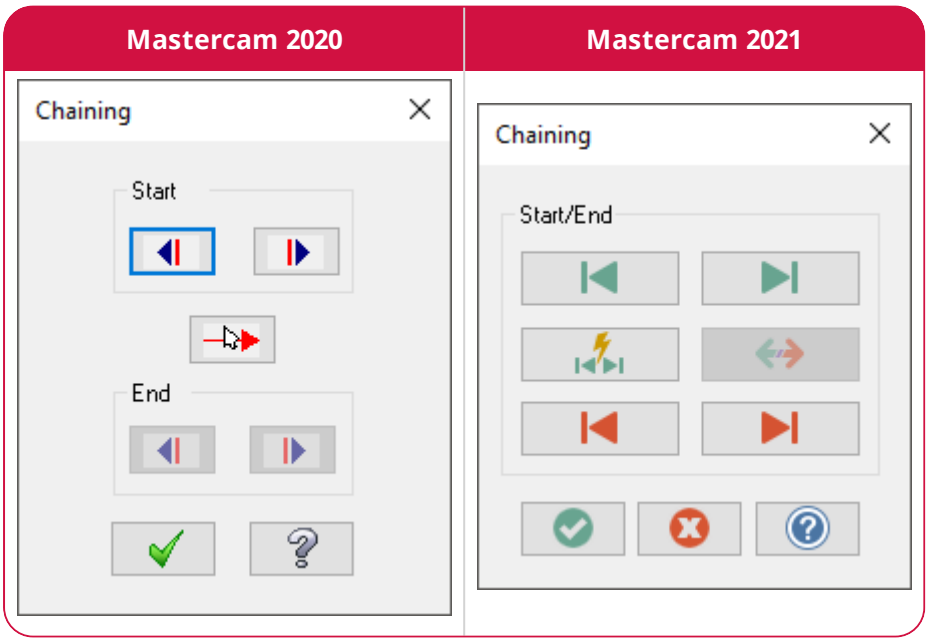
In previous versions of Mastercam, the chain direction was not indicated when using the Window, Polygon, Vector, and Area selection methods. Mastercam 2021 now indicates the chain direction when using these selection methods. In addition, if multiple wireframe geometry is chained, the **Reverse** option will switch the direction of all active chains.

Improved Visual Feedback Using Cplane Mode

When using **Cplane** mode, Mastercam now highlights and allows you to select only the entities parallel to the active Cplane.

Consistent Start and End Point Controls

The **Chaining** dialog box that opens when selecting **Start point** from the right-click menu of the Chain Manager has been updated. The controls are now consistent with the **Wireframe Chaining** dialog box.

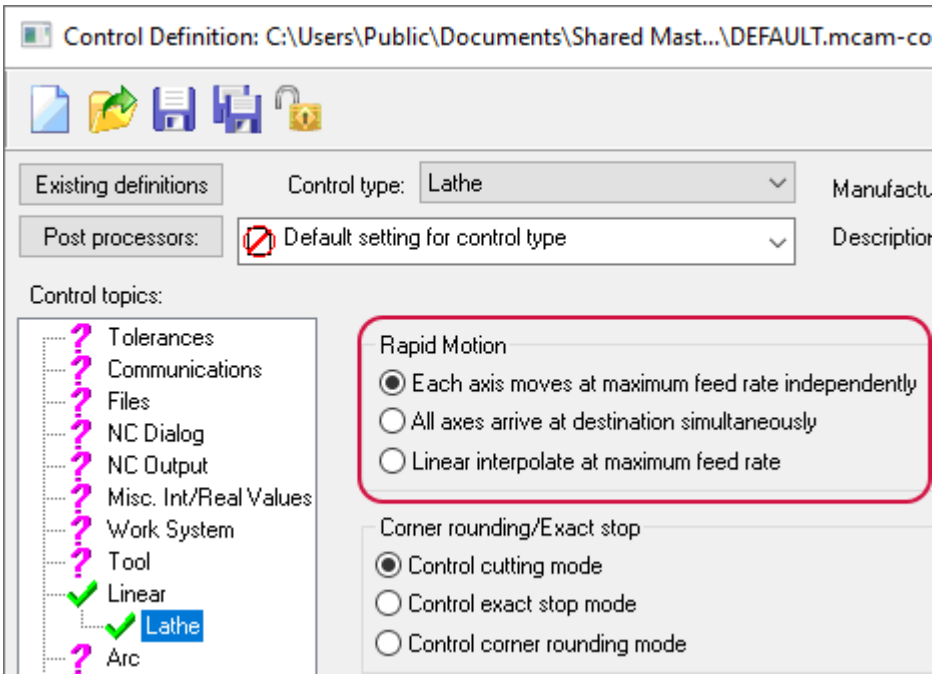


Mastercam Simulator Enhancements

Listed below are enhancements made to Mastercam Simulator. Additionally, peck drilling motion has been updated for Mastercam Simulator and Classic Backplot to better reflect the machine motion and provides a much more accurate cycle time calculation.

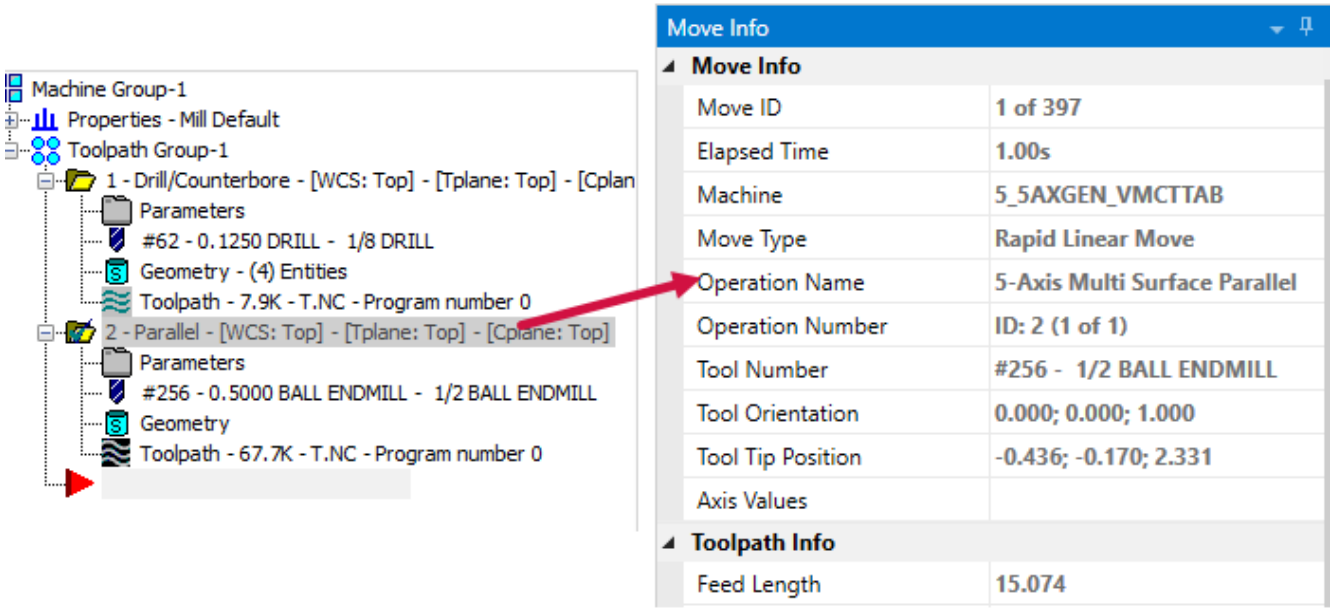
Displaying Lathe Reposition Moves

Mastercam Simulator now properly displays the selected **Rapid Motion** option located on the **Linear**, **Lathe** page in the **Control Definition** dialog box when simulating a lathe machine.



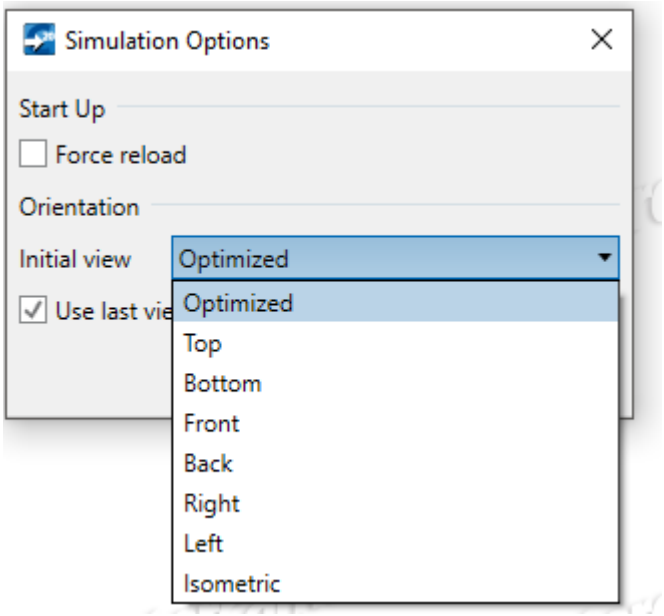
Display the Operation Name in the Move List

During simulation, the **Move** list now properly identifies multiaxis toolpaths.



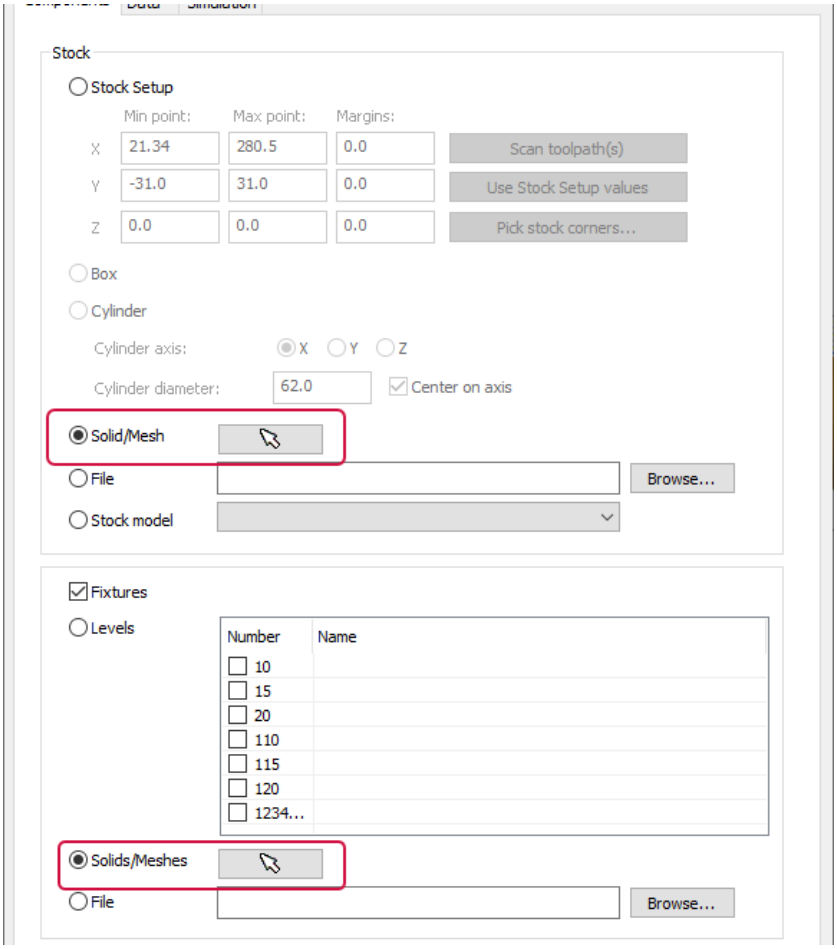
Optimizing the Initial View

If **Initial view** is set to **Optimized** in the **Simulation Options** dialog box when simulating a mill-turn toolpath, it now displays correctly in Mastercam Simulator.



Selecting Meshes for Fixtures or Stock

In the **Simulator Options** dialog box, you can now select meshes in addition to solids and surfaces to use as fixtures or stock.



Additionally, the new **Restore settings** button resets settings in the tab to their default values.

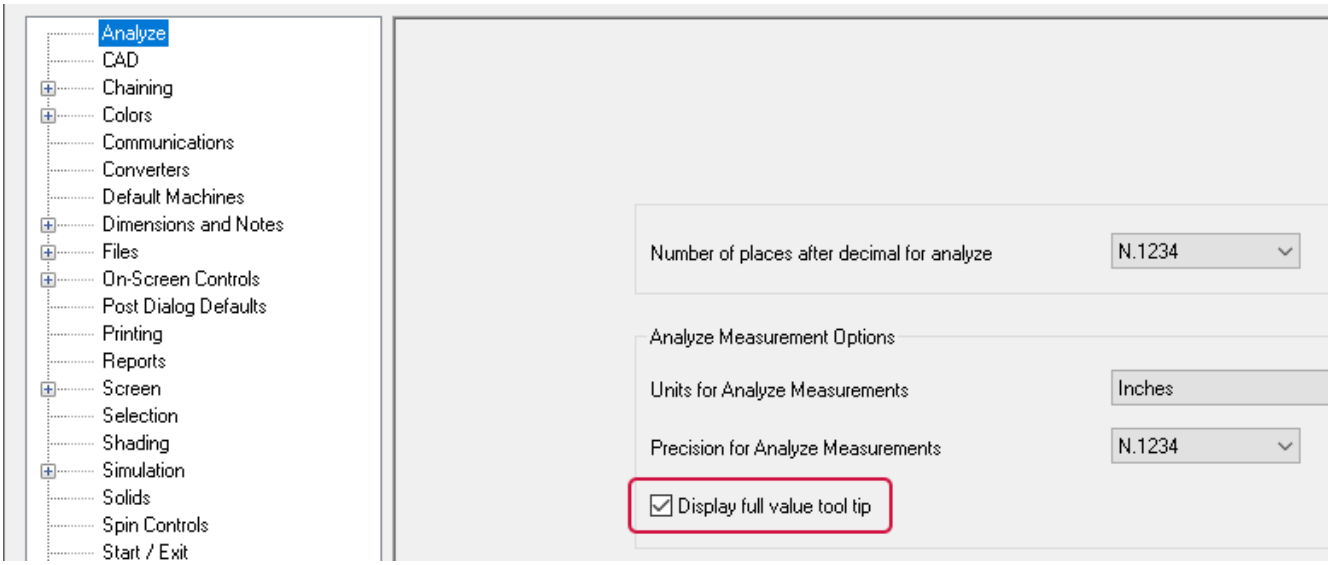
System Configuration Enhancements

Listed below are enhancements made to options in the **System Configuration** dialog box, located on the **File** menu.

Displaying a Full Parameter Value

In previous versions of Mastercam, the Analyze function limited the display of a measurement up to the precision you set in System Configuration dialog box or in the Analyze dialog box itself. In Mastercam 2021, you can now hover over a value in the **Analyze** dialog box and see the actual value without having to change the precision.

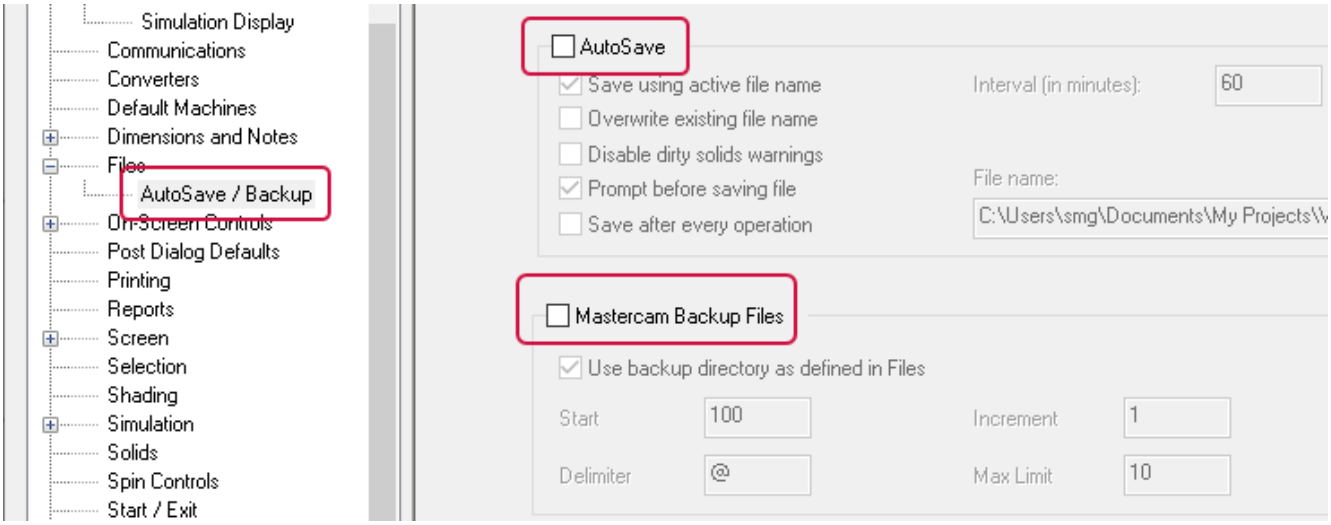
To see a measurement's full value, select the new **Display full value tool tip** option in the **Analyze** page.



- When selected, Analyze displays the full value of each number in a tooltip when you hover over the field even if it is greater than the display precision in that field. The checkbox is selected by default.
- When deselected, you must change the display precision to see the full value.

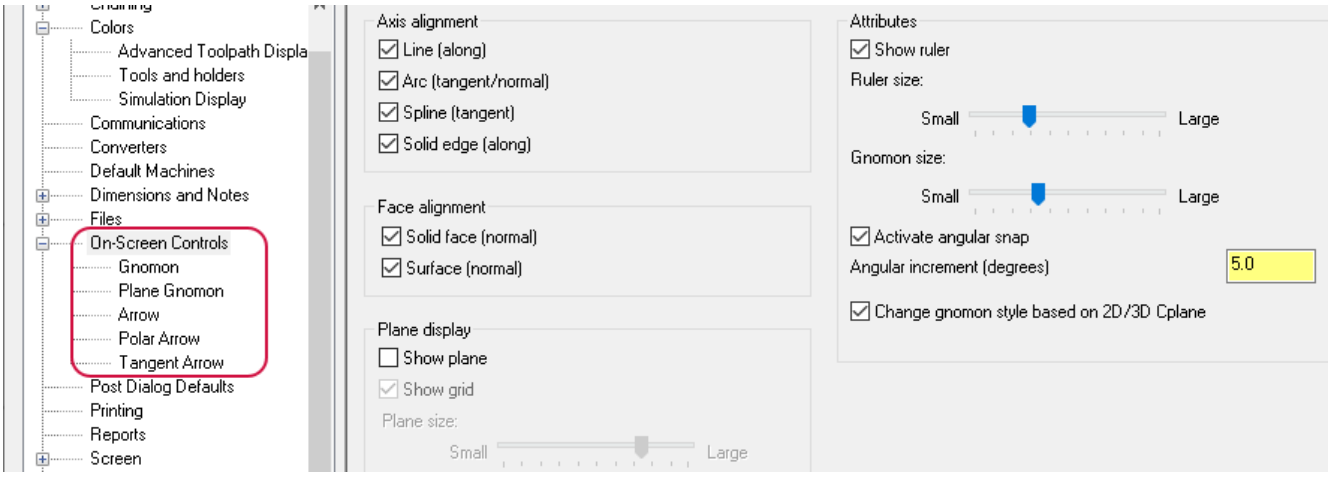
Combined AutoSave/Backup Options with System Configuration

The functionality of the standalone **AutoSave / Backup** dialog box has been merged with the **AutoSave / Backup** page of **System Configuration** dialog box. This provides a single location to configure the AutoSave options.



Modifying the On-Screen Control Settings for Gnomons and Arrows

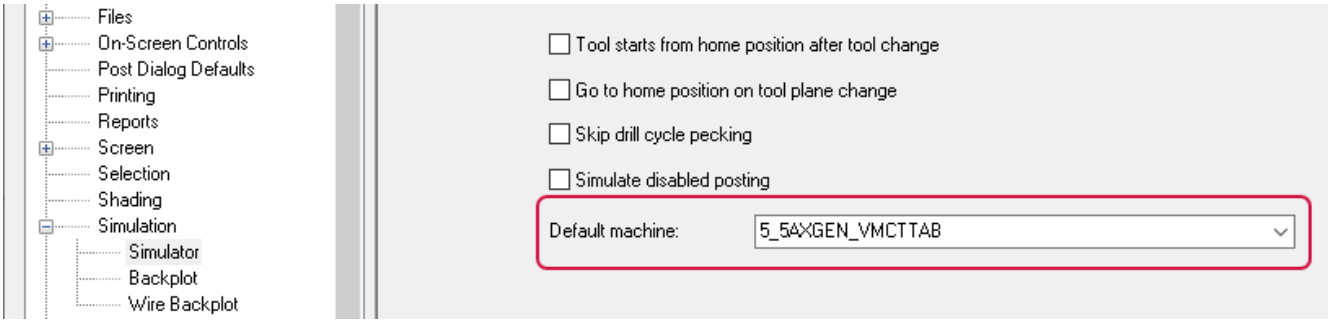
The on-screen control settings for gnomon, plane gnomon, arrow, polar arrow, and tangent arrow are now located under the **On-Screen Controls** page. You can access these settings by right-clicking the on-screen control in the graphics window or by clicking **File**, **System Configuration**.



Some parameter names and controls have been updated to maintain consistency across pages.

Setting the Default Machine for Mastercam Simulator

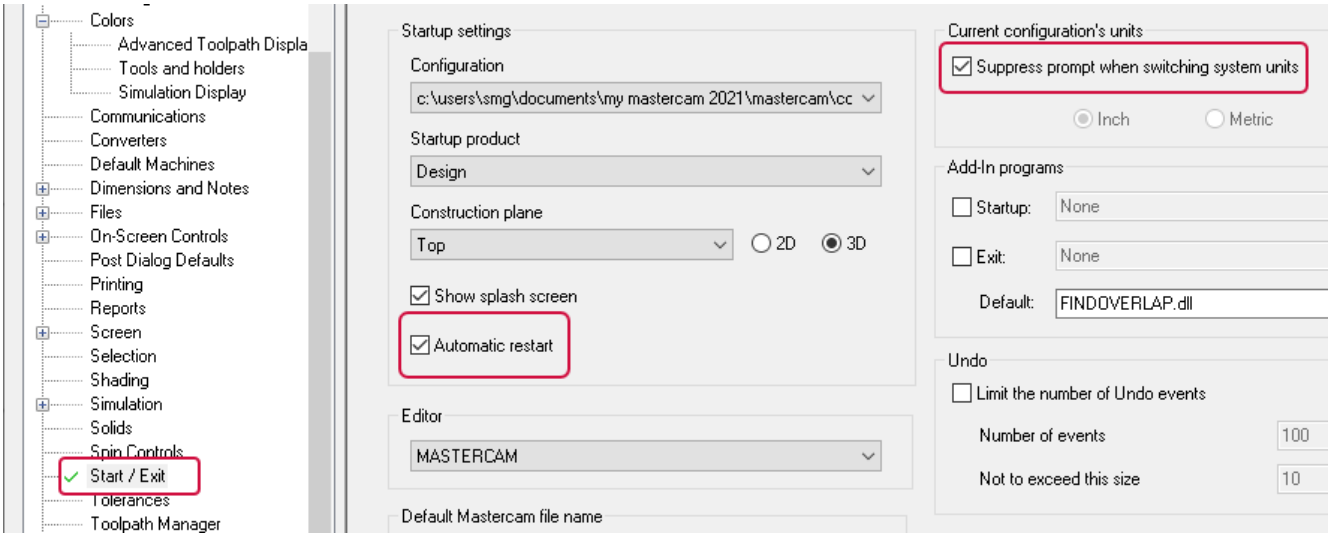
You now can set the default machine for Mastercam Simulator in **Simulation** mode. You can change the default machine on the **Simulator** page.



Disabling Automatic Restart After a Crash

The **Start / Exit** page now has an option to disable an automatic restart after a crash of Mastercam. The **Automatic restart** option is on by default.

The option to **Prompt when switching system units** has also been moved from Mastercam Advanced Configuration to the **Start / Exit** page. The new option, **Suppress prompt when switching system units**, applies to the current configuration (inch or metric) and is off by default.



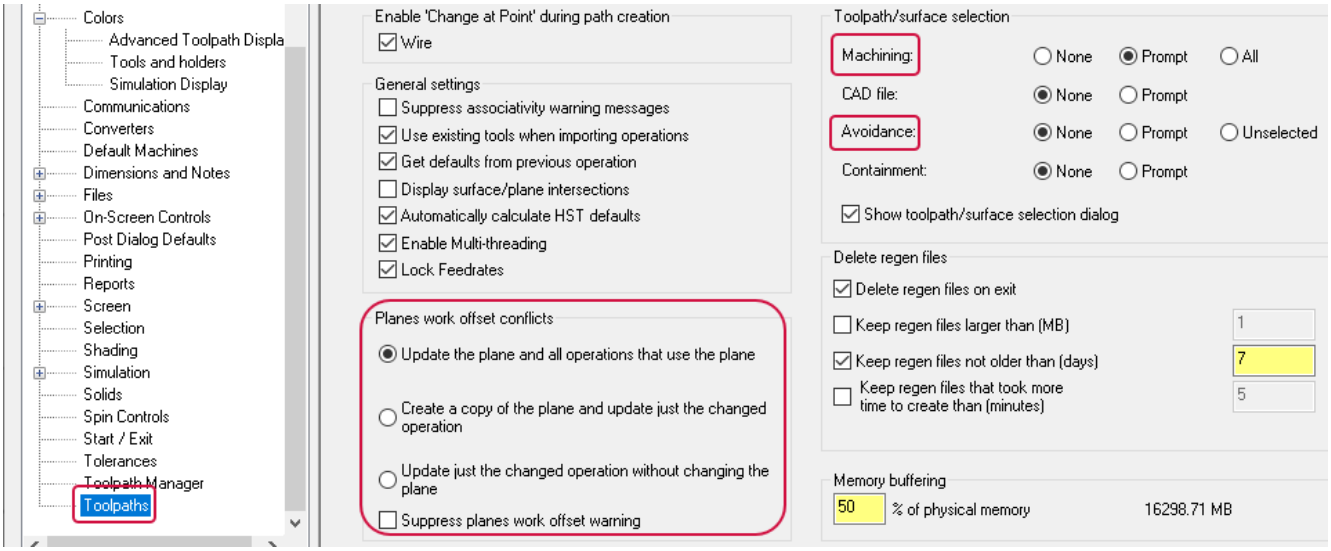
Work Offset Warning Options Moved to System Configuration

The option **Planes work offset warning** has been moved from Mastercam Advanced Configuration to the **Toolpaths** page and broken out into three separate options:

- Update the plane and all operations that use the plane.
- Create a copy of the plane and update just the changed operation.
- Update just the changed operation without changing the plane.

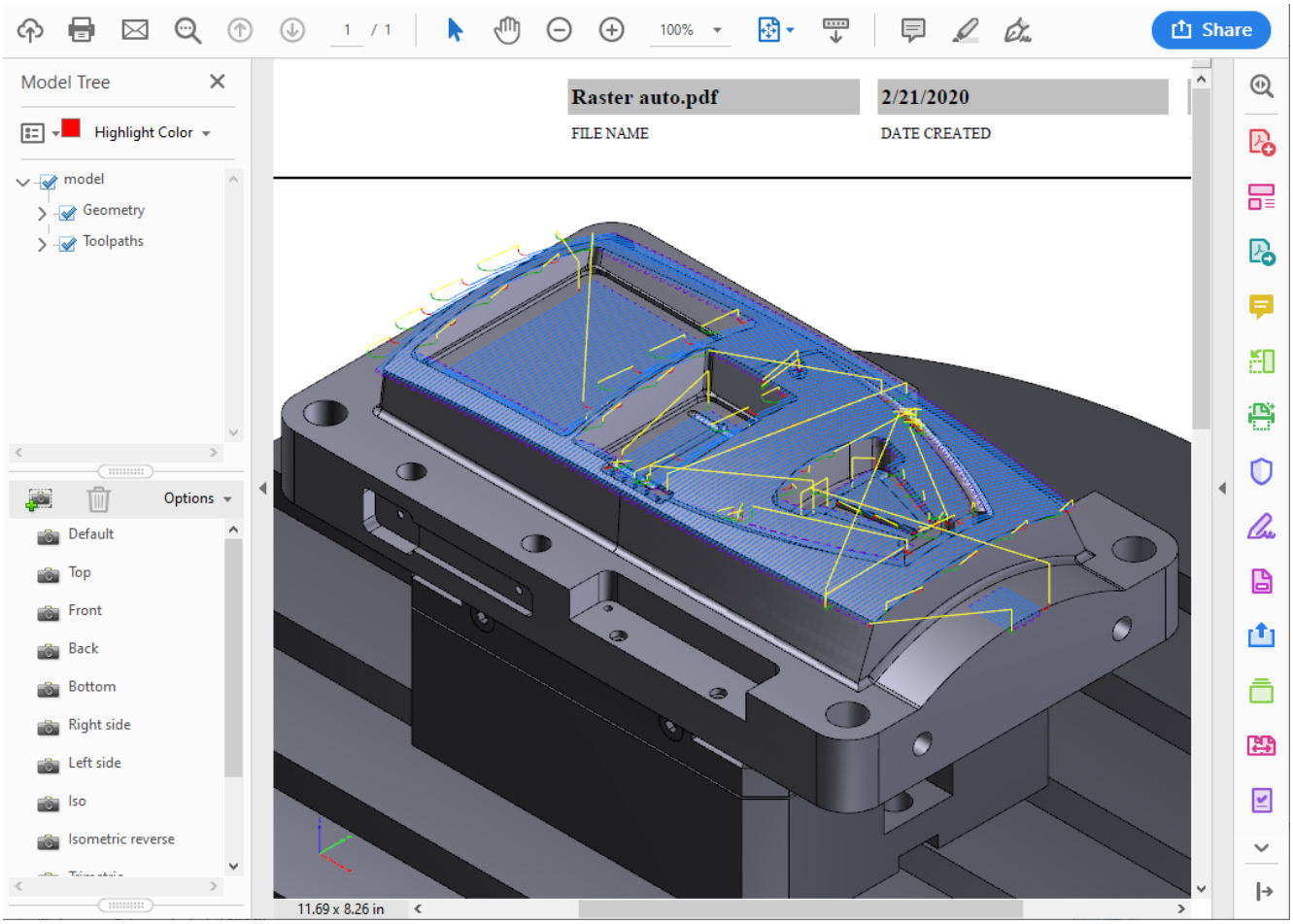
The new **Suppress planes work offset warning** option disables the warning message that displays when a work offset value has been changed. The option is off by default.

Two **Toolpath/surface selection** options have been renamed to match the options on the **Model Geometry** page for 3D high speed toolpaths. **Drive** is now named **Machining** and **Check** is now named **Avoidance**.



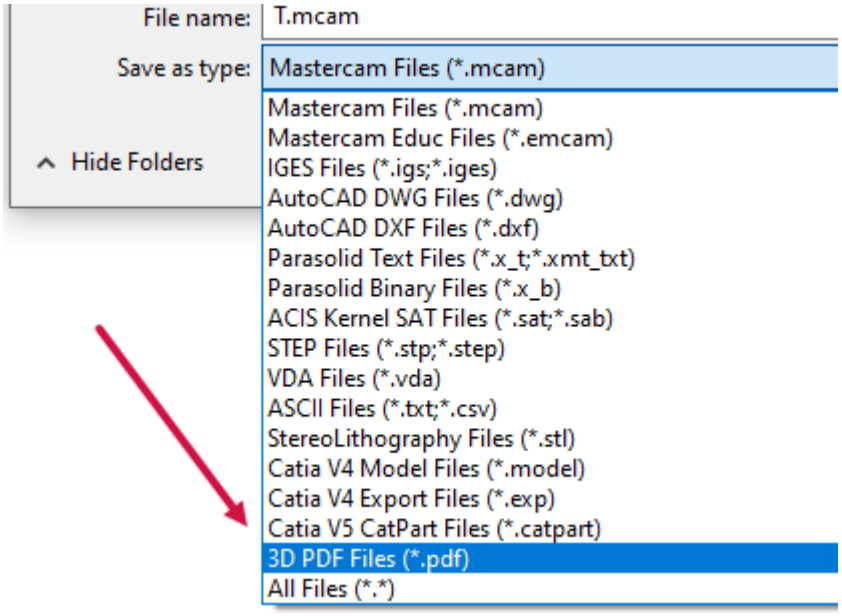
Saving Files as 3D PDFs

Mastercam 2021 now includes an option for you to save all the currently visible entities and toolpaths in your part as a 3D PDF. A 3D PDF is a dynamic PDF file that displays an embedded 3D model. Anyone with a 3D PDF-enabled viewer, such as Adobe® Acrobat Reader® (available for free from Adobe.com), can view and interact with the part in this format.

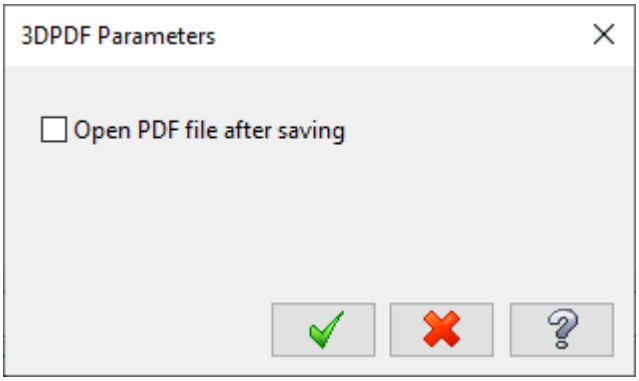


Mastercam preserves important part data such as planes, entities, levels, dimensions, and toolpath operations. When viewing the 3D PDF, you can control the display of this data by selecting or deselecting items in the PDF viewer's **Model Tree** panel. In addition to the standard PDF controls, you can use the mouse or Adobe's 3D toolbar functions to interact with the embedded model.

To save a file as a 3D PDF, choose **3D PDF Files (*.pdf)** from the **Save as type** drop-down menu.



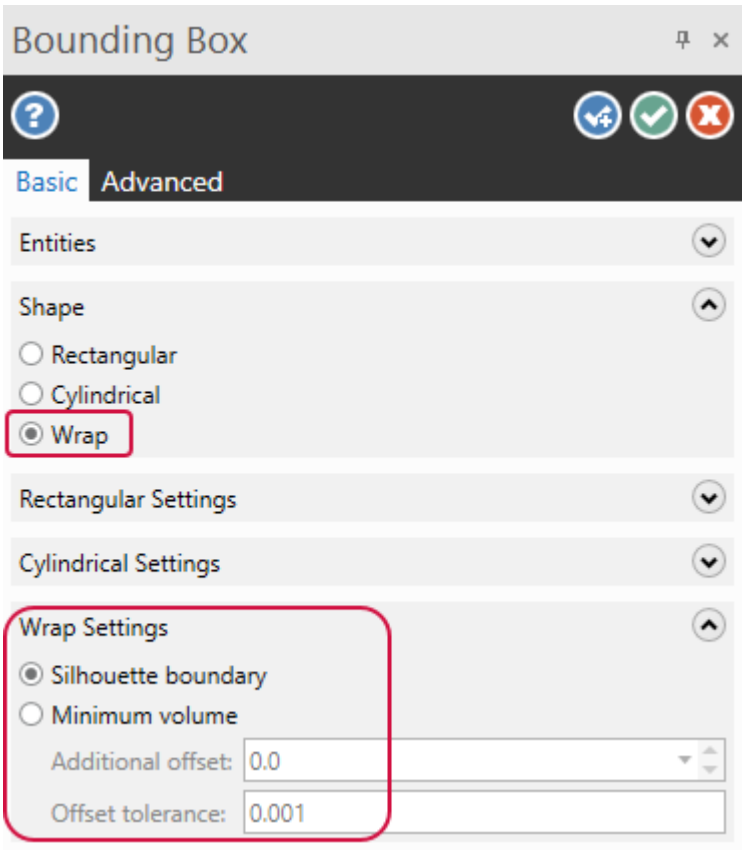
Select **Options** to open the **3DPDF Parameters** dialog box. Select **Open PDF file after saving** to open the PDF after saving the file.



You can also set this option as the default on the **Converters** page of the **System Configuration** dialog box.

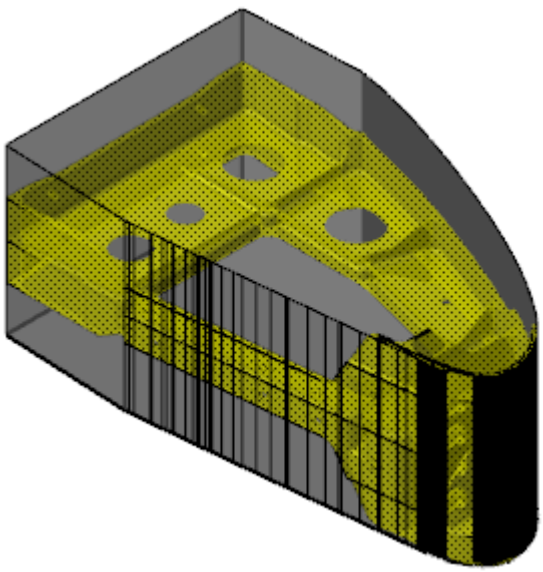
Wrapping Entities with Bounding Box

When creating a bounding box in Mastercam 2021, the new **Wrap** option allows you to create the smallest bounding box possible.

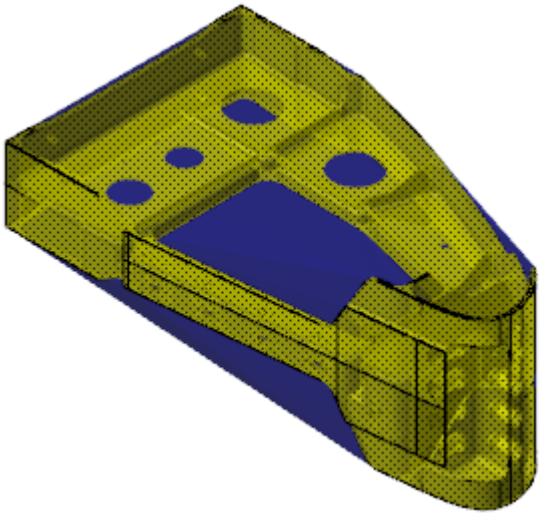


Select **Wrap** on the **Basic** tab in the function panel. **Wrap** aligns the bounding box along the largest, flattest face of the selected geometry. Then, choose how Mastercam generates the shape of bounding box from the following methods:

Silhouette boundary creates a boundary curve around the selected entities and then extrudes the bounding box as a solid or a mesh.



Minimum volume virtually drapes a mesh over the selected solid to create the bounding box. This results in a bounding box with minimal volume. You can then set an **Additional offset** and **Offset tolerance**.

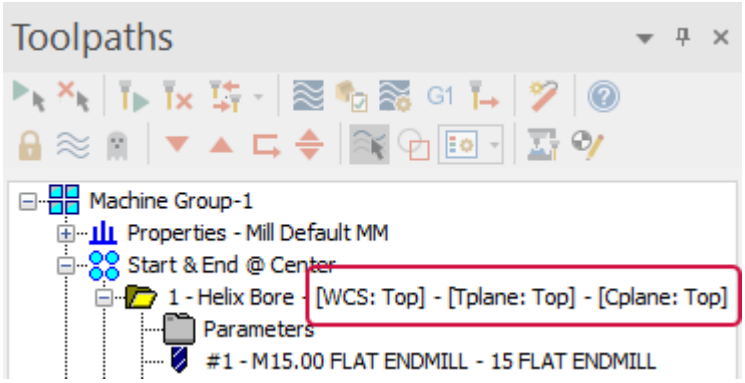


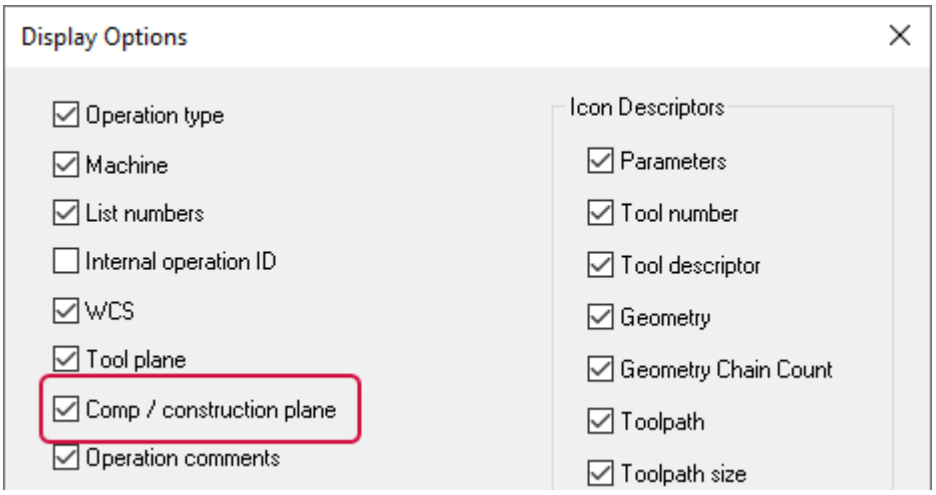
The **Additional offset** parameter allows you to specify a 3D offset value to add dimension to the bounding box. The **Offset tolerance** value adjusts the accuracy of the offset bounding box.

Additionally, the **Auto** option under the **Orientation** group now always provides the smallest possible result for the selected **Shape**. **Auto** also now supports all Mastercam entity types instead of only solids.

Displaying the Cplane of a Toolpath

You can now display the Cplane of a toolpath in the Toolpaths Manager. This is controlled by the new **Comp / construction plane** option in the **Display Options** dialog box, which can be accessed by right-clicking in the Toolpaths Manager and selecting **Display options**. The option is off by default.

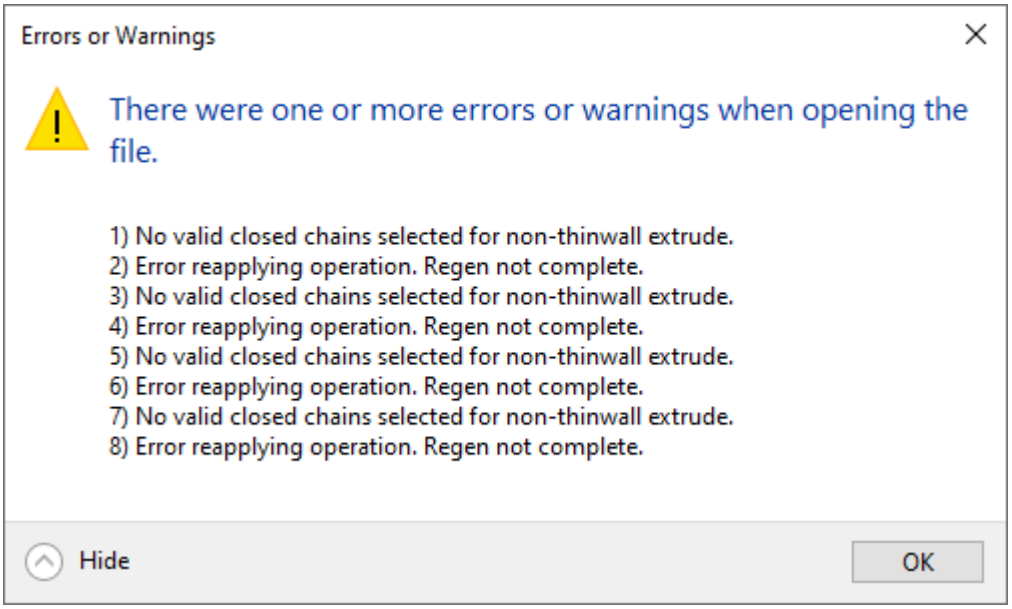




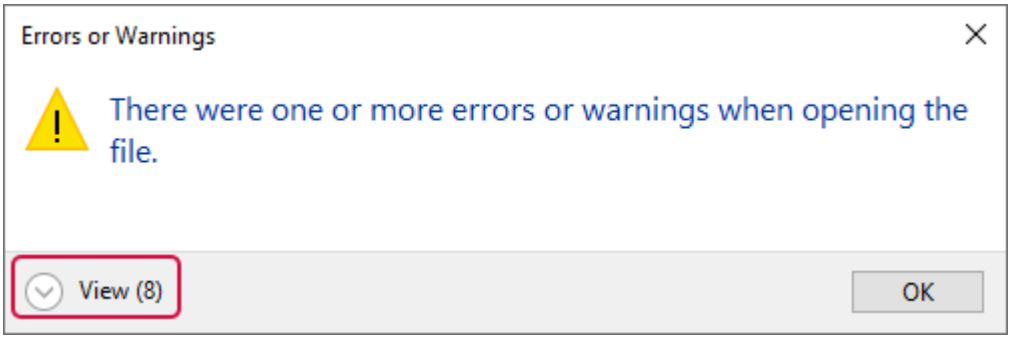
Improved Handling of Error and Warning Messages

In previous versions of Mastercam, some error or warning messages that occurred when opening a file required you to confirm them by clicking **OK** before proceeding. Now, in Mastercam 2021, these error and warning messages are collected and displayed in a single window for you to review. Files with messages that previously required a confirmation will now load in Mastercam without your intervention. Errors that require you to take action before proceeding will still prevent the file from opening until resolved.

If fewer than three confirmation-only errors or warnings occur when opening a file, the messages automatically display in the dialog box. Click **OK** or [**Esc**] to dismiss the dialog box.



If there are more than three errors or warnings, the messages are hidden.

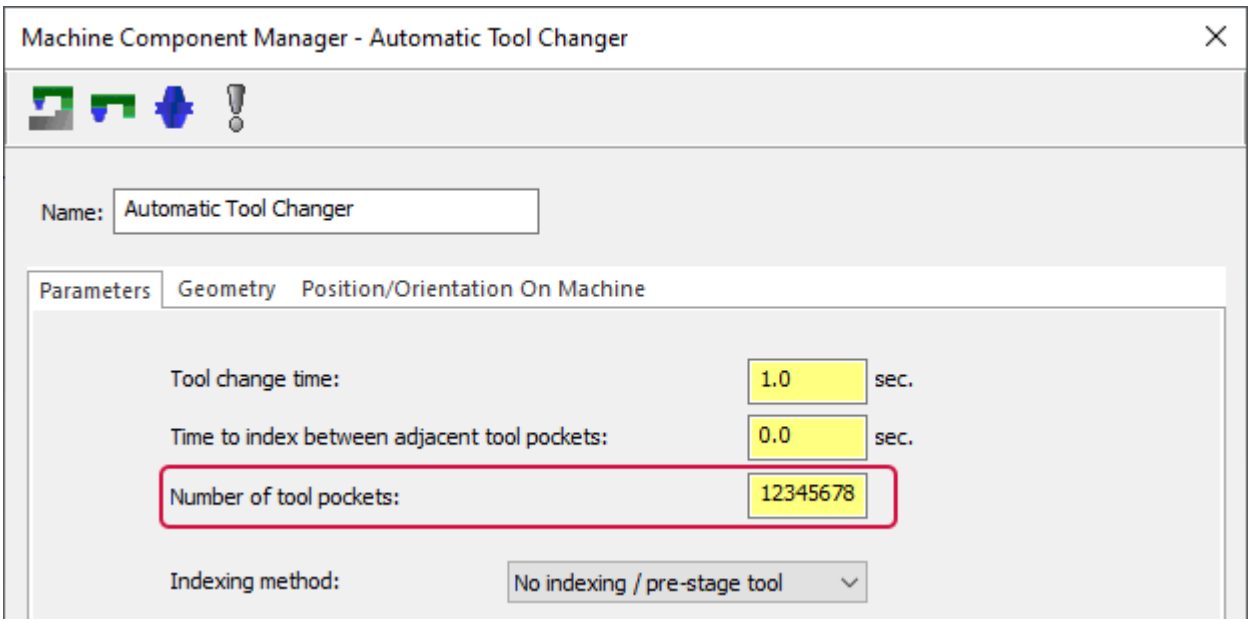


Click **View** to show the messages. Click **OK** or press [**Esc**] to dismiss.

> **NOTE**
> These messages also occur when generating and regenerating operations.

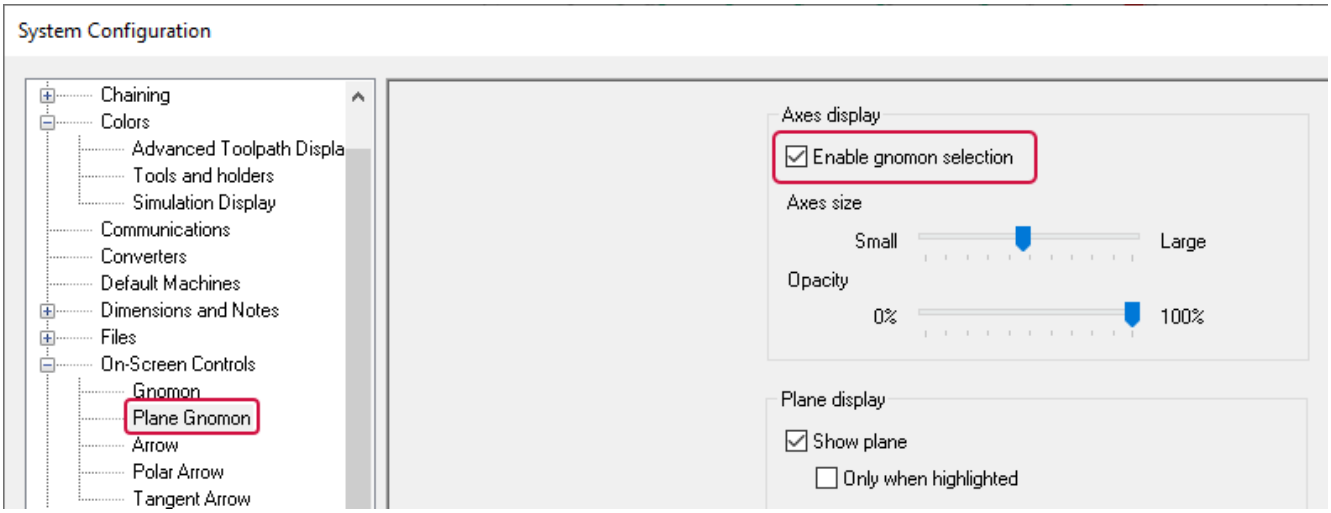
Increasing the Allowed Tool Pockets

The number of allowed tool pockets for an Automatic tool changer (ATC) component has been increased from 32000 to over 2 billion. Since Mastercam uses this value to clamp the allowed tool numbers when programming your part, the new limit lets Mastercam machine definitions accommodate modern tool number formats, such as group numbers that require 8 integer digits.



Enabling and Disabling the Selection of Plane Gnomons

You can now enable and disable the selection of plane gnomons using the **Enable gnomon selection** option in the **Plane Gnomon** dialog box or the **Plane Gnomon** page in the **System Configuration** dialog box. This option is selected by default.



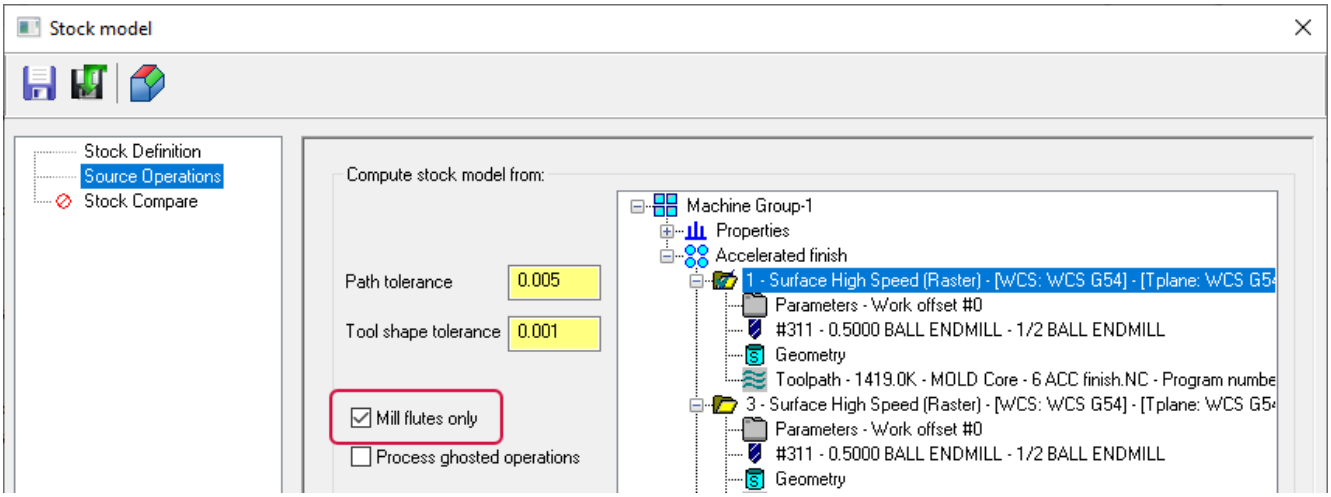
When deselected, plane gnomons will not highlight when you hover over them and cannot be selected. Disabling the selection of plane gnomons is helpful when you need to select geometry that is behind the gnomon. You can temporarily disable this option by holding down the [**G**] key. This allows you to select any geometry that is behind the gnomon.

Default Naming of Stock Models

There are two new enhancements to stock model creation. There is now a default name for stock model operations, so you do not have to enter a name manually. The default **Name** is **1**. Any subsequent stock model operations are incremented from that value.

> **NOTE**
> If you bring in a file from a previous Mastercam version that contains multiple stock models with the same name, an underscore and number (for example, **_4**) is added to the name after you open the operation's parameters page and click **OK**.

Additionally, the **5-axis tip only** checkbox has been renamed **Mill flutes only**.



When selected, **Mill flutes only** uses just the fluted portion of the tool to compute against the stock model.

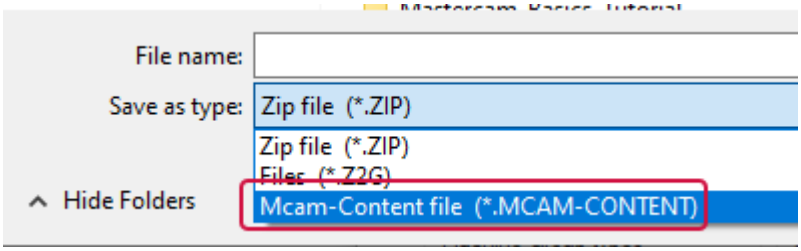
Enhanced CATIA Support

With this release of Mastercam, we are modifying the installation of the CATIA translators. While a license of the appropriate CATIA add-on is still required to import and export CATIA files, Mastercam no longer includes a separate installation. The Common\CatiaData subfolder, which used to be installed via that separate CATIA installer, is now automatically installed when Mastercam is installed.

Because of a re-design of the CATIA V5 importer and exporter, this subfolder is now much smaller than the subfolder which had been installed via the separate CATIA installation in the past. Also, as part of the re-design of the CATIA V5 importer, both the CATIA V5 Read and CATIA V5 Read with PMI licenses will now import PMI/3D Annotation data.

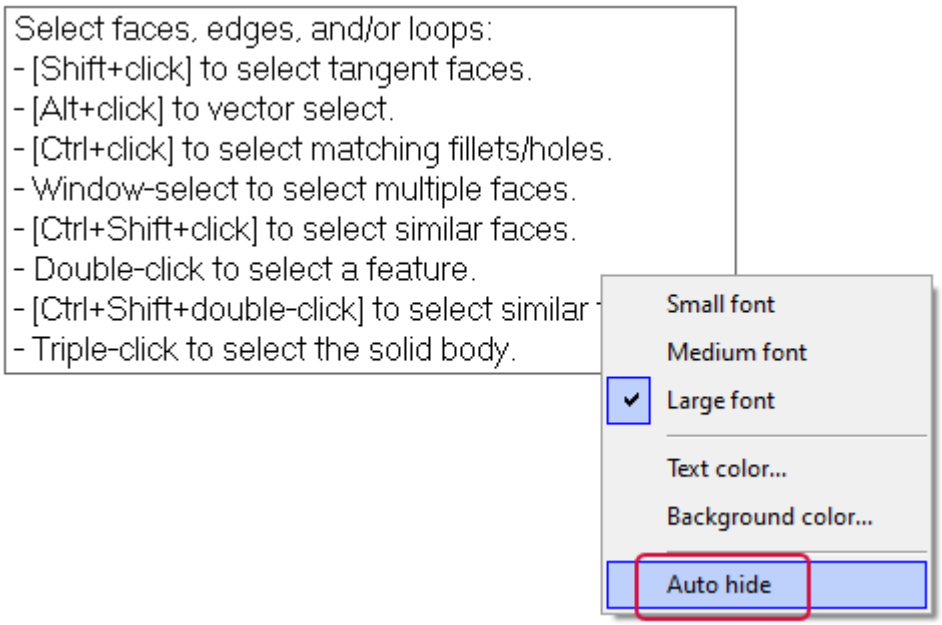
Creating .Mcam-content Files with Zip2Go

Zip2Go now creates an ***.mcam-content** file, in addition to creating ***.ZIP** and ***.Z2G** files. After choosing **File**, **New** in the **Zip2Go** dialog box, you have the option to select a different file type in the **Save file** dialog box.

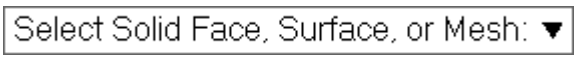


Auto Hiding Large Prompts

In Mastercam 2021, you can right-click a large prompt window and choose **Auto hide** to minimize the prompt to a single line. This helps when a large prompt window obscures areas of the graphics window.

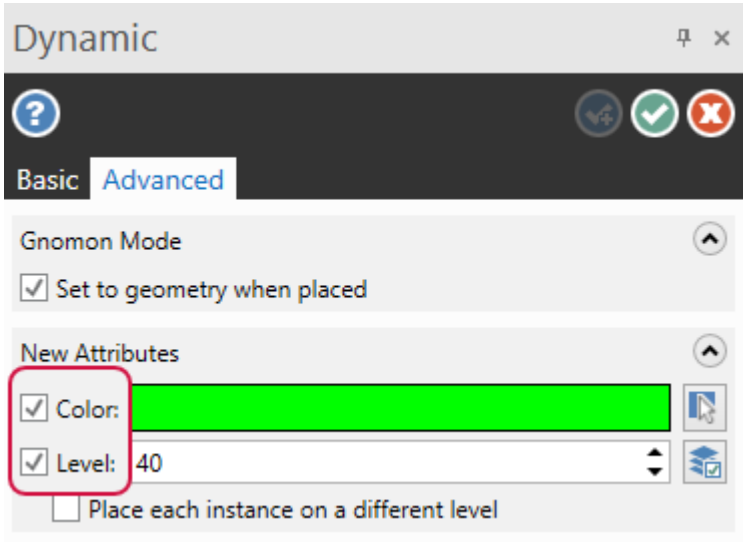


A small down arrow indicates that the prompt is minimized. Hover over the prompt to briefly expand it to full size, or right-click and deselect **Auto hide** to show the full prompt.

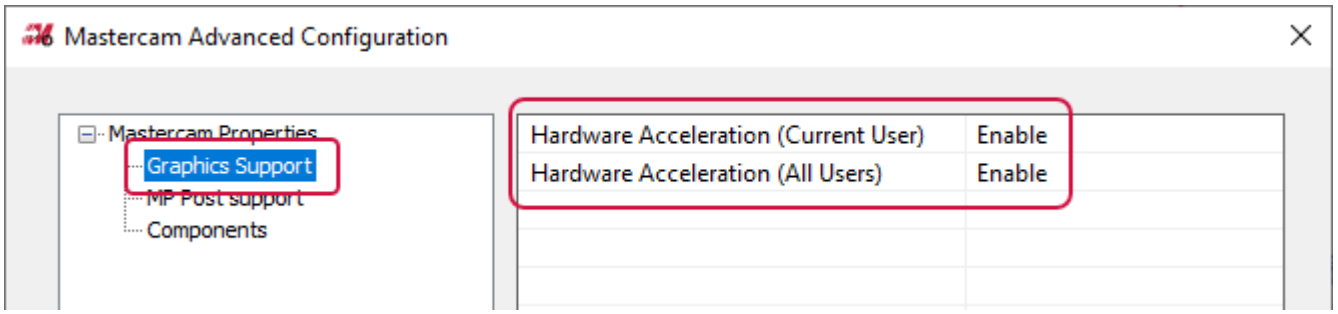


Miscellaneous Enhancements: General

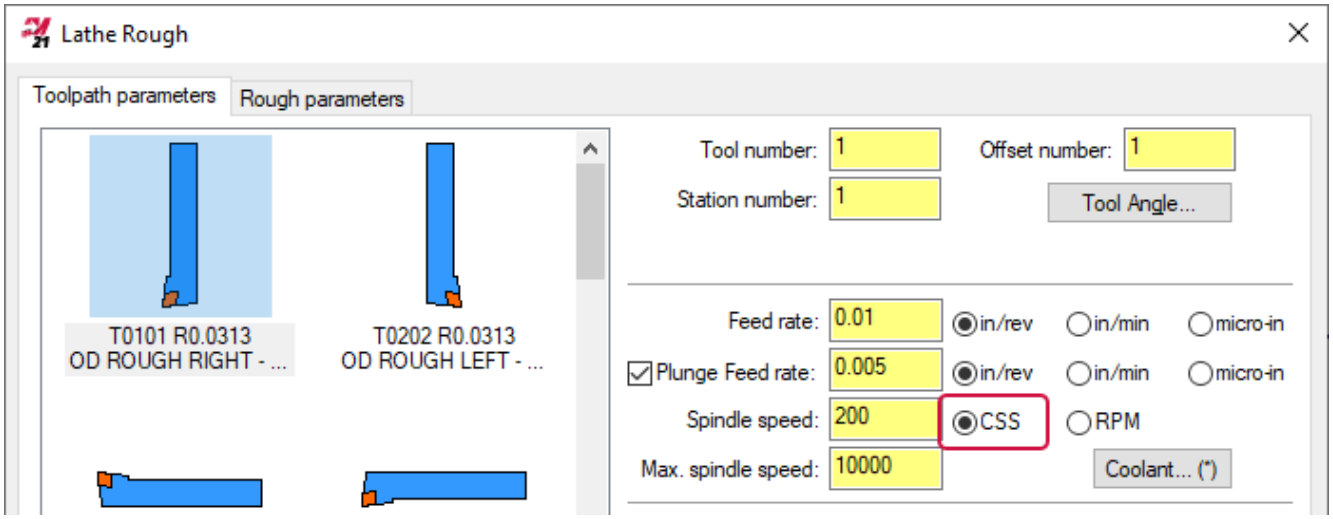
- To improve performance, you now have the option to ignore hidden entities in CATIA V5, ProE/Creo, and Unigraphics/NX files when importing these files into Mastercam.
- Trim to Entities ignores solid edges when wireframe selection is possible. The first pick must always be wireframe.
- The Ras2Vec function's algorithm has been improved, providing cleaner and more accurate results.
- The **File type** drop-down in the **Open** and **Merge** dialog boxes for Alibre files (**.ad_prt*, **.ad_smp*) has been renamed to Alibre Design. Formally listed in these dialog boxes as Alibre/Geomagic Design, this change reflects the acquisition of the Alibre Design product line by Alibre, LLC.
- You can now independently control the color and/or level of transformed entities.



- **Hardware acceleration** has been moved from the **Mastercam Launcher** dialog box to the **Graphics Support** page in the **Mastercam Advanced Configuration** dialog box. This option is now more accessible for you to enable or disable graphics card hardware acceleration.



- Classic Backplot's cycle time report has been improved for Lathe toolpaths with **CSS** (constant surface speed) selected on the **Toolpath parameters** tab.



- When creating an Agie 4-axis wirepath, instead of allowing you to have a cut point without a thread point, Mastercam now displays an error stating that you must chain a thread point, followed by the chain, and the cut point.

POST

Listed below are the new post and released machines for Mastercam 2021.

MP 5X Posts Released

- Haas UMC-750 5X Mill
- HEIDENHAIN TNC 5X Mill - TNC530/620/640
    - Supported configurations:
        - Table/Table — AC
        - Table/Table — BC
        - Head/Head — AC
        - Head/Head — BC
        - 3X
- Makino D200Z 5X Mill - Pro6 (Fanuc30i-32i)
    - Supported configuration: Table/Table — BC
- Robodrill 5X Mill - Fanuc30i-32i
    - Supported configuration: Table/Table — BC

Machine Environments: Lathe

Available for purchase with a Mill + Lathe license.

Machine Series	Control
Doosan Lync	Fanuc i
Haas ST	Haas CNC
Haas DS	Haas CNC
DMG Mori Seiki NLX	Fanuc 31i-A
Okuma Genos	OSP-P200L
Okuma LB	OSP-P300L
Hardinge Talent 51 MSY	Fanuc 31i-A

Machine Environments: Mill-Turn

Available for purchase with a Mill-Turn license.

Machine Series	Control	Machine Series	Control
Biglia Smart Turn	Fanuc 31i-B5	Mazak Integrex i	Mazatrol Matrix 2
Okuma Macturn	OSP-E100L	Mazak Integrex IV	Mazatrol Matrix
Okuma Multus B	OSP-P200	Mazak Integrex e	Mazatrol Matrix 2
Okuma Multus BII	OSP-P300	Doosan Puma SMX	Fanuc 31i | 31i-5
Okuma Multus U	OSP-P300	Doosan Puma TT	Fanuc 31-A
Okuma LT	OSP-P300L	DMG Mori Seiki NTX	Fanuc 31i-B | 31i-B5
Okuma LU	OSP-P300L	DMG Mori Seiki NT	Fanuc 31i–A (MAPPS IV)
Mazak SQR	Smooth G		

ATTENTION! UPDATES MAY BE AVAILABLE. PLEASE REFER TO MASTERCAM.COM/SUPPORT FOR THE LATEST DOWNLOADS.

CNC Software, Inc.
671 Old Post Road
Tolland, CT 06084

www.mastercam.com

Mastercam®