



WHAT'S NEW IN MASTERCAM 2022

May 2021

Mastercam[®]

WHAT'S NEW IN MASTERCAM 2022

May 2021

© 2021 CNC Software, Inc. – All rights reserved.

Software: Mastercam 2022

Terms of Use

Use of this document is subject to the Mastercam End User License Agreement. The Mastercam End User License Agreement can be found at:

<http://www.mastercam.com/companyinfo/legal/LicenseAgreement.aspx>

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	11
Release Highlights	11
Mastercam Resources	11
Contact Us	12
Milling Enhancements	13
3D High Speed Dynamic OptiRough Available for Mill and Router Product Levels	13
Check Tool Reach Enhancements	14
Finding Unreachable Areas Due to Undercuts	14
Accessing Tools Loaded in the Part File	16
Checking the Roughing Tool Diameter and Corner Radius When Using Stock	17
Setting the Conventional Feed Rate and Steptover	19
2D Enhancements	20
Automatically Select Machining, Air, and Avoidance Regions	20
Expanded Air Region and Avoidance Region Support	21
Improvements to Chain Selection Order When Machining	23
Overriding Feed and Speed Options for Finish Passes	24
2D Contour Toolpath Enhancements	25
Reversing the Order of Spring Passes	25
Smooth Sharp Corner Machining for Control Compensation	26
Editing the Line Length and Arc Options for Contour Profile Ramp Moves	27
Using a Percentage or Radius in the 2D Face Roll In Option	28
Slot Mill Linking and Retract Motion Now Matches Contour	29
Hole Making Enhancements	30
Checking for Collisions in Hole Making Toolpaths	30
Displaying Collisions Directly From the Linking Parameters Page	33
Automatically Prevent Gouges During Linking Moves	34
Selecting Solid Holes from the Solids Manager	34

Selecting Individual Hole Segments	35
Controlling Z Motion Between Multi Passes	35
Changing Direction and Orientation During Selection	37
Using the Arc Filter/Tolerance Page for Hole Making Toolpaths	38
Linearizing the Circle Mill Toolpath	39
Changes to the Toolpath Type Page	40
Advanced Drill Enhancements	42
Populating Drill Segments From a Solid Hole	42
Using a Rapid Move to the Top of Stock Position	42
Rapidizing Down to the Initial Toolpath Position	43
Cutting, Copying, and Pasting Drill Segments	43
Helix Bore Toolpath Enhancements	44
Adding Spring Passes to Helix Bore	44
Locking Values for Helix Bore Toolpaths	45
Maintaining CS/SFM in Chamfer Drill	46
3D Enhancements	47
More Powerful and Flexible 3D Linking	47
Adding Leads to Transition Moves	48
Avoiding Tall, Long, or Steep Transition Moves	49
Trimming Passes to Fit Transition Moves	50
Additional Changes to the Linking Parameters Page	51
Improved Approximate Start Point Selection	52
Dragging and Dropping Geometry Groups	53
3D High Speed Blend Toolpath Enhancements	54
Optimizing the Cut Order	54
Machining with Spiral Cutting	55
3D High Speed Waterline Toolpath Enhancements	55
Machining with a Spiral Pattern and New Contour Directions	55
Machining Flat Areas	57
Optimizing the Cut Order	61

Filtering Out Small Toolpath Segments with Equal Scallop	61
Selecting Machining, Avoidance, and Remaining Entities from the Right-click Menu	62
Multiaxis Enhancements	64
New Multiaxis Unified Toolpath	64
Deburr Toolpath Enhancements	68
Performing Multiple Cuts	68
Limiting Tilt Motion	69
Controlling Climb and Conventional Cutting	69
Swarf Milling Toolpaths Enhancements	72
Dropping the Tool to the Floor	72
Sorting Spiral Multicuts	73
Maintaining a Single Tool Axis Orientation with Accelerated Finishing Tools	75
Rotating a Toolpath Around an Offset Point	76
Automatic Arcs and Automatic Tool Axis Orientation for Lead In/Out Moves	76
Previewing Multiaxis Toolpaths	77
Expanded Options for Excluding Undercuts for Triangular Mesh	78
Filtering the Convert to 5-Axis Toolpaths for Size and Noise	78
Rotary Advanced Toolpath Enhancements	80
Shifting a Tool Off-Center	80
Rounding Sharp Corners	81
Blending to Undercut Areas for Multiaxis Pocketing	82
Tooling Enhancements	85
Building Mill Tool Assemblies	85
Using the Mill Tool Manager in Design	86
Setting Your Active Display Tool	87
Exporting Tool Components	88
Turning Enhancements	89
Working with Mill-Turn Steady Rests	89
Creating Steady Rest Components	90
Selecting Steady Rests in Job Setup	92

- Steady Rest Support in Part Handling Strategies 93
 - The New Steady Rest Operation 93
 - The New Steady Rest Point Operation 95
- Adding Steady Rest Events to Part Handling Strategies 96
- Using Multiple Adapters in 3D-Turning Tooling 97
- Subprograms Supported in Mill-Turn 99
- Automatically Retracting to Center 100
- Generic .machine Files Installed with Mastercam 102
- Setting the Stock Diameter as an Arc or Radial Face 104
- Adding Incremental Spindle Moves 105
- Design Enhancements 106**
 - New Mesh Functions and Support for Mesh Bodies 106
 - Creating Mesh Bodies 106
 - Modifying Mesh Bodies 107
 - Refining Mesh Bodies 108
 - Simplifying Mesh Bodies 109
 - Trimming Mesh Bodies 110
 - Filling Holes and Smoothing Mesh Edges 111
 - Modifying Mesh Facets 111
 - Smoothing Mesh Areas 112
 - Exploding Meshes 114
 - Identifying Errors in a Mesh Model 115
 - Adding or Removing Colors on Mesh Facets 116
 - Performing Top-level Editing for Wireframe Entities 117
 - Editing with Point Controls 117
 - Editing with Line Controls 118
 - Editing with Spline Controls 118
 - Editing with Arc Controls 119
 - Model Prep Enhancements 120
 - Applying Push-Pull Modifications to Multiple Bodies 120

Recognizing Interrupted and Intersecting Holes	121
Improved Selection for Hole Axis	122
Surface Enhancements	123
Creating and Controlling Flowlines with Overflow UV	123
Creating Surfaces in Notes	124
Creating Draft Surfaces in Multiple Planes	125
Flipping Surface Normals Using On-screen Controls	125
Wireframe Enhancements	125
Enhancements to Support Multiaxis Machining	125
Editing Tool Axis Control Lines	126
Expanded Line Normal Functions	127
Creating Cross-Sections Along a Curve	128
Creating a Midline Between Wireframe	129
New Name and Interface for Curve Slice	130
Transform Enhancements	131
Distributing Entities Along a Chain	131
Copying Entities in Two Directions	132
Rotating Geometry Around Plane Origins	133
Dimension Enhancements	136
Create Conjugate Angle Dimensions with On-Screen Trigger Points	136
Associativity to Solid Geometry	136
Dimensioning Your Part in Multiple Planes	137
Better Cursor Control When Dimensioning Multiple Z Depths in 3D Mode	137
Using the Interactive Plane Depth Arrow	138
New Solids Manager Icons	139
Simulation Enhancements	140
Analyzing Distances Using Smart Measuring	140
Improved Toolpath Processing Time for Mastercam Simulator	141
Viewing Collisions and Alerts Using the New Report Pane	141
Setting Proximity Alerts for Verifying and Simulating	142

Simulating with a Specific External Post	143
Toolpath Utility Enhancements	144
Viewing Your Tool Assembly During Part Programming	144
Selecting Individual Solid Faces for Stock Model	148
Toolpaths Manager Enhancements	149
New Toolpaths Manager Icons	149
Exploring Toolpaths Manager Display Options	150
System Enhancements	151
Planes and Planes Manager Enhancements	151
Linking and Unlinking Planes	151
Directly Locking and Unlocking Planes	153
Toggling Display and Section for Locked Planes	154
Controlling Plane Associativity	154
Associating the Plane Origin to New Geometry	155
Simplifying the Mastercam Ribbon	156
Setting the View with the Dynamic Gnomon	156
Copying and Moving Selected Entities Between Levels	157
Providing Feedback During Your Mastercam Session	157
My Mastercam Login Enhancements	160
Automatically Change Geometry Color When Opening a File	161
Selection and Chaining Enhancements	162
Rapidly Select Solid Edges	162
Dynamically Adjust the Start and End Point of Chains	163
Setting the Color for Lathe Solid Chain Profile Preview	163
Re-selecting and Ending Chains for Lathe Operations	164
Chaining and Extruding Notes	164
File Management Enhancements	166
Working with Mastercam Content Files	166
File Translator Enhancements	167
Support for 3MF Files	167

Importing and Exporting CAD Files with New File Translators	167
Multiple STL Files Merged as Separate Level Names	167
Merge Enhancements	167
Merge Multiple Files in One Operation	167
See Level Options Immediately Upon Merging Files	168
General Enhancements	169
Borrowing Mastercam Network Licenses	169
Expanding the Functionality of Mastercam Demo/Home Learning Edition	170
Interact with Mastercam with the Toolpath Dialog Box Open	172
Accepting and Generating a Toolpath with a Single Click	173
System Configuration Enhancements	174
Customizing Mesh Color Settings	174
Showing Mesh Edges with a Greater Degree of Control	174
Moving a Shared Folder Without Copying Files	175
Setting Chaining Mode Defaults	176
Creating and Editing .NET Scripts in Mastercam	176
Finding Arcs and Splines by Length	178
Retaining Wire Work Settings	179
Posts and Machine Environments	180
Lathe Machine Environments	180
Mill-Turn Machine Environments	182
Multiaxis Posts	184
Wire Posts	185

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 Technical Previews.

Welcome to Mastercam 2022! Mastercam 2022 features new functionality focused on delivering speed and efficiency to your machining jobs.

Release Highlights

Only have a few minutes? Listed below are a few of the highlights from this release, including a new toolpath and enhanced functionality.

- ["3D High Speed Dynamic OptiRough Available for Mill and Router Product Levels" on page 13](#)
- ["New Mesh Functions and Support for Mesh Bodies" on page 106](#)
- ["Performing Top-level Editing for Wireframe Entities" on page 117](#)
- ["New Multiaxis Unified Toolpath" on page 64](#)
- ["More Powerful and Flexible 3D Linking " on page 47](#)
- ["Planes and Planes Manager Enhancements " on page 151](#)
- ["Borrowing Mastercam Network Licenses" on page 169](#)
- ["Expanding the Functionality of Mastercam Demo/Home Learning Edition" on page 170](#)

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 2022 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 over hundreds of 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 ENHANCEMENTS

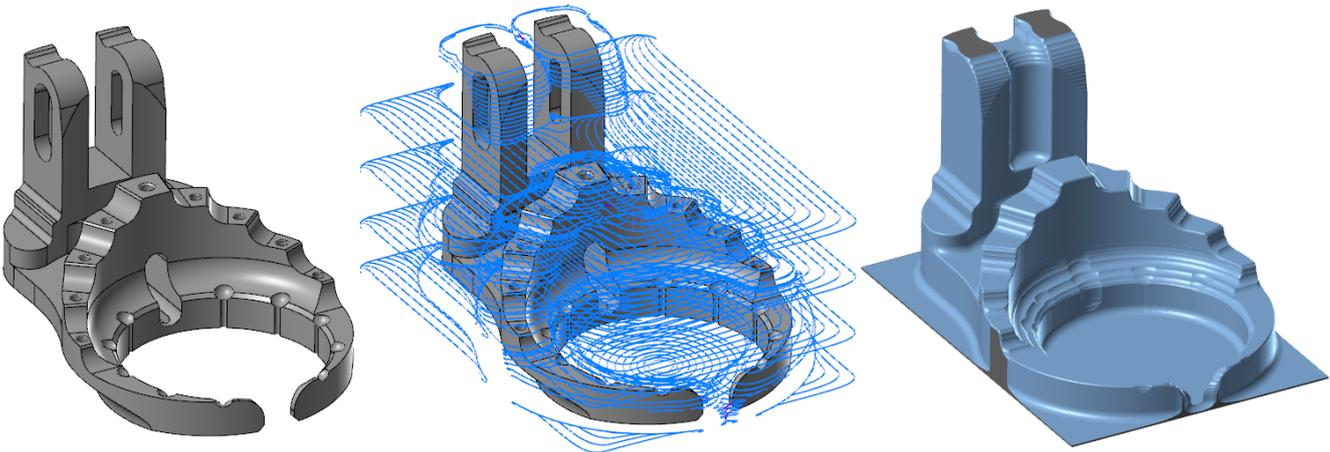
Listed below are major enhancements to the Mill product. These include improvements to 2D, 3D, and Multiaxis toolpaths.

NOTE

Unless otherwise stated, the new features and functionality listed in this section apply to both Mill and Router licenses.

3D High Speed Dynamic OptiRough Available for Mill and Router Product Levels

The 3D High Speed Dynamic OptiRough toolpath is now available for all Mill and Router product levels. Previously, Dynamic OptiRough was only available for Mill 3D. The Dynamic OptiRough toolpath machines very large cut depths with a bi-directional cutting strategy to remove the maximum amount of material with the minimum number of stepdowns.



The Dynamic OptiRough toolpath allows you to create a single toolpath to machine a part, instead of creating multiple 2D operations to achieve the same goal. This toolpath is collision aware of the part and for complex machining, the toolpath can also be aware of your tool holder by using options on the **Holder** page.

Access the Dynamic OptiRough toolpath from the **3D** gallery in the **Mill Toolpaths** contextual tab.

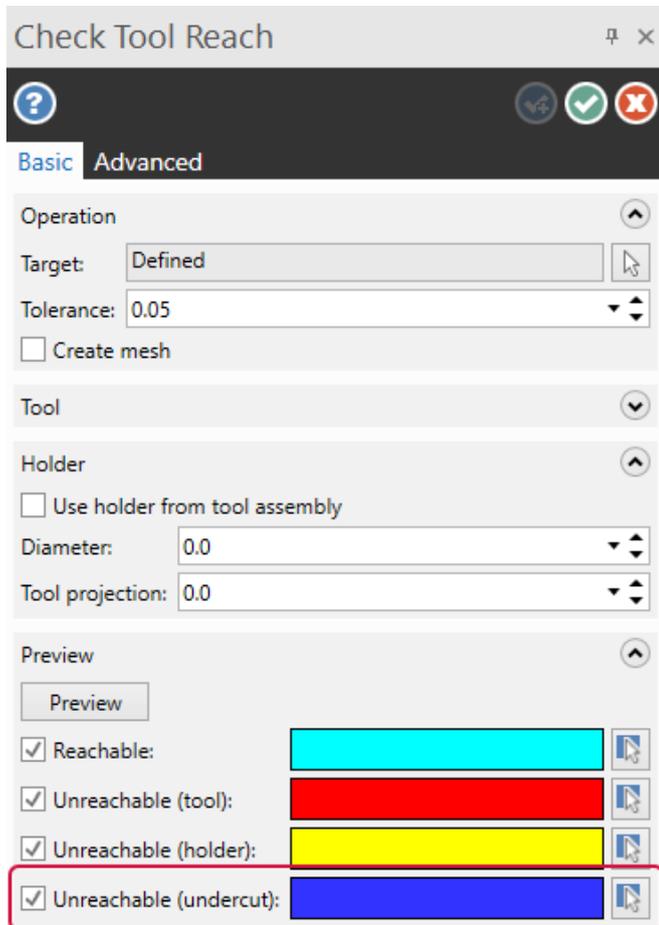


Check Tool Reach Enhancements

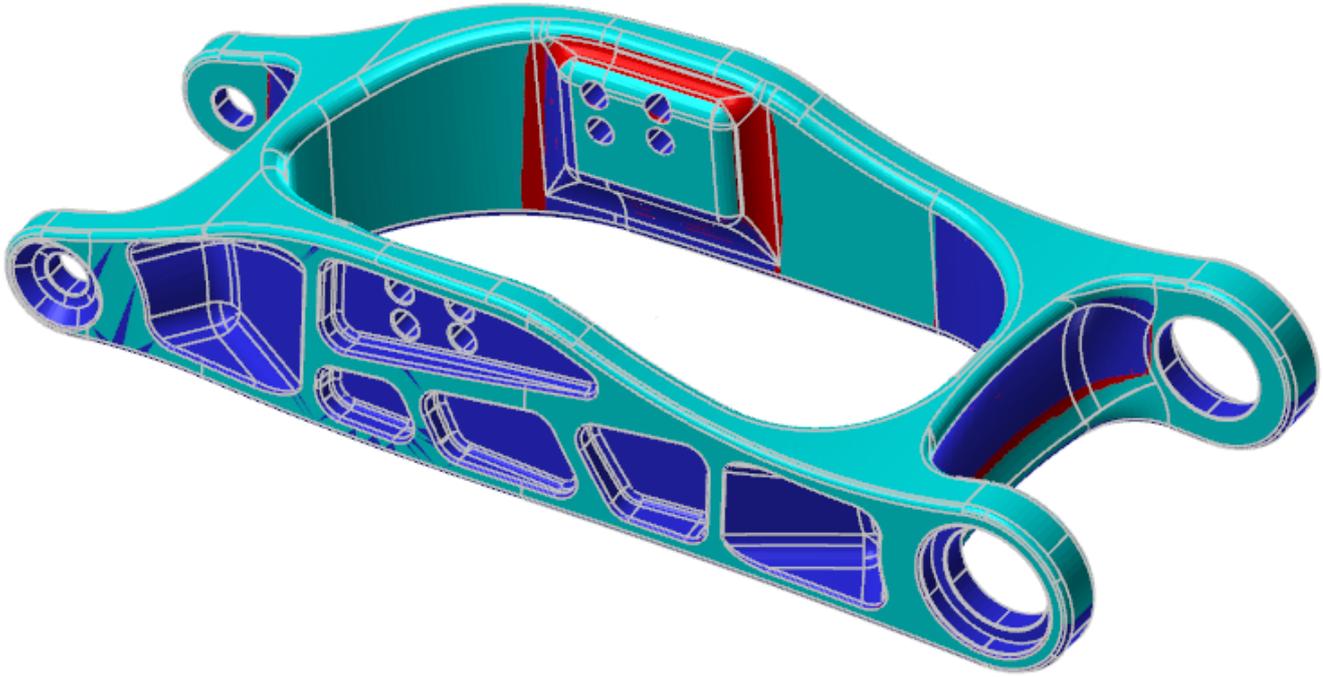
Listed below are enhancements to the **Check Tool Reach** function, located on the **Mill Toolpaths** contextual tab.

Finding Unreachable Areas Due to Undercuts

Check Tool Reach now shows the unreachable areas of a model due to undercutting. Use the **Unreachable (undercut)** option to color the model. This lets you visually detect undercuts on your models, allowing you to make more informed decisions during programming.

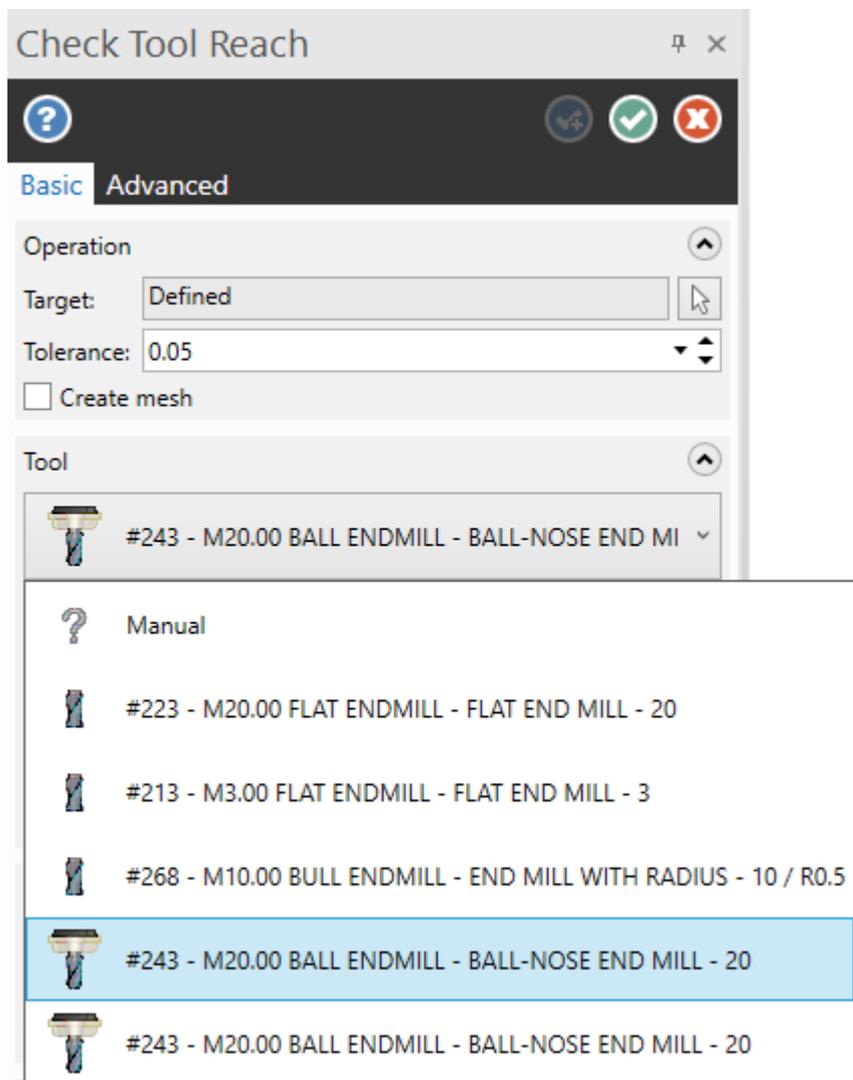


The following example shows the unreachable areas of the model due to undercutting by coloring the areas dark blue.



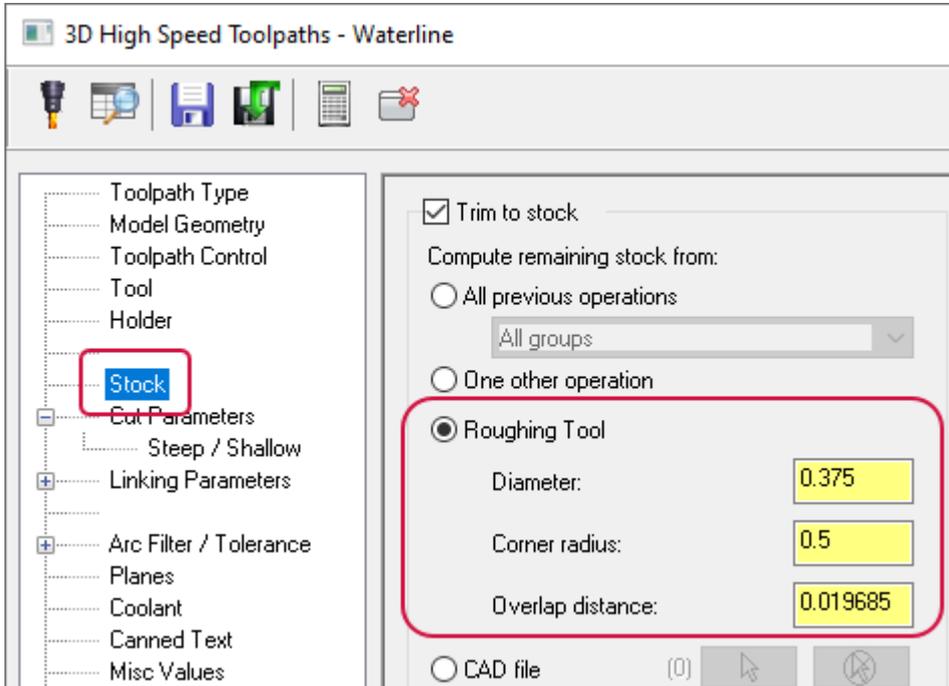
Accessing Tools Loaded in the Part File

You can now access tools loaded in the part file in the **Tool** drop-down when using **Check Tool Reach**. Mastercam automatically filters out unsupported tools from the list. Only Flat, Bull, and Ball end mill tools are supported. This allows you to check your part against tools you have already prepared, instead of modifying the options in the function panel to create your tools.

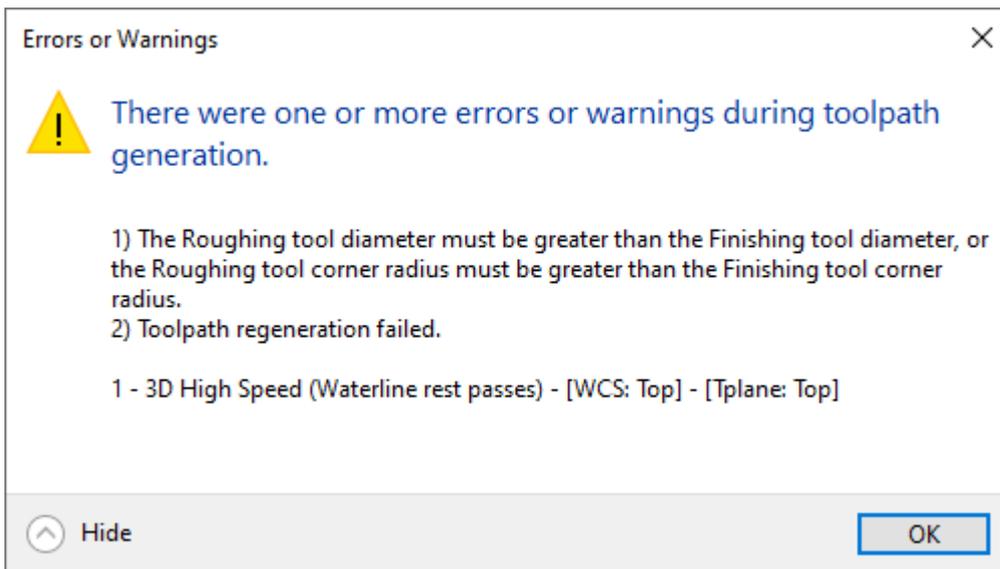


Checking the Roughing Tool Diameter and Corner Radius When Using Stock

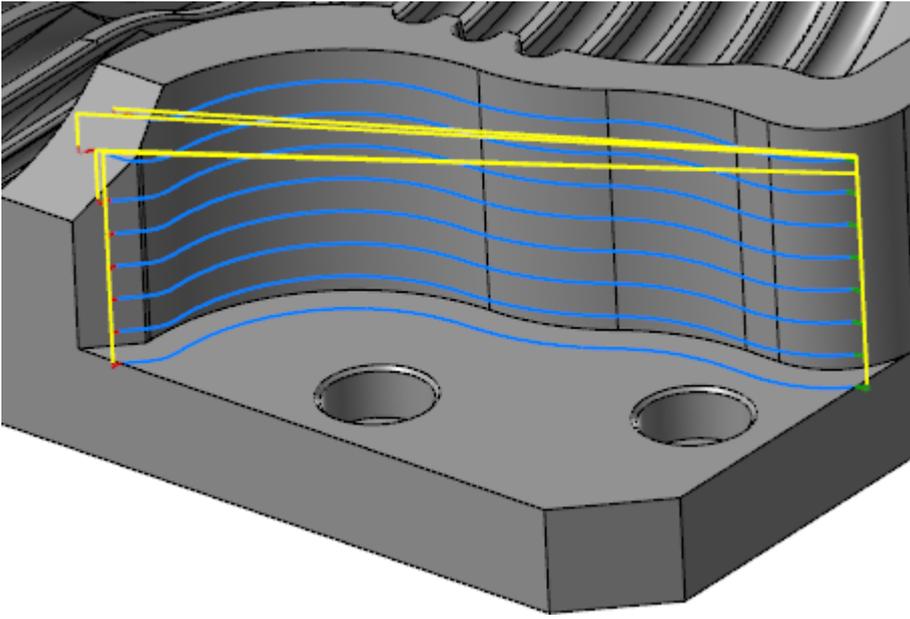
Mastercam includes an enhancement to the **Roughing tool** option on the **Stock** page for 3D high speed, 2D Area Mill, and 2D Dynamic Mill toolpaths. In previous releases, Mastercam assumed that if the set cutting tool diameter was smaller than the finishing tool diameter, then there was no material to be machined and displayed an error message.



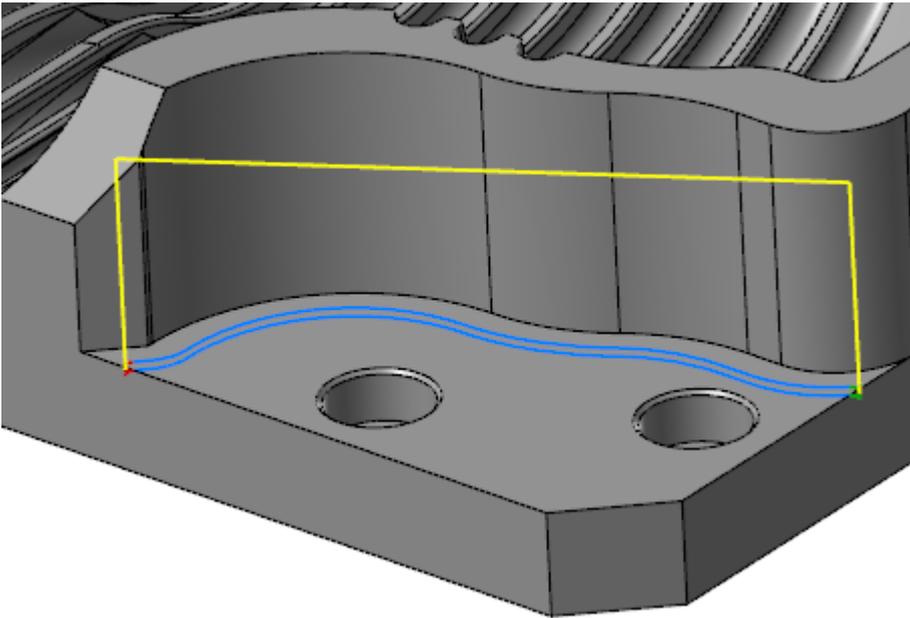
In Mastercam 2022, if the set **Diameter** is smaller than the finishing tool, Mastercam then checks the **Corner radius** parameter before setting up the finishing motion. If Mastercam detects an issue, you will receive the following warning:



For example, the part shown below is first machined with a Waterline toolpath using a bull end mill with a **Tool diameter** of 16 and a **Corner radius** of 2.



The second toolpath is machined using a flat end mill with a **Tool diameter** of 20 and a **Corner radius** of 0.

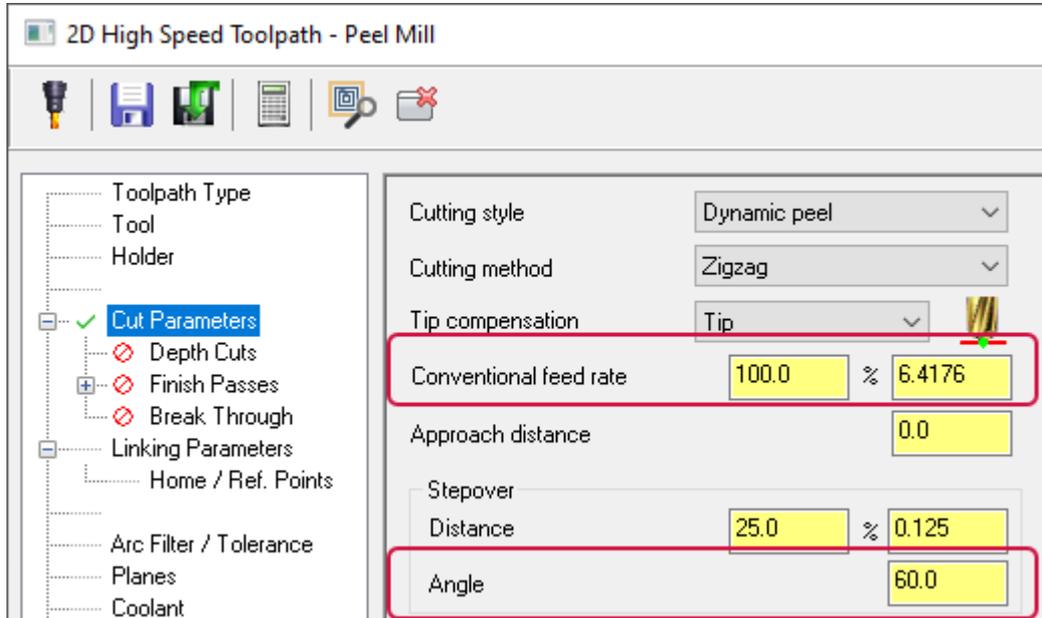


In previous releases, you would receive an error and the toolpath would not generate. Now, Mastercam generates a toolpath based on the new criteria.

Setting the Conventional Feed Rate and Stepover

You can now set the **Conventional feed rate** as a percentage for Dynamic Mill, Peel Mill (when **Cutting style** is set to **Dynamic peel**), and Dynamic OptiRough toolpaths. This parameter is found on the **Cut Parameters** page.

Conventional feed rate sets an alternate feed rate for the conventional motion of a zigzag cut. This parameter is only available when **Cutting method** is set to **Zigzag**.



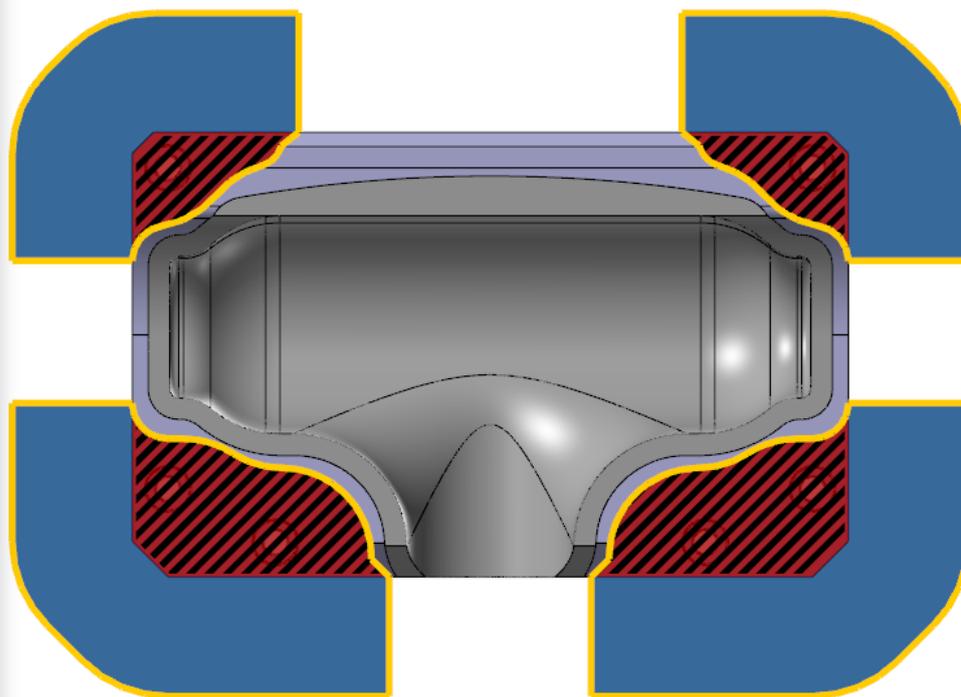
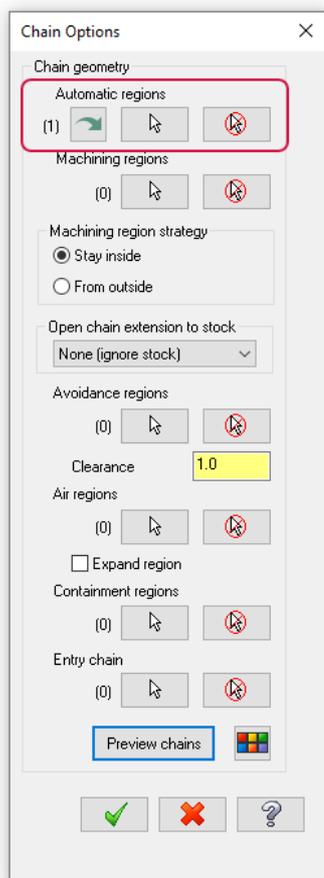
You can now also set the **Stepover** as an angle instead of a distance.

2D Enhancements

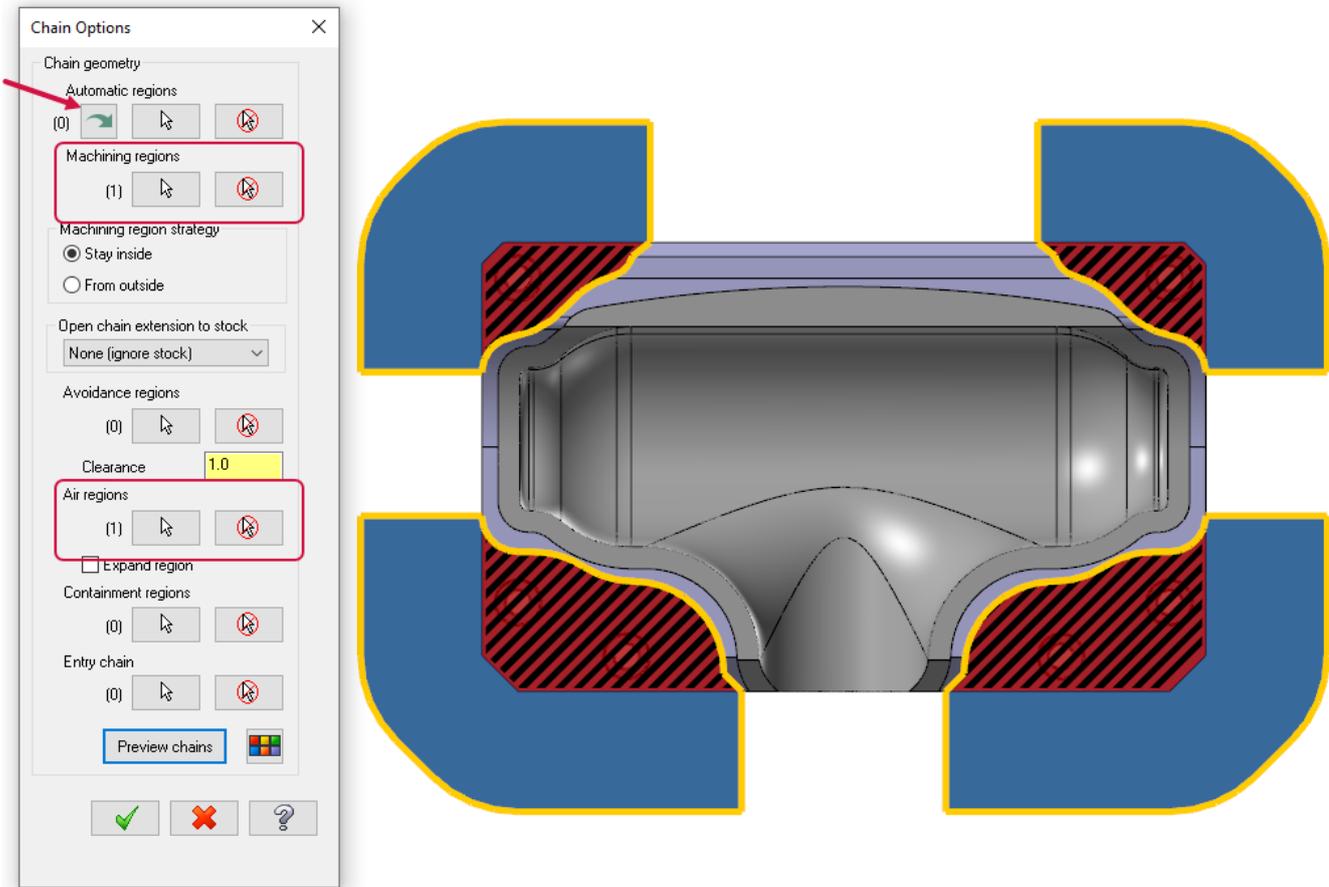
Listed below are enhancements to the 2D milling toolpaths.

Automatically Select Machining, Air, and Avoidance Regions

When selecting geometry for a 2D Dynamic Mill, Area Mill, or Dynamic Mill toolpath, you can use the new **Automatic regions** selection to automatically create **Machining**, **Air**, or **Avoidance regions** based on the selected solid geometry. After you make your selection, Mastercam automatically adds in **Air** or **Avoidance regions** in addition to the selected **Machining regions**.

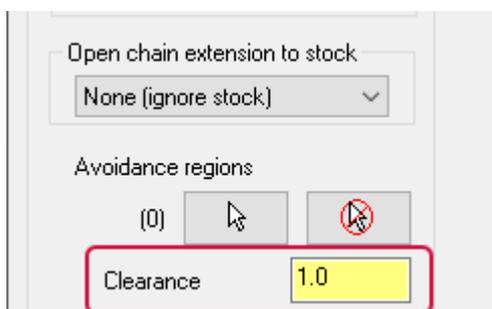


You can also convert the selected **Automatic regions** into **Machining**, **Air**, or **Avoidance regions** by using the **Convert automatic regions** button in the **Chain Options** dialog box, the **Toolpath Type** page, or from the **Chain Manager** right-click menu.

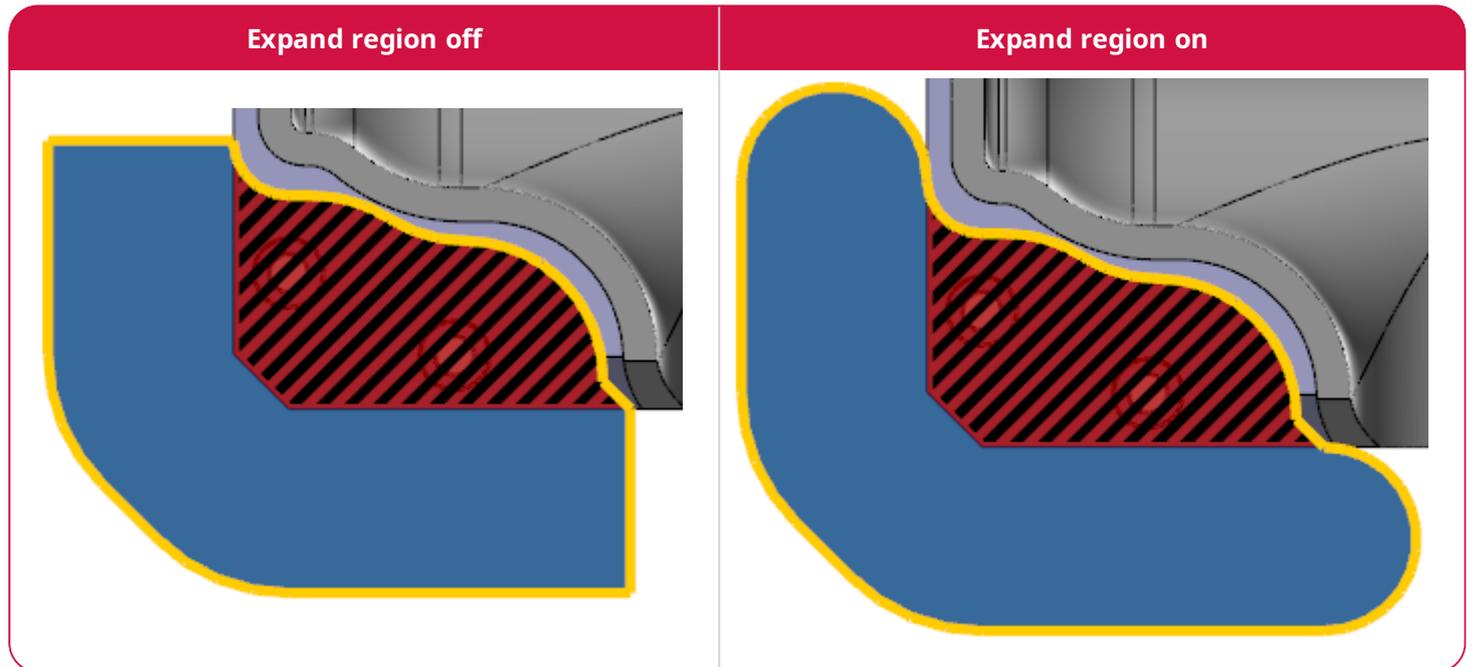
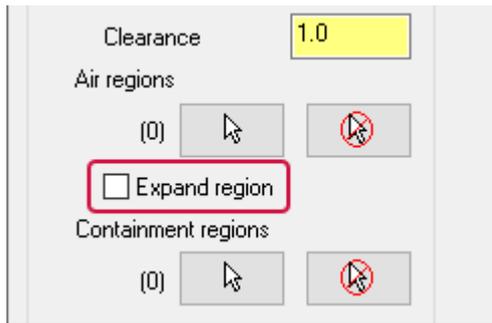


Expanded Air Region and Avoidance Region Support

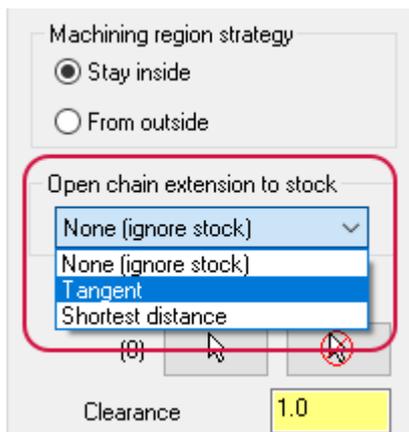
The **Chain Options** dialog box has improved. The **Avoidance regions** option now includes a **Clearance** value. Use this value to set the distance the tool will stay away from the selected **Avoidance regions**. Additionally, the entered amount is added to or subtracted from the **Stock to leave on walls** and **Stock to leave on floors** values found on the **Cut Parameters** page.



The new **Expand region** option expands the selected air regions so that the tool can move while trying to reach the material. This improves the amount of reachable material on certain open pocket shapes.



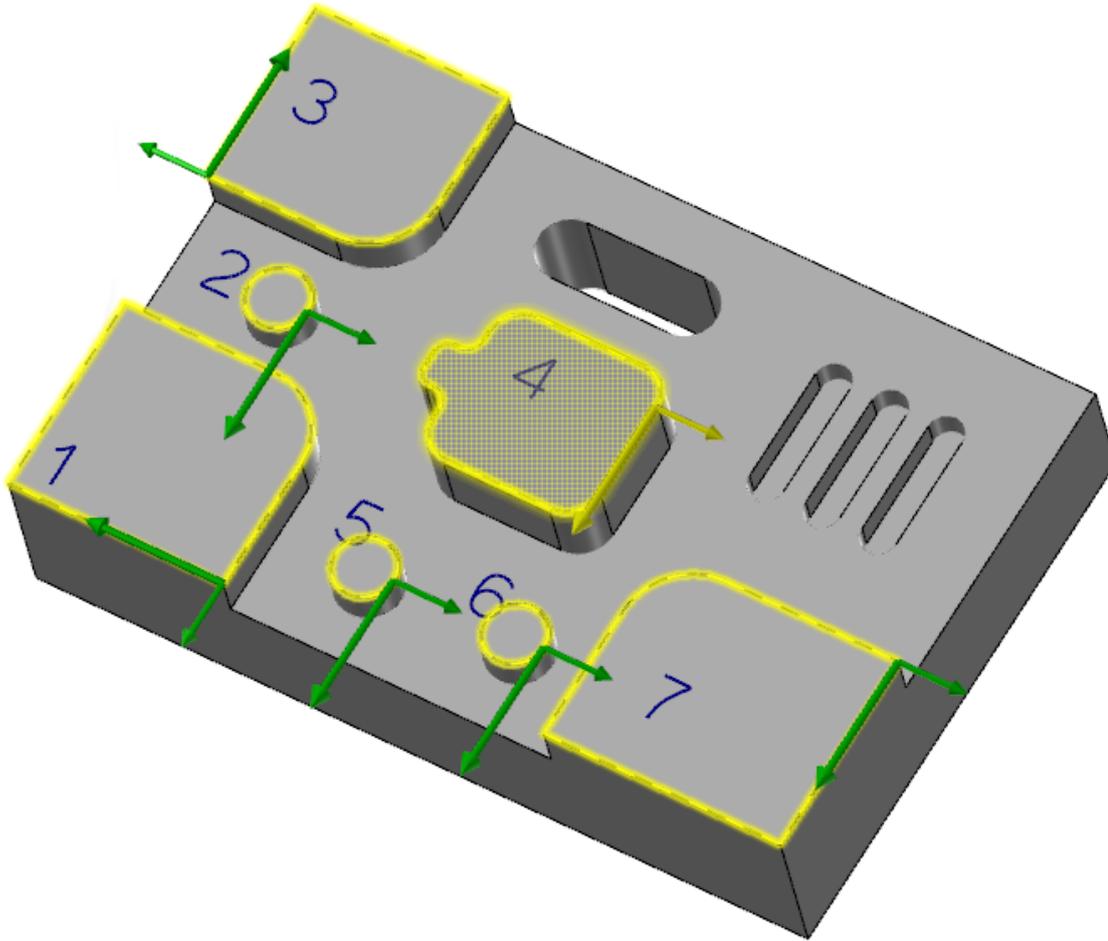
Additionally, the **Open chain extension to stock** options are now a drop-down instead of radio buttons.



Improvements to Chain Selection Order When Machining

Dynamic Mill, Area Mill, and Dynamic Contour toolpaths include improvements to the selection order of chains. Mastercam now respects the chain order shown in the Chain Manager and the selected chain's **Z depth**. Additionally, you can benefit from the available sorting options. The toolpath starts machining the chains located at the highest **Z depth** and then proceeds to chains located at the next highest **Z depth**, and so on.

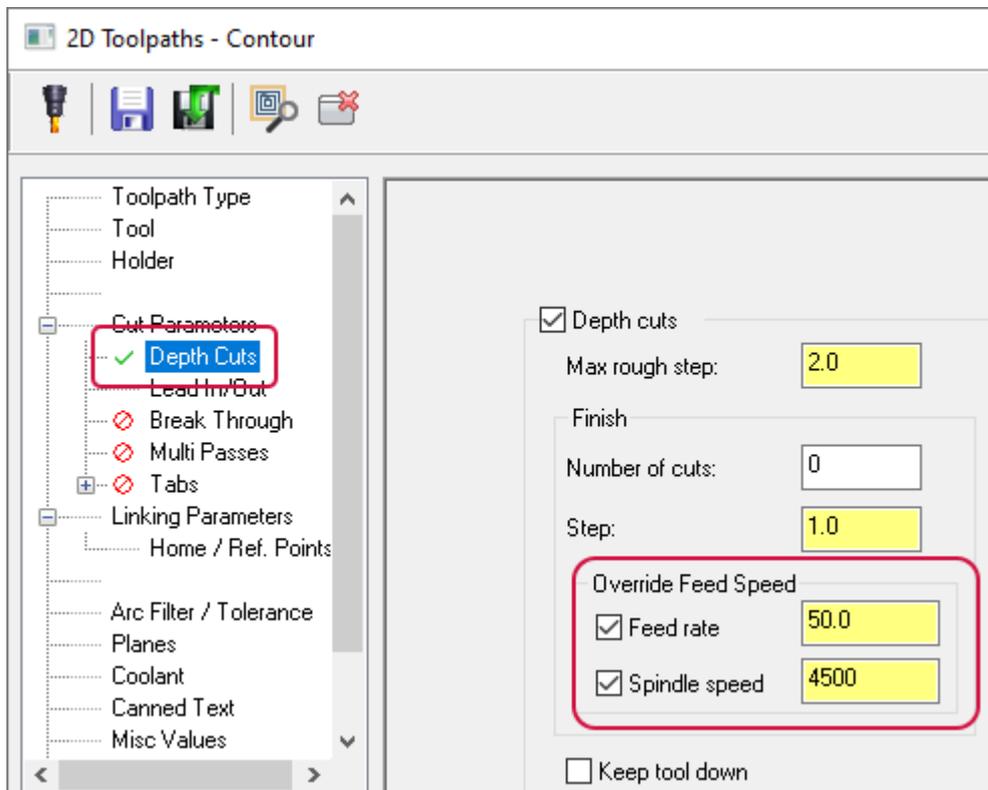
For example, in the image below, chains 2, 5, and 6 are at a **Z depth** of **-0.5**. Chains 1, 3, 4, and 7 are at a **Z depth** of **0**. When machining, the chain order is 1, 3, 4, 7, 2, 5, and 6.



If all the selected chains are at a **Z depth** of **0**, the machining order is 1, 2, 3, 4, 5, 6, and 7.

Overriding Feed and Speed Options for Finish Passes

All tree-style 2D toolpaths now include an option to override feed and speed values for finish passes. These options are available on the **Depth Cuts** page. This gives you a better control for floor finishing and allows you to create one toolpath instead of creating several to machine the final pass.



You can individually set overrides for feed rate or spindle speed, or set overrides for both.

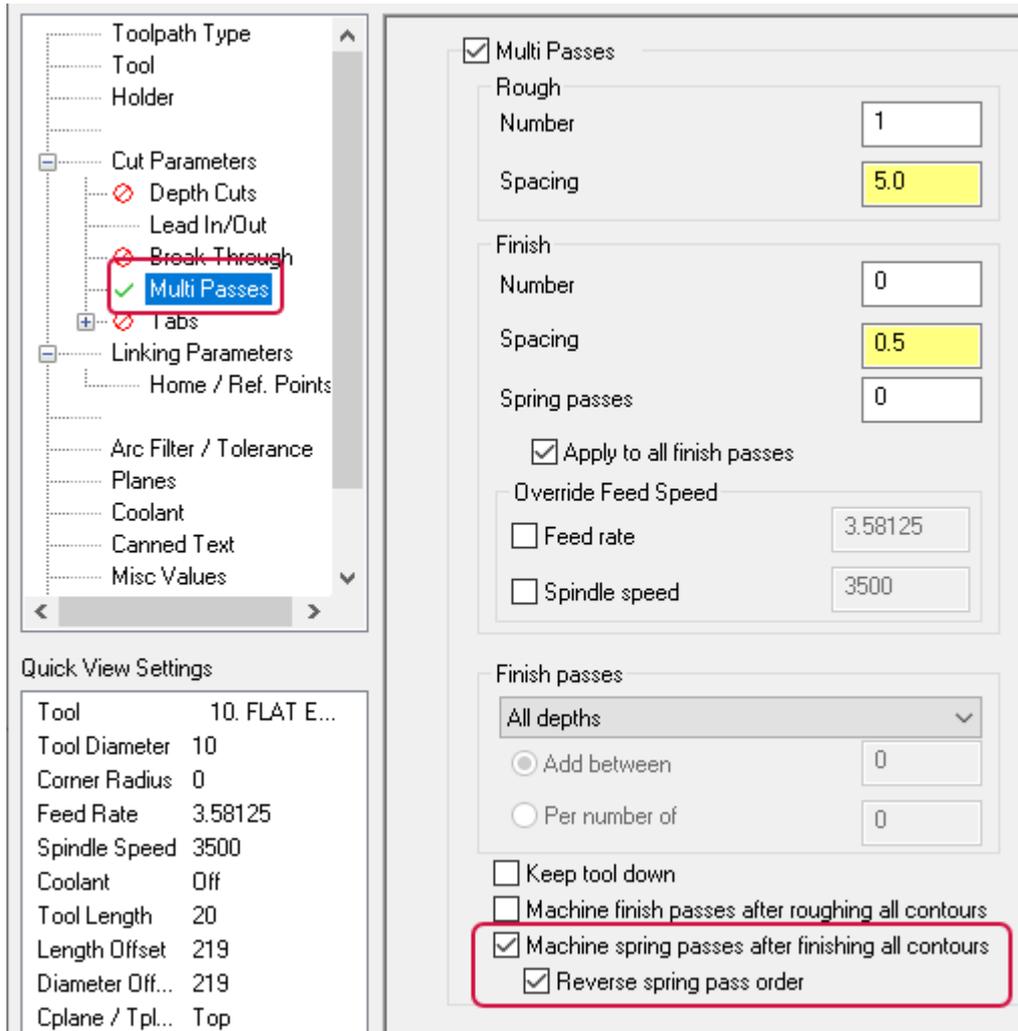
- **Feed rate:** Sets a different feed rate for the finish passes than the feed rate used for roughing.
- **Spindle speed:** Sets a different spindle speed for the finish passes than the spindle speed used for roughing.

2D Contour Toolpath Enhancements

Listed below are enhancements to the 2D Contour toolpath.

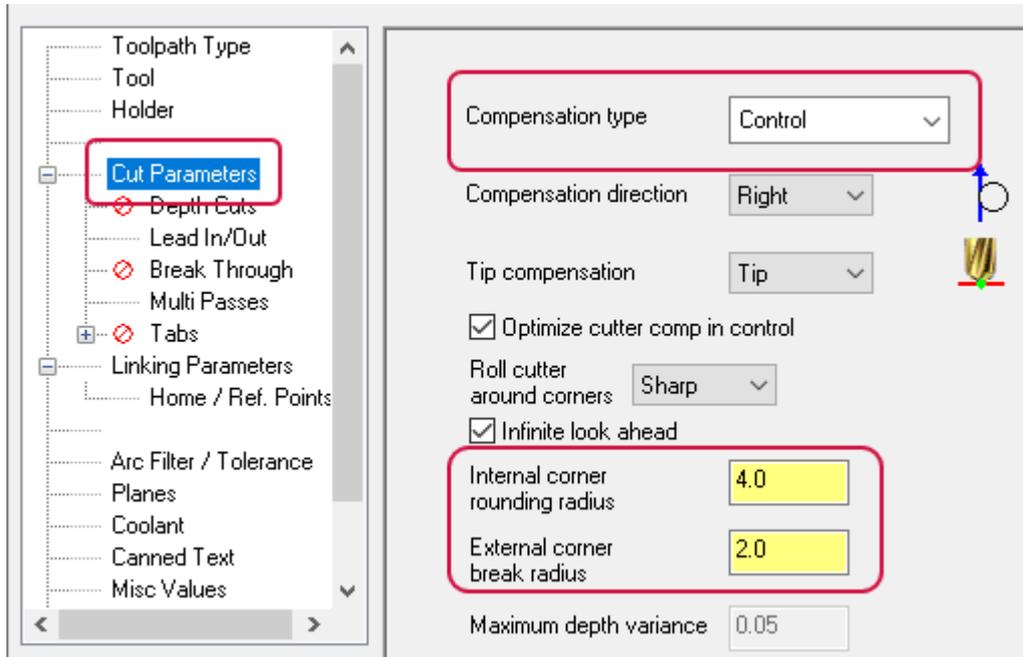
Reversing the Order of Spring Passes

The cut order of spring passes can now be reversed. Spring passes are additional finish passes that run along the same path as the last finish pass. Use the **Reverse spring pass order** option on the **Multi Passes** page to reverse the cut order of spring passes. This option is only enabled when **Machine spring passes after finishing all contours** is selected.



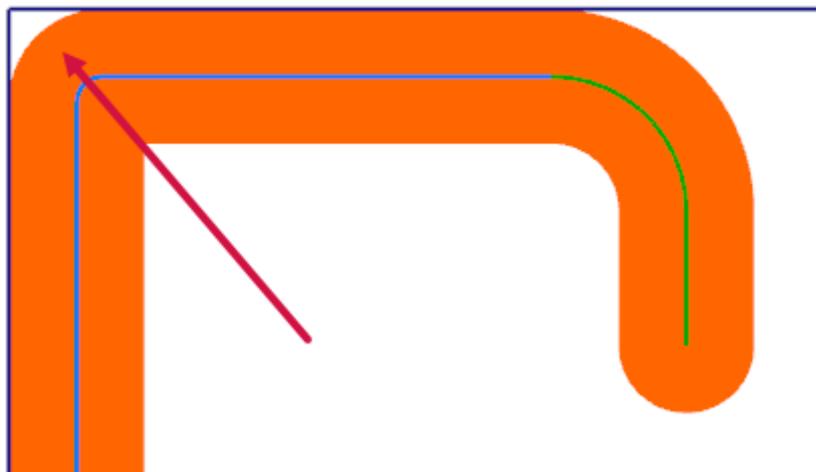
Smooth Sharp Corner Machining for Control Compensation

You can now set the **Internal corner rounding radius** and **External corner break radius** when **Compensation type** is set to **Control** on the **Cut Parameters** page.



These options do the following:

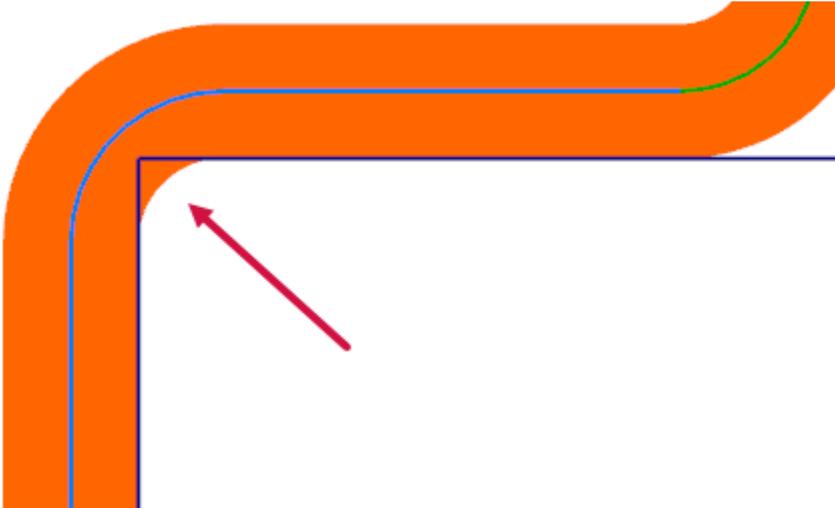
- **Internal corner rounding radius:** Creates smoother tool motion in sharp corners. Corner smoothing reduces tool wear and makes your tool motion more efficient.



NOTE

The value of the **Internal corner rounding radius** must be greater than the **Tool radius**. If not, Mastercam ignores this setting.

- **External corner break radius:** Breaks external sharp corners within a chain to create a smooth, rounded corner.

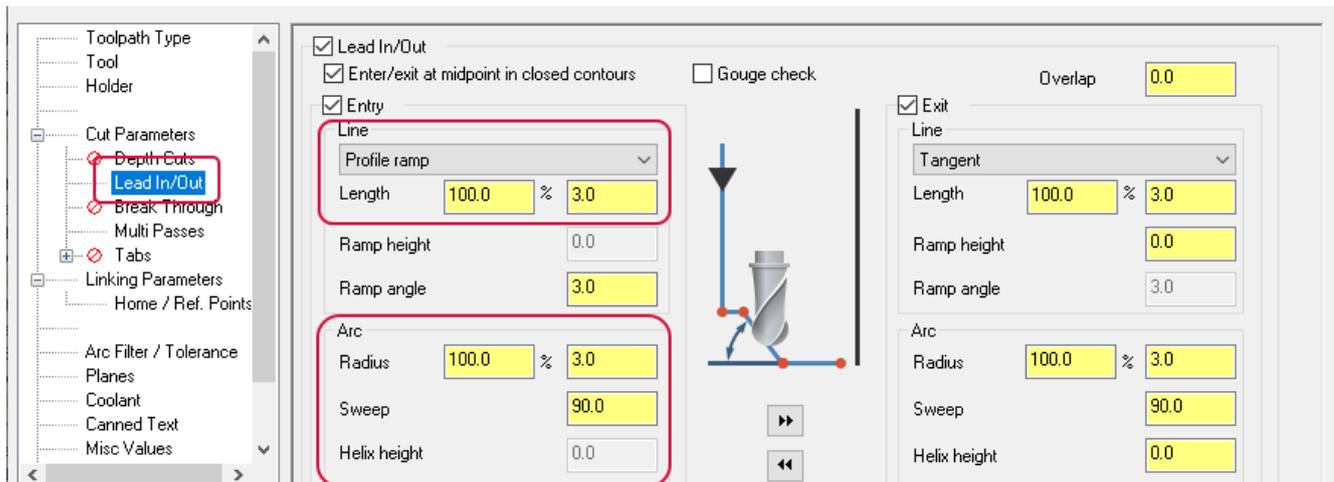


WARNING

This option will violate the selected chains and cut into the geometry.

Editing the Line Length and Arc Options for Contour Profile Ramp Moves

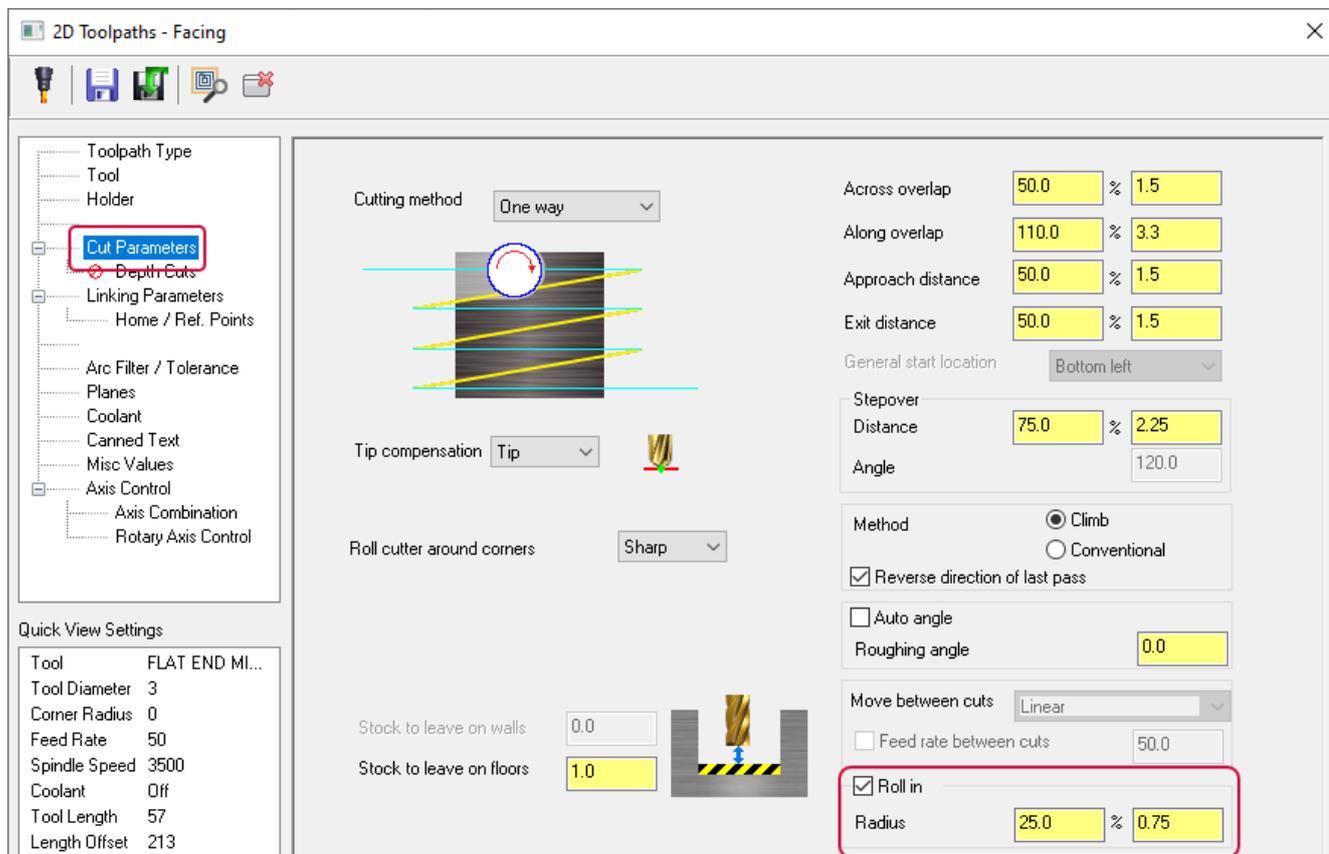
You can now edit the **Length** option and **Arc** options on the **Lead In/Out** page when **Line** is set to **Profile ramp**.



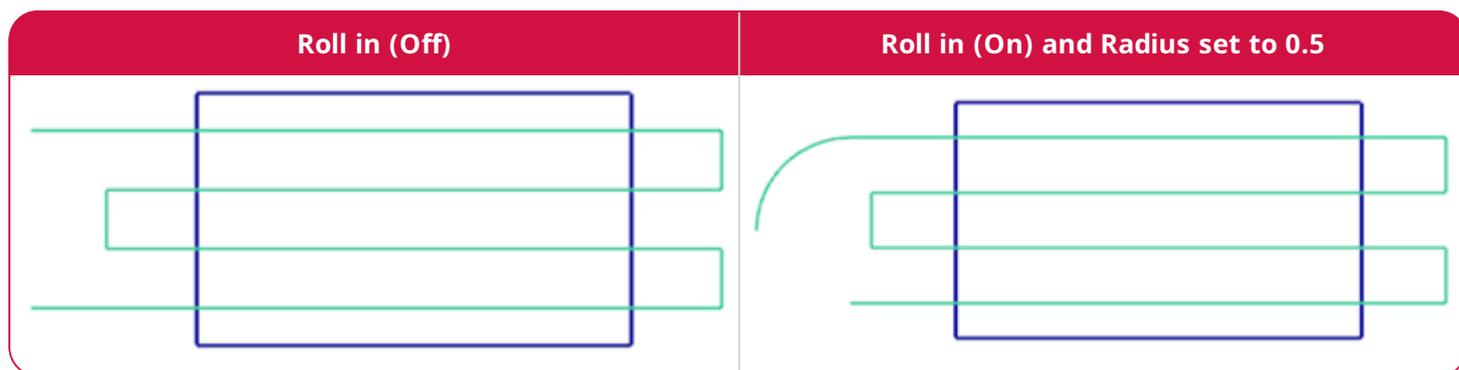
Profile ramp creates an entry or exit move that follows a ramp motion. Instead of a straight line, the tool will track around the contour of the profile as it enters or exits for a smoother transition in or out of the cut. **Length** sets the line length for entry or exit lines. The **Arc** options define the entry and exit arcs, by setting its sweep angle, radius, and helix height.

Using a Percentage or Radius in the 2D Face Roll In Option

The **Roll in** parameter, found on the **Cut Parameters** page for Face toolpaths, now includes a **Tool percentage** option in addition to the **Radius** option. This change is consistent with the other available parameters on this page.



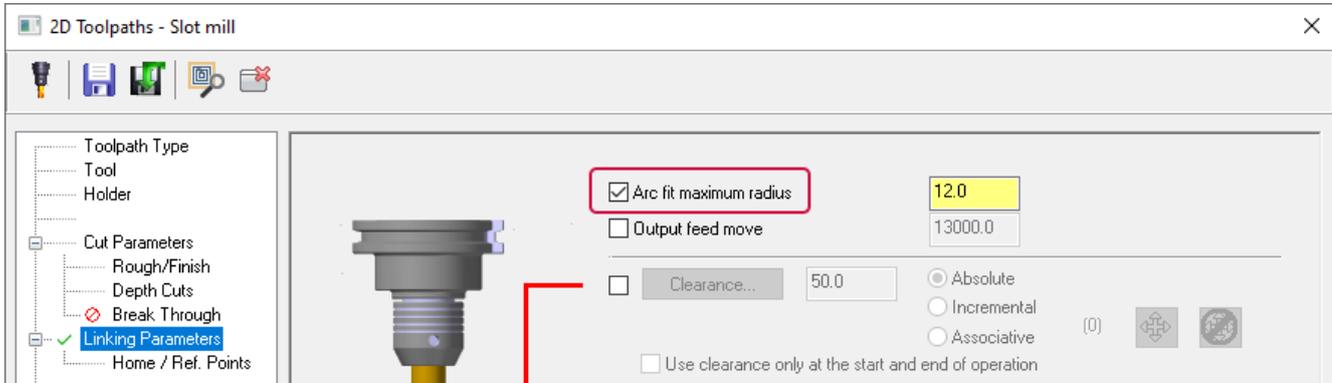
The **Roll in** option adds an arc move to the lead in move. The direction of this move is always opposite from the stepover direction. This option improves tool engagement, resulting in less tool stress.



When using the **Roll in** option, Mastercam ignores the **Along overlap** value.

Slot Mill Linking and Retract Motion Now Matches Contour

The Slot Mill toolpath's retract and linking motion now mimics the retract and linking motion of a 2D Contour toolpath. This change better supports the **Arc fit maximum radius** option on the **Linking Parameters** page, which attempts to fit a 90-degree arc move into rapid and clearance moves.

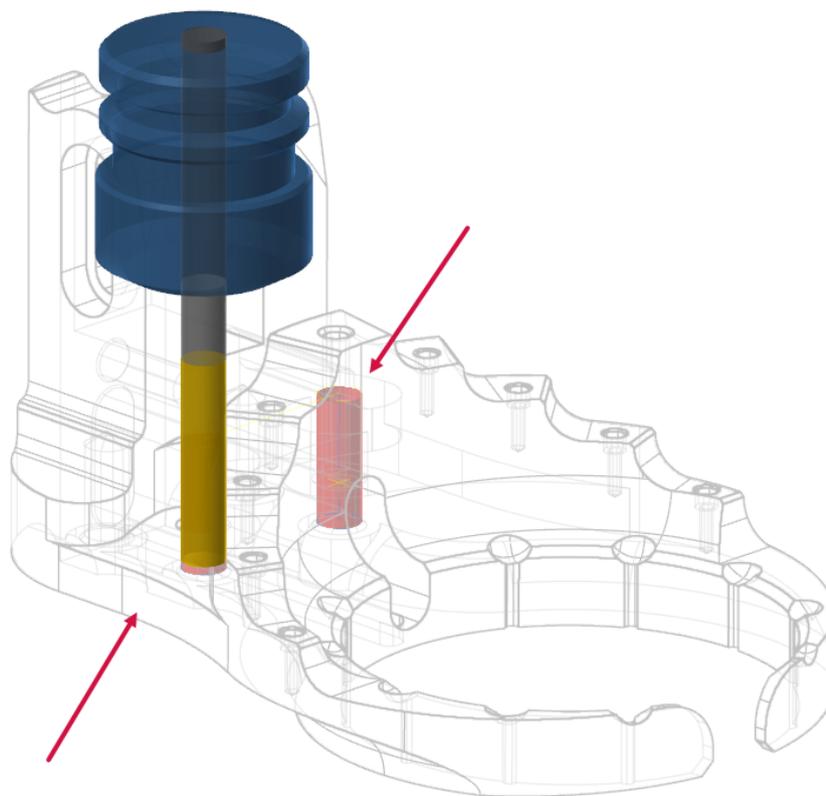
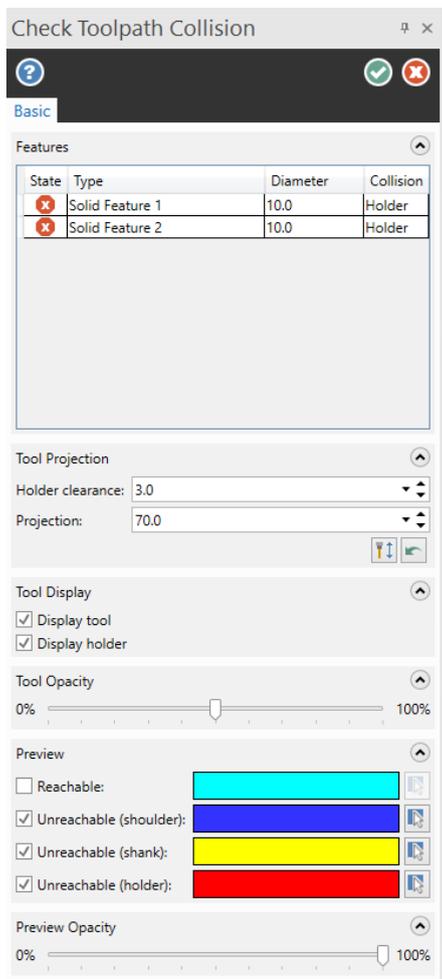


Hole Making Enhancements

Listed below are enhancements to the Hole Making toolpaths.

Checking for Collisions in Hole Making Toolpaths

The new **Check Collision** function allows you to determine if you can machine the part with the defined holder and tool combination. If the tool is long enough, **Check Collision** can also automatically adjust the height to be clear of any collisions.

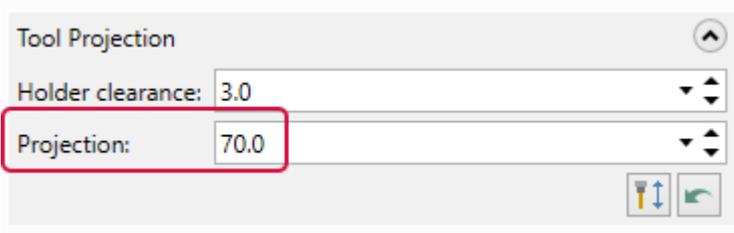


Mastercam displays one of three conditions for each hole in the **Features** list:

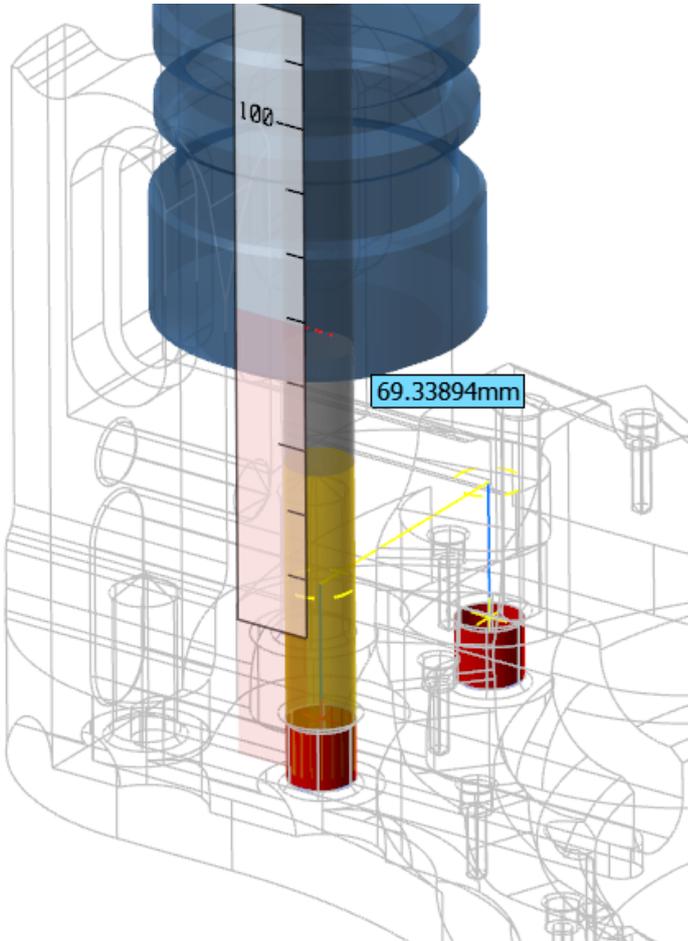
Icon	Description
	The green check indicates no problems.
	The yellow exclamation indicates a problem, such as the hole depth being greater than the flute length.
	The red X indicates a gouge.

If there is a collision or warning, the collision check tells you what part of the tool is causing the problem: shank, shoulder, or holder. If the holder is gouging, you can do one of the following:

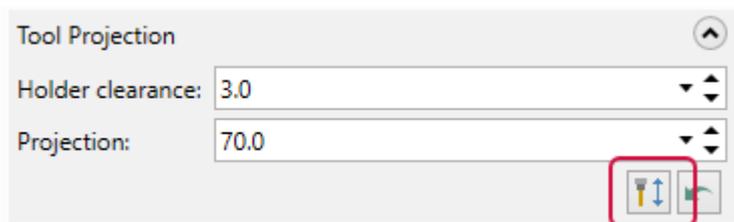
- Adjust the projection manually by typing in a new number.



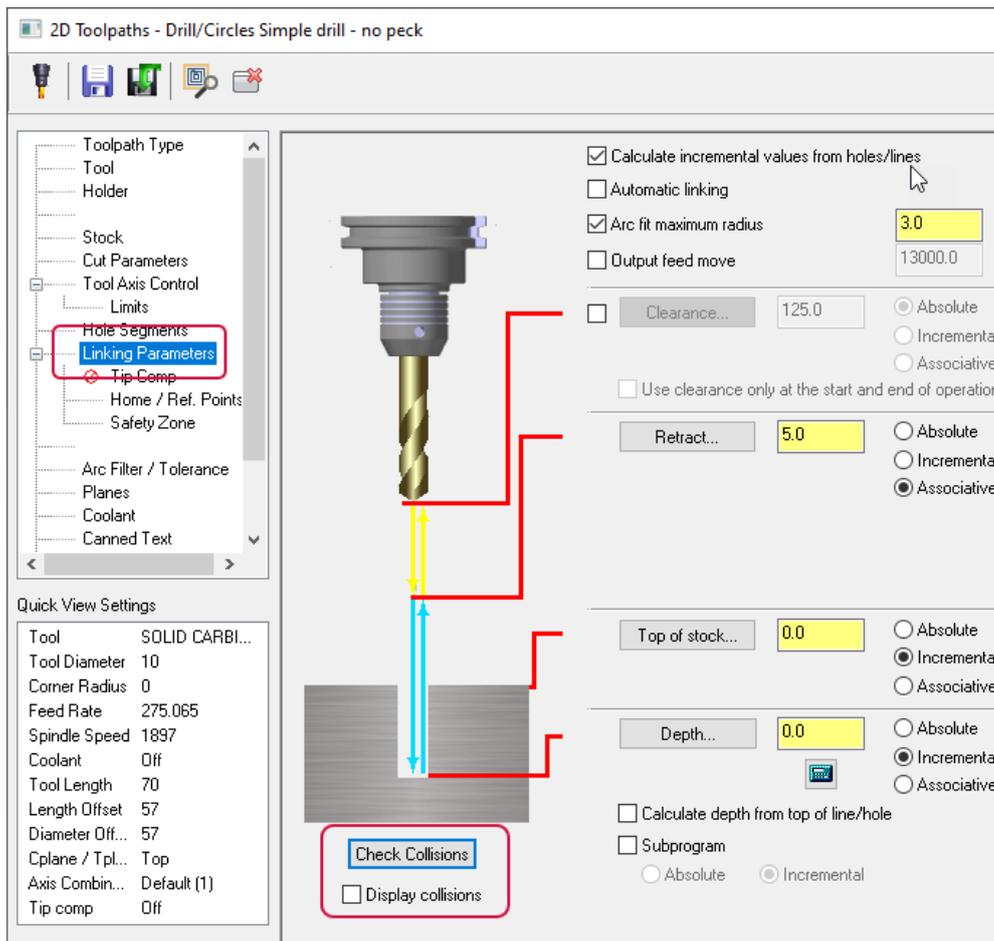
- Click-and-drag the holder on the graphics screen.



- Select the **Optimize projection** button to get the minimum extension needed.

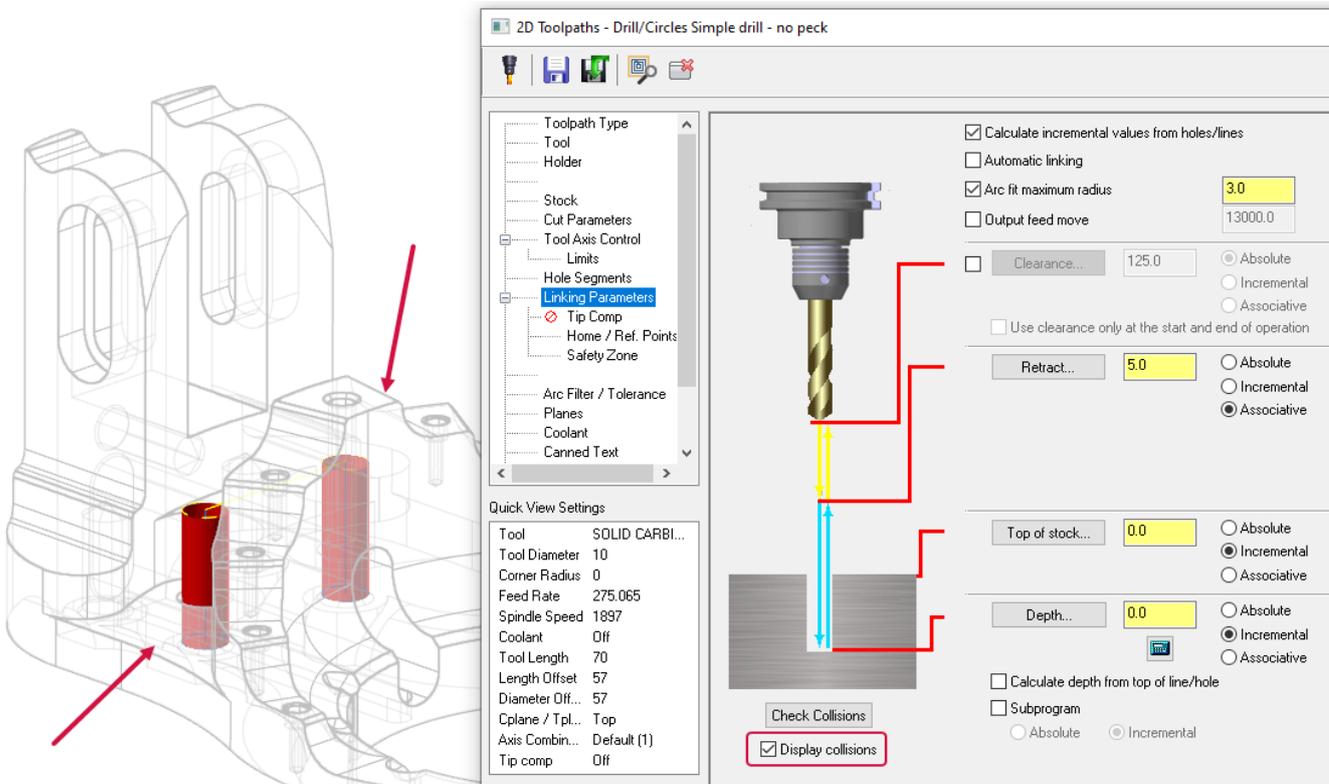


The **Preview Opacity** slider controls the translucency of the hole walls in the graphics window, allowing you to view any issues with your part. Access this function by clicking the **Check Collisions** button in the **Linking Parameters** page of hole making toolpaths.

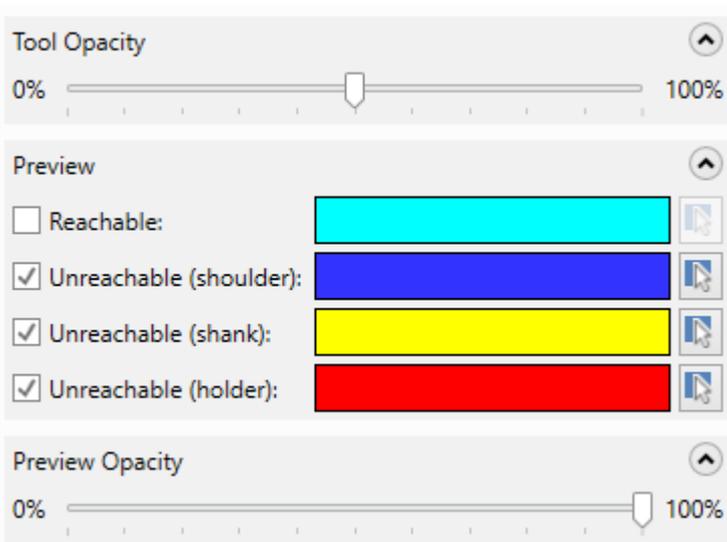


Displaying Collisions Directly From the Linking Parameters Page

To quickly see the current status of your linking parameters, use the **Display collisions** option to show collisions using the current settings. By selecting this option, you can continue to make modifications to the parameters on this page while simultaneously viewing any collision issues in the graphics window.

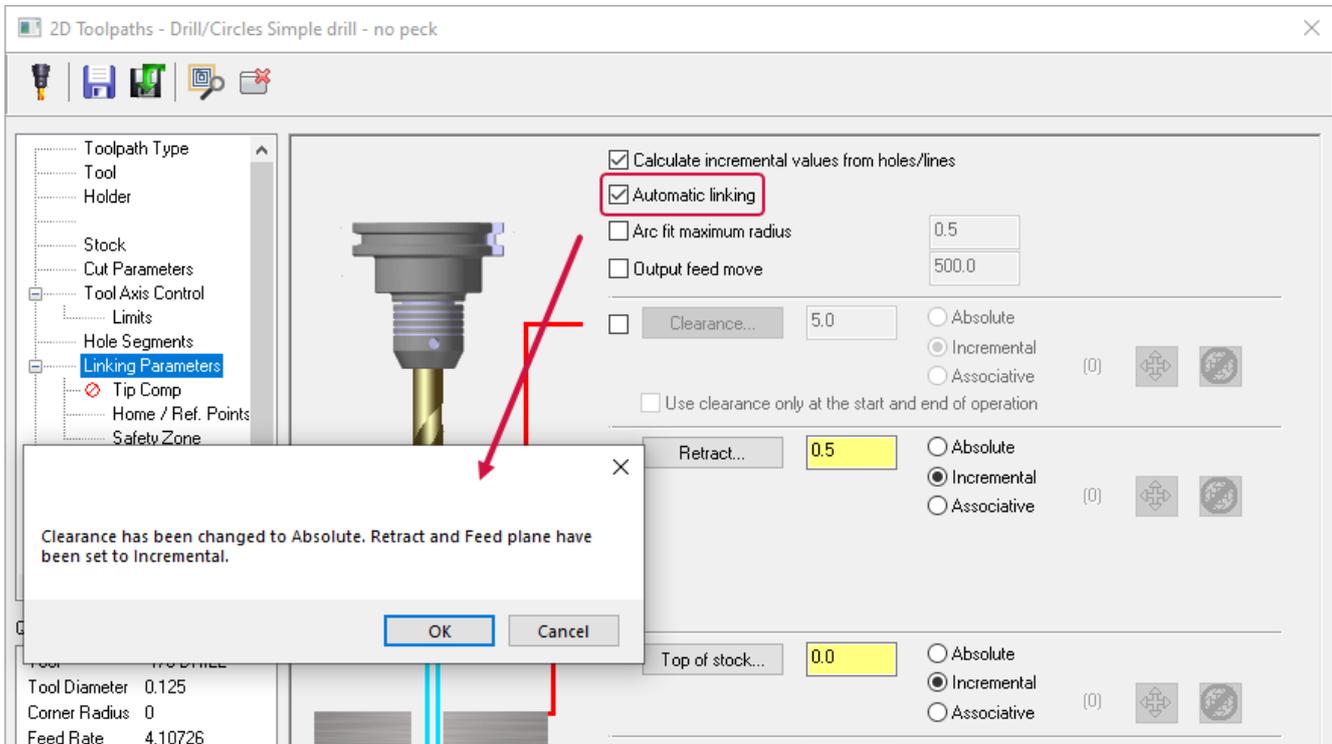


In the graphics window, Mastercam displays a cylinder that represents the entire stroke for each hole. If you selected a wireframe point or line where the diameter is not available, Mastercam uses the diameter of the tool for the hole width. The opacity of the cylinders and the display of the bands/areas of collisions is controlled by the **Check Toolpath Collisions** function panel options.



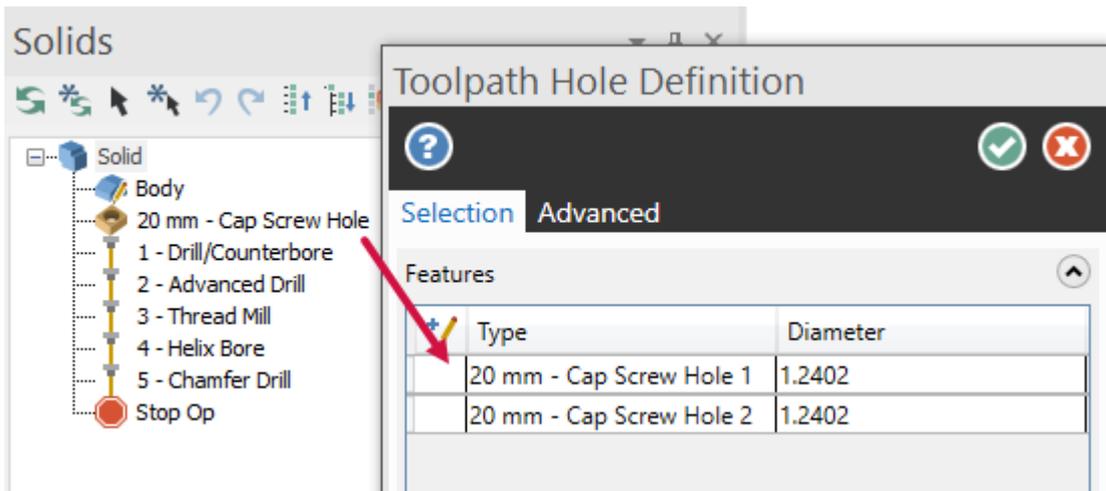
Automatically Prevent Gouges During Linking Moves

A new option, **Automatic linking**, is now available on the **Linking Parameters** page for hole making toolpaths. When selected, Mastercam modifies your **Clearance**, **Retract**, **Top of stock**, and **Depth** parameters to avoid collisions with the **Solid model** or the selected **Avoidance geometry**.



Selecting Solid Holes from the Solids Manager

When creating a hole making toolpath, you can now select a solid hole operation from the Solids Manager to populate the **Feature list** in the **Toolpath Hole Definition** function panel, or use standard Mastercam selection controls to select the solid from the graphics window. You can also remove individual positions from those holes to be machined.



Selecting Individual Hole Segments

You can now select different hole segments when creating a hole making toolpath using the **Hole Segments** page. This page is active when you select a Solid Hole operation as the machining geometry.

The screenshot shows the '2D Toolpaths - Advanced Drill' window. On the left, a tree view lists various toolpath settings, with 'Hole Segments' highlighted in a red box. Below the tree is a 'Quick View Settings' panel showing parameters like Tool (1/8 DRILL), Tool Diameter (0.125), Corner Radius (0), and Feed Rate (4.10726). The main area is divided into two sections: 'Active Segments' and a 3D model.

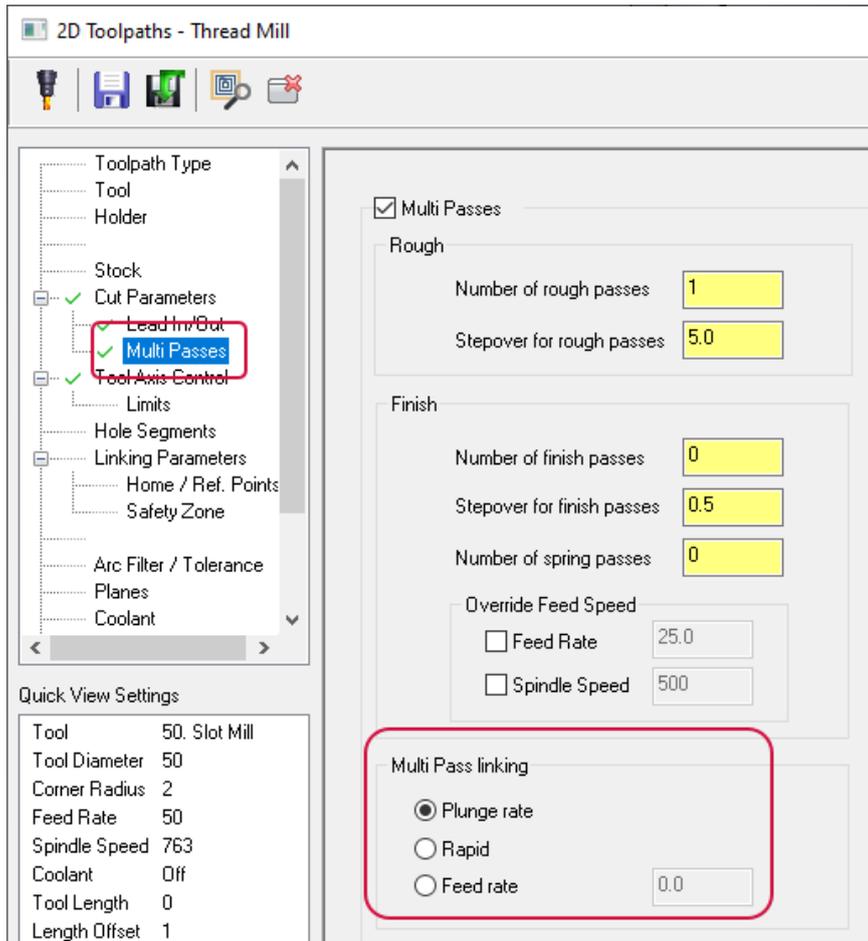
Segment	Major Ø	Taper	Length
Counterbore	1.24016	0.0	0.7874
Hole Diameter	0.81693	0.0	1.7126
Tool Tip	0.81693	0.0	0.24543

The 3D model shows a hole with a major diameter of 1.240, a hole diameter of 0.817, and a total length of 2.745. The hole diameter segment is highlighted in red, and the counterbore segment is highlighted in blue.

After you select a hole operation, Mastercam populates the **Hole Segments** page. From this page, you choose the segments that drive the linking parameters. Additionally, this new page directs Mastercam to import toolpaths that already identify which part of the solid hole to reference. You can then automate creating multiple toolpaths that are needed to machine complex holes to the correct depths.

Controlling Z Motion Between Multi Passes

New Multi pass linking options increase your control over the motion between multi passes on a Thread Mill toolpath.

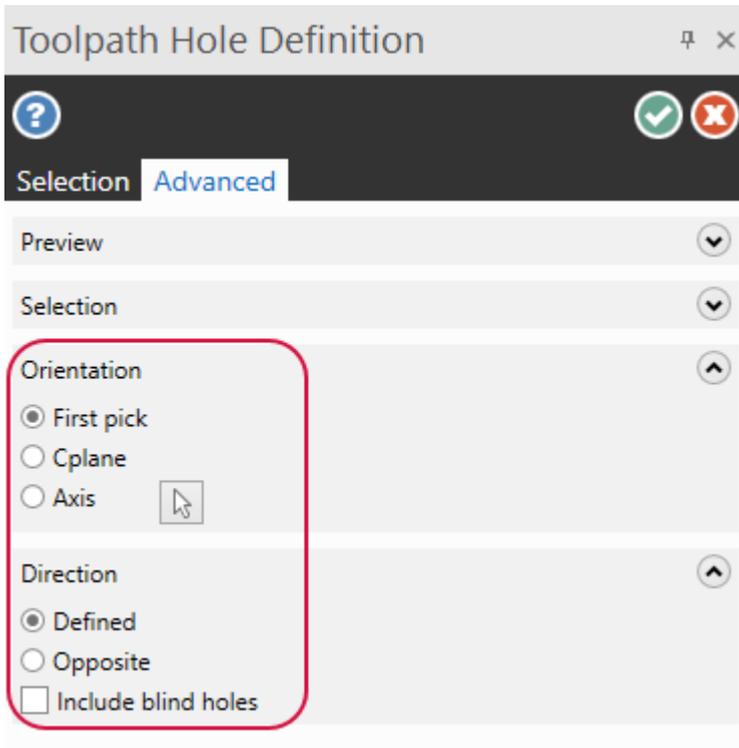


Use the new **Multi pass linking** options on the **Multi Passes** page to do the following:

- Set the feed rate to equal plunge rate.
- Set the feed rate as rapid motion.
- Set a new feed rate specifically for linking between multi passes.

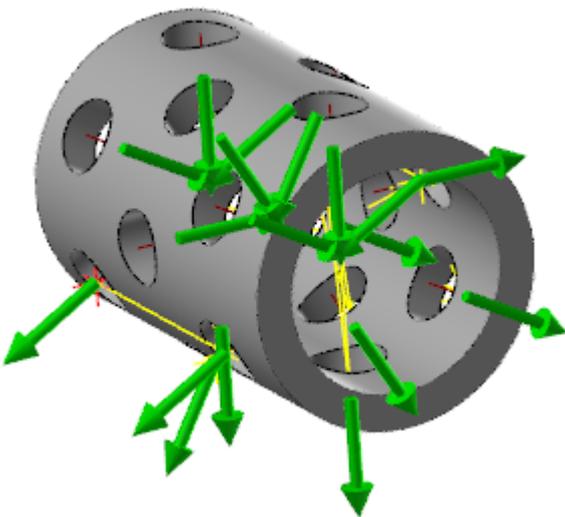
Changing Direction and Orientation During Selection

The new **Orientation** and **Direction** options on the **Advanced** tab of the **Toolpath Hole Definition** function panel allow you to change the hole direction and orientation, or to align the hole to an axis while making selections for a hole making toolpath.

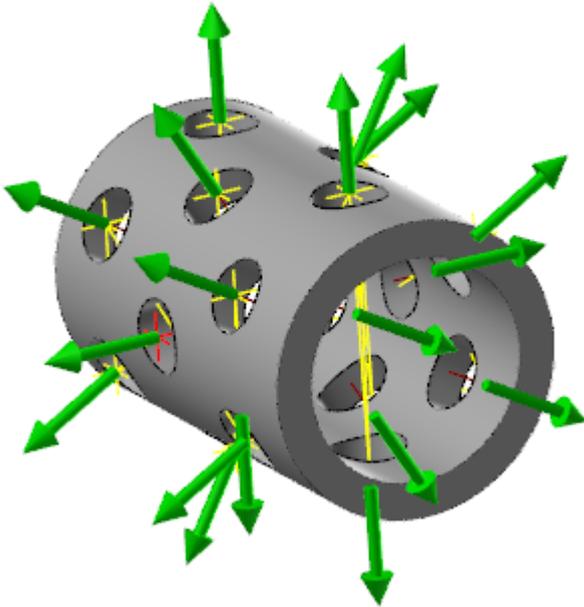


This is particularly helpful when working on round parts, as it makes manual selection easier.

The first image below shows the axis alignment for **First pick** when all holes are selected.

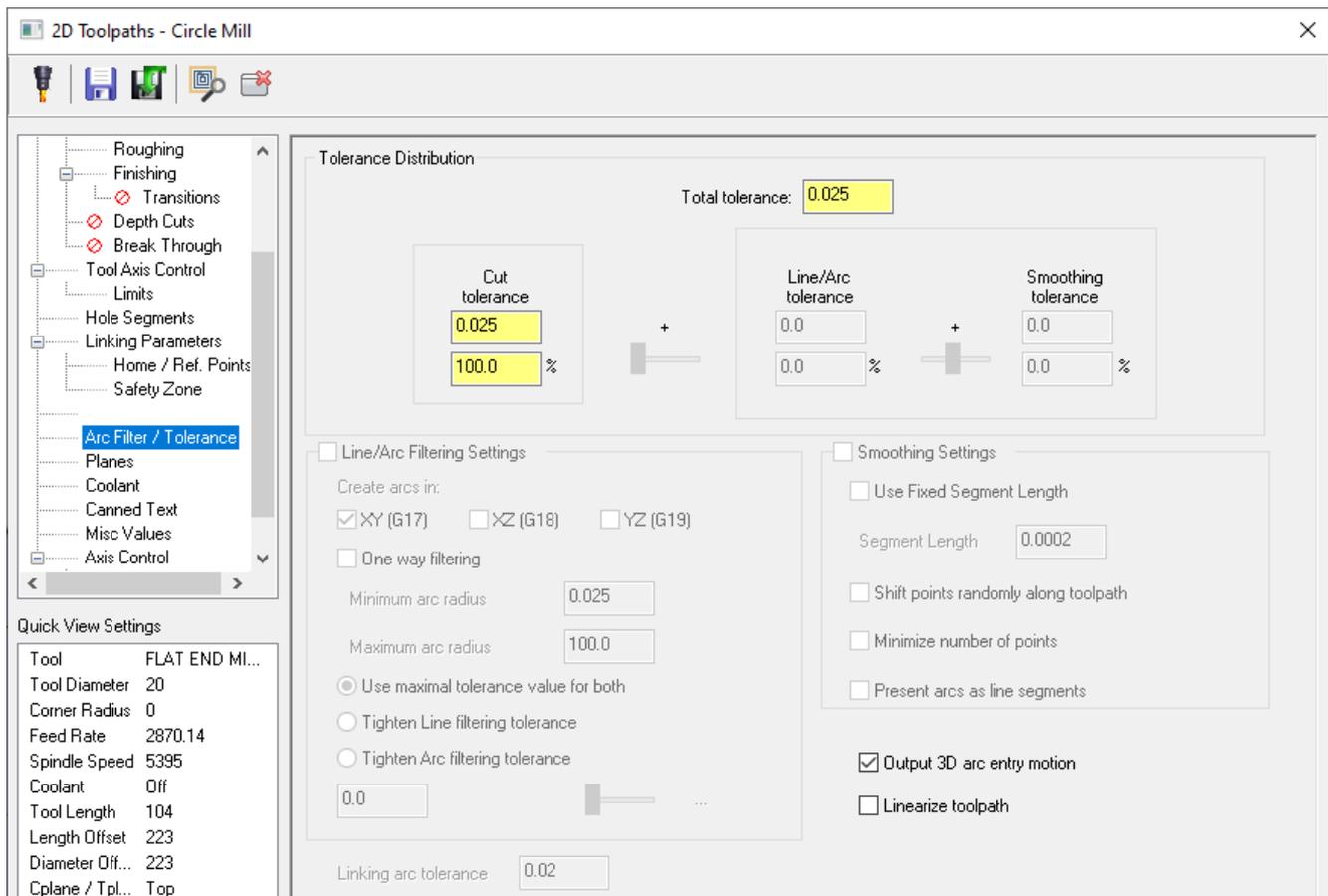


The second picture shows the orientation when **Axis** is selected.



Using the Arc Filter/Tolerance Page for Hole Making Toolpaths

The **Arc Filter/Tolerance** page has been added to hole making toolpaths' tree-style dialog boxes. Use this page to control toolpath tolerance, which Mastercam uses to smooth and refine motion wherever possible.

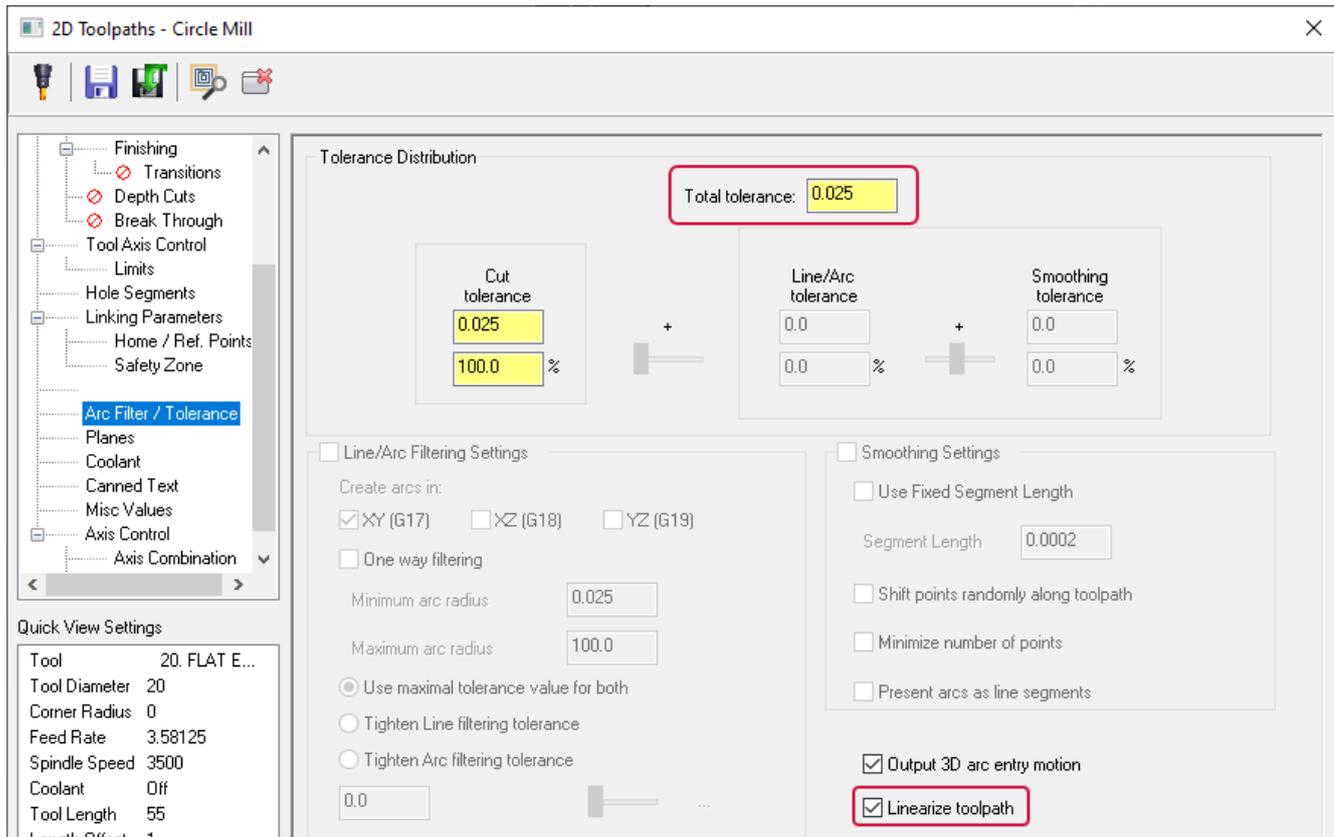


This update also makes the following changes for the **Arc Filter/Tolerance** page:

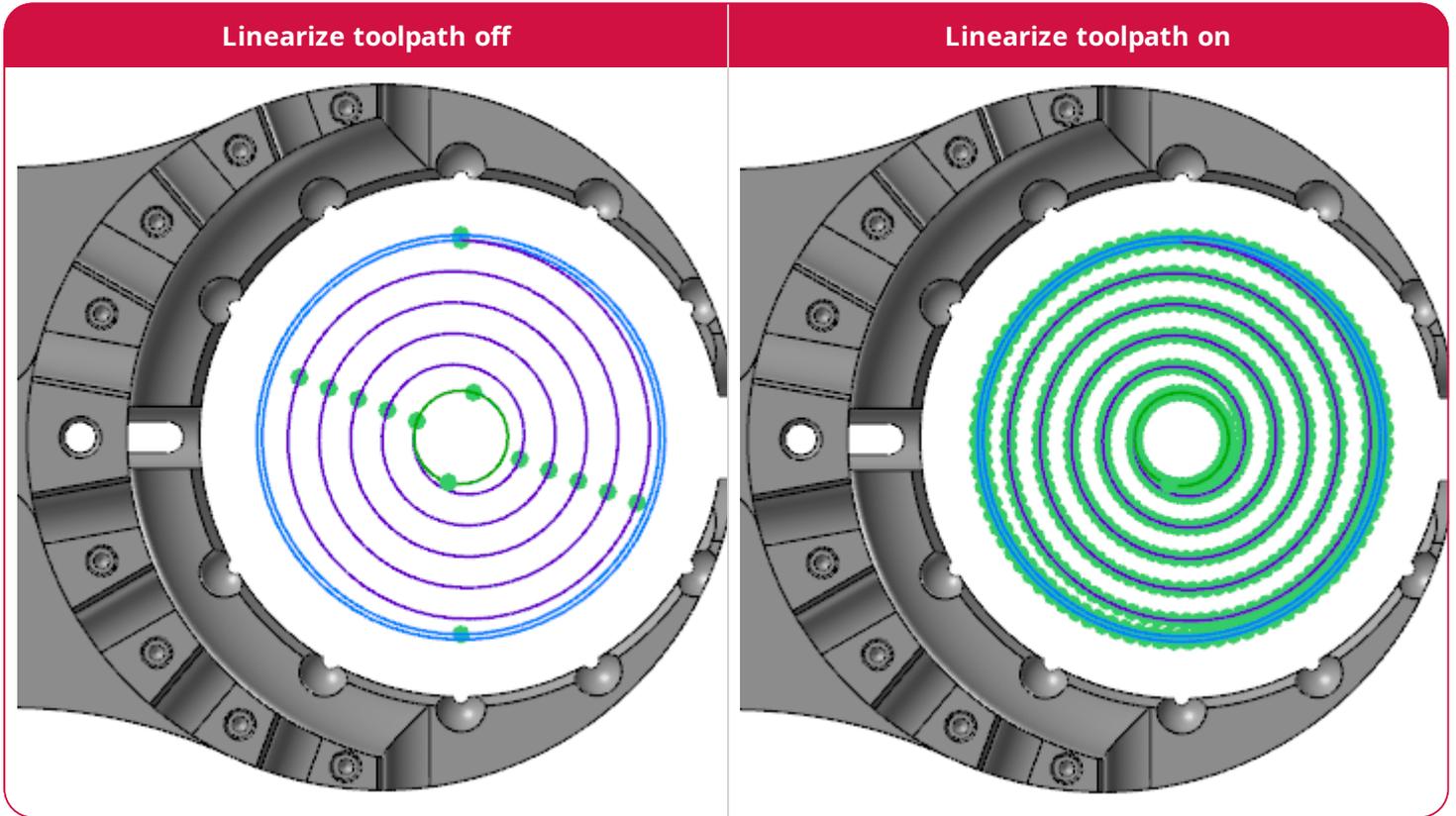
- **Output 3D arc entry motion** has been moved from the Circle Mill **Roughing** page to the new **Arc Filter/Tolerance** page.
- **Output arc moves for helices** has been moved from the Helix Bore **Rough/Finish** page to the new **Arc Filter/Tolerance** page.
- **Linearize helices** has been moved from the Thread Mill **Cut Parameters** page to the new **Arc Filter/Tolerance** page.
- **Tolerance** has been moved from the three aforementioned pages to the new **Arc Filter/Tolerance** page.

Linearizing the Circle Mill Toolpath

You can now linearize a Circle Mill toolpath with the new **Linearize toolpath** option located on the new **Arc Filter/Tolerance** page.



The toolpath is linearized to the specified **Total tolerance** value, also located on the **Arc Filter/Tolerance** page. This functionality allows you to individually control which parts of the Circle Mill toolpath are linearized. Previously, you could only control the entry motion.



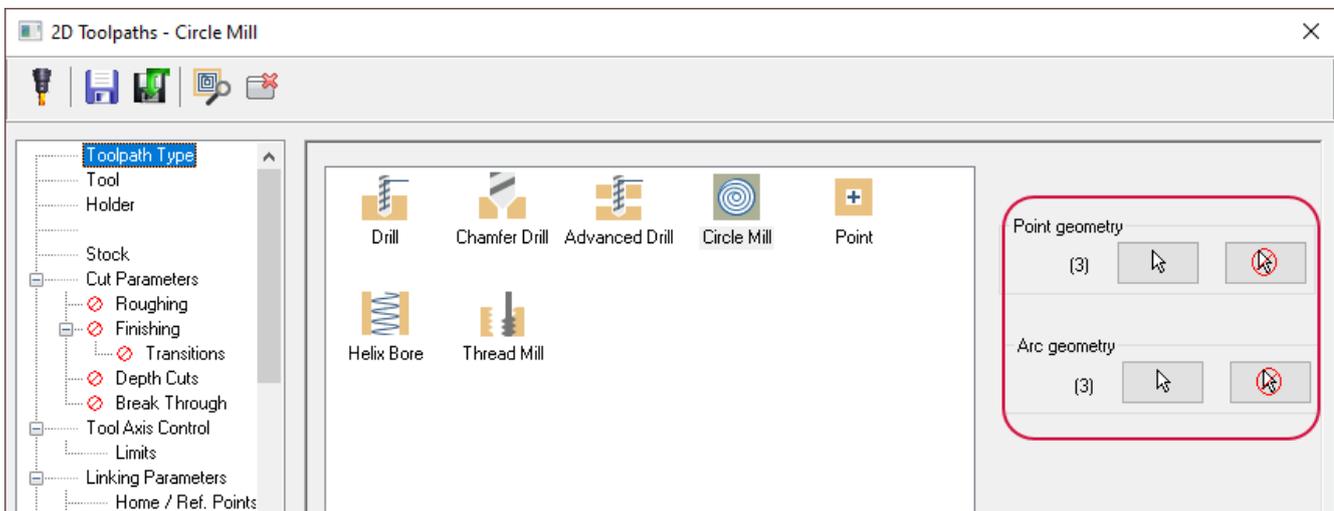
NOTE

For more information about the new **Arc Filter/Tolerance** page, refer to ["Using the Arc Filter/Tolerance Page for Hole Making Toolpaths"](#) on page 38.

Changes to the Toolpath Type Page

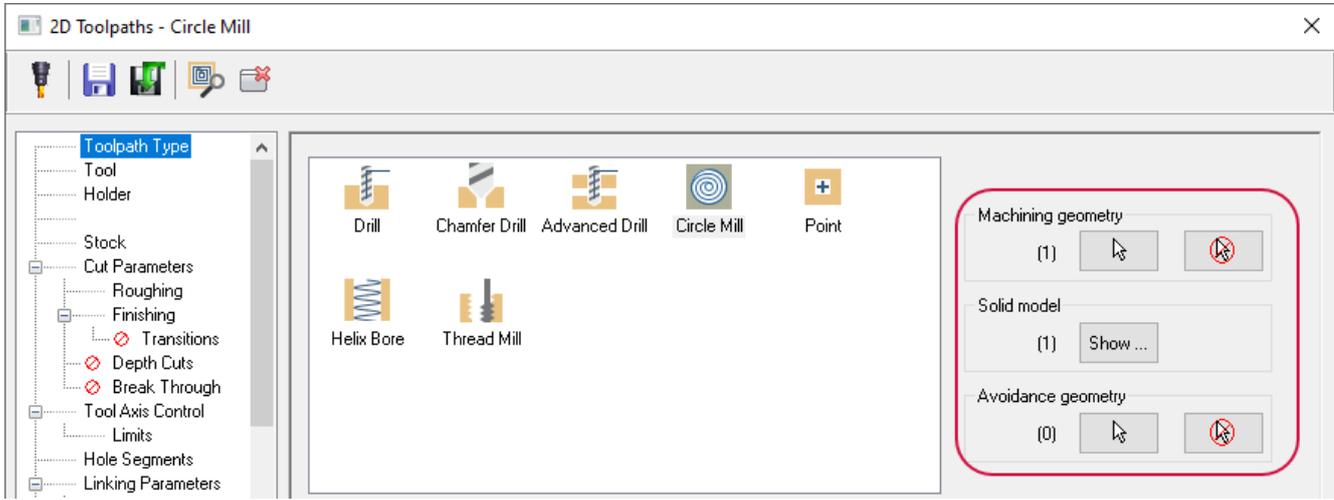
The geometry options on the **Toolpath Type** page have changed for Mastercam 2022. Previously, you could define two types of geometry: **Point geometry** and **Arc geometry**.

2021:

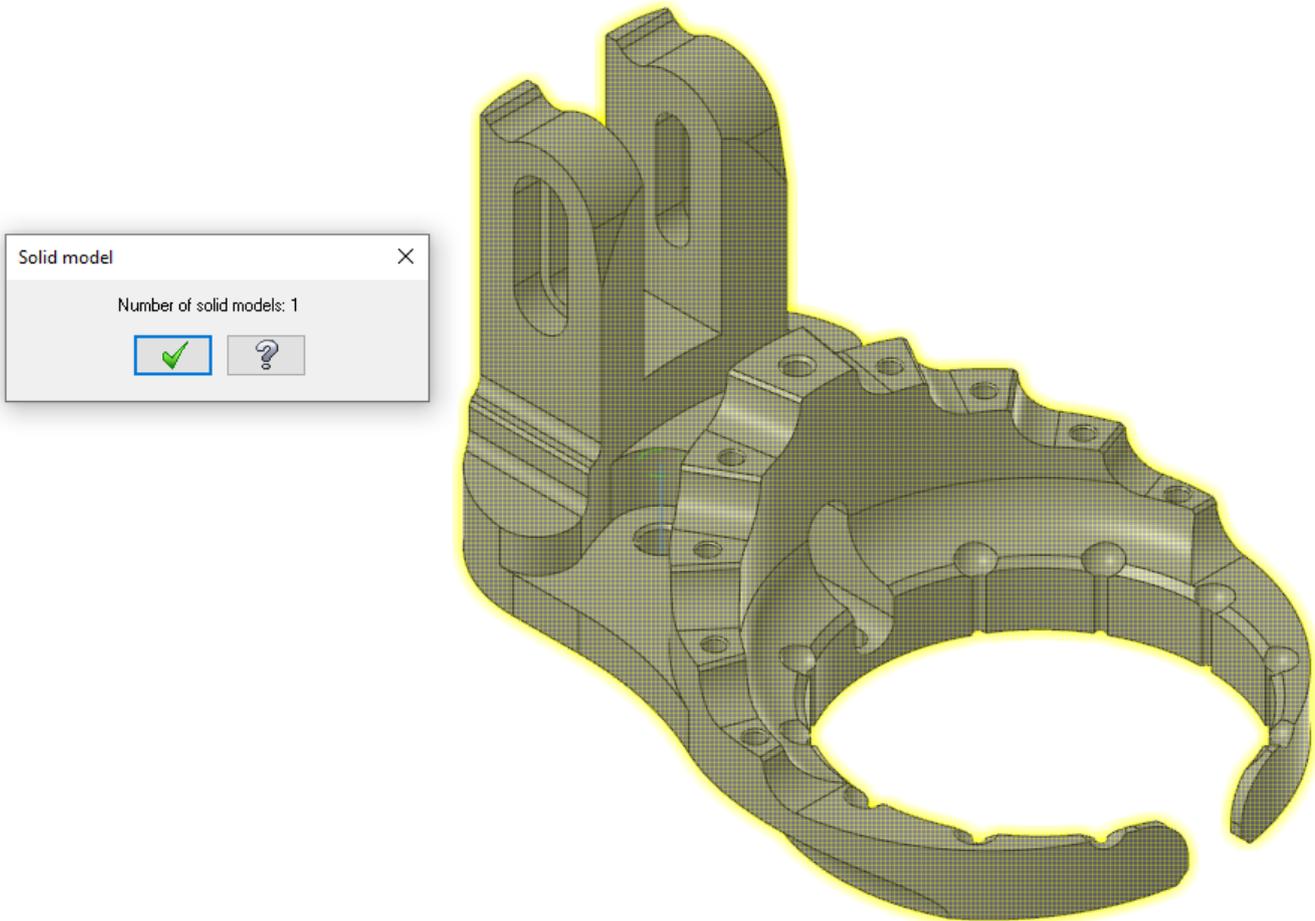


In Mastercam 2022, **Point geometry** has been renamed to **Machining geometry**, and **Arc geometry** has been removed. This change standardizes and simplifies the interface and geometry selection across Mastercam's toolpath interfaces. Standardizing the geometry selection means that, you can now select **Avoidance geometry** for hole making toolpaths.

2022:



Hole making toolpaths now automatically associate to the solid models from which the **Machining geometry** was selected. Select **Show** under the **Solid model** group to hide the dialog box and highlight the solids associated with the toolpath in the graphics window.

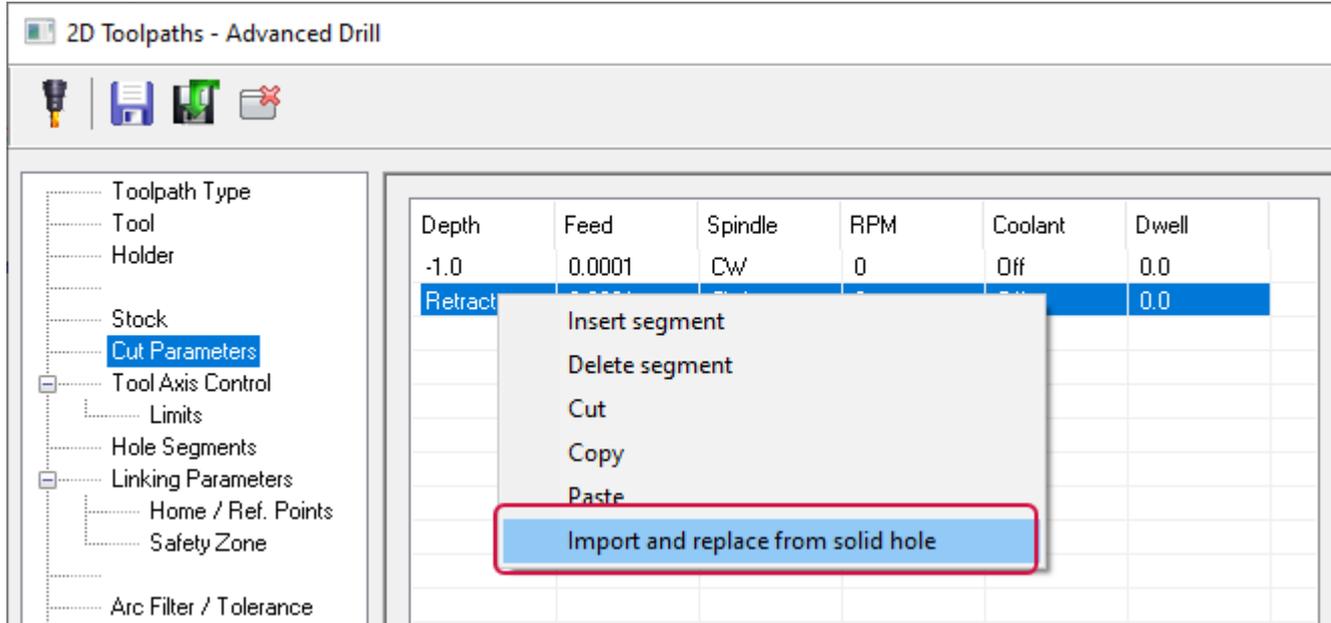


Advanced Drill Enhancements

Listed below are enhancements to the Advanced Drill toolpath.

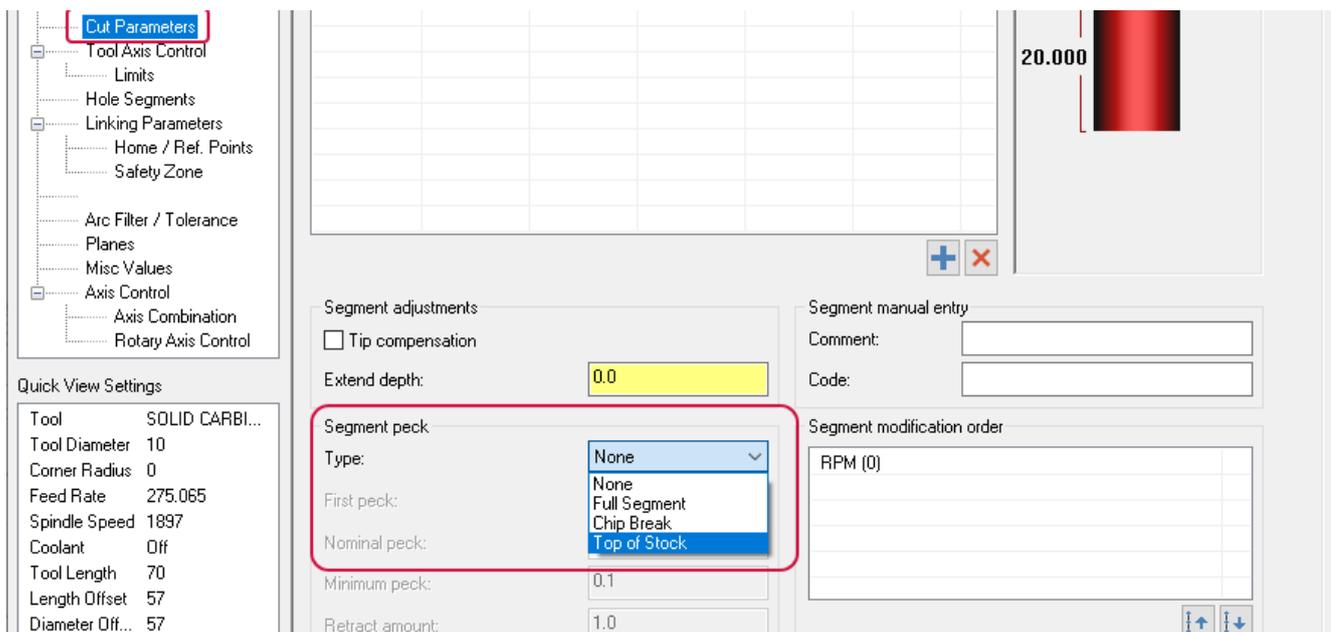
Populating Drill Segments From a Solid Hole

If a solid hole operation is selected as the machining geometry for the Advanced Drill toolpath, you can use the right-click menu of the grid in the **Cut Parameters** page to import and replace segments from the solid hole operation. This new option imports segments from the solid hole operation, replacing any segments except for the retract.



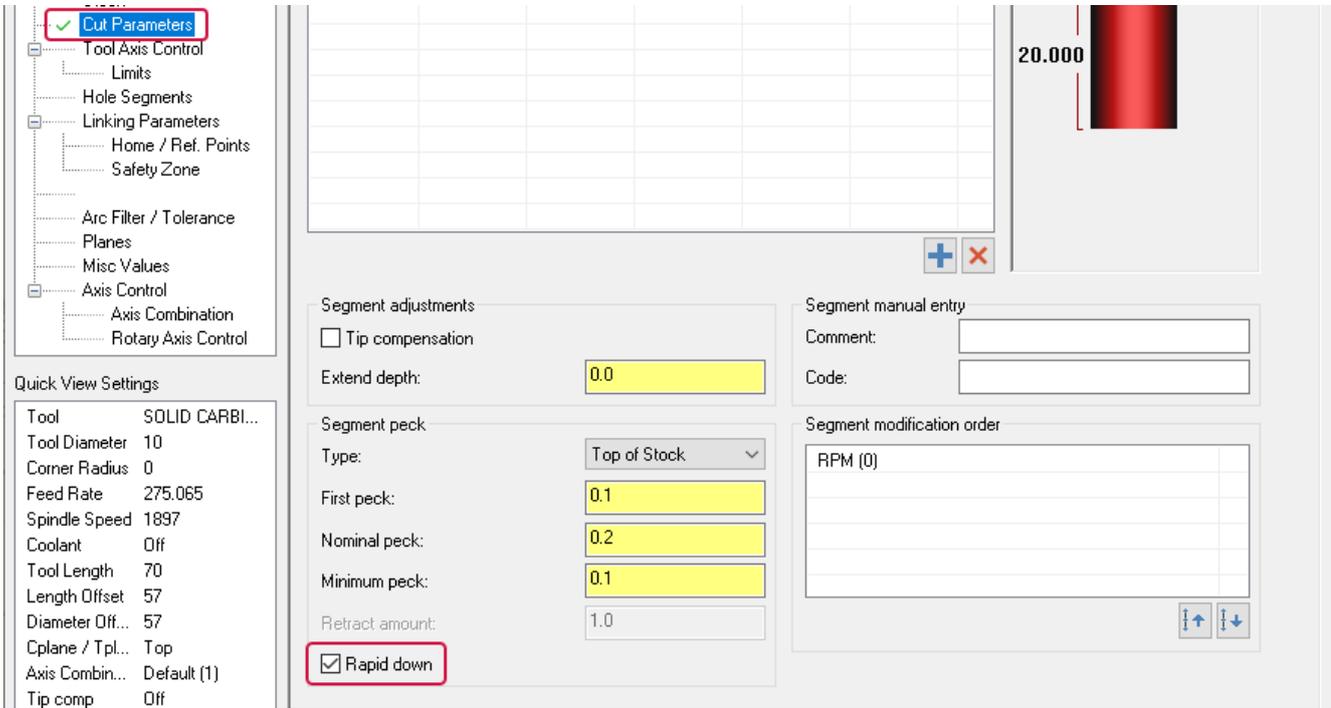
Using a Rapid Move to the Top of Stock Position

When you choose **Top of Stock** from the **Type** drop-down on the **Cut Parameters** page, the tool will rapid to the **Top of Stock** position for every peck. Use this option when you need to get to the top of a part, not just the top of a segment, to clear chips.



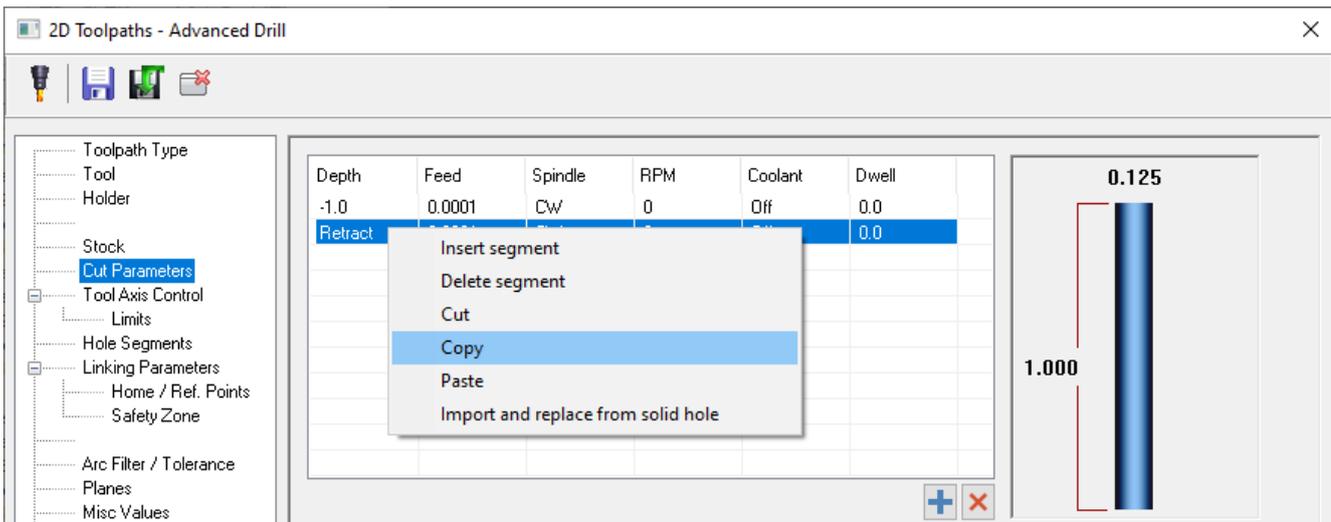
Rapiding Down to the Initial Toolpath Position

A new option, **Rapid down**, is now available on the **Cut Parameters** page. When selected, the toolpath rapids back down to the initial position before drilling further. For example, if you are pecking a hole from **0mm** to **-20mm** with a **5mm** peck, the first drill will go to **-5**, rapid to **0**, rapid back to **-5**, and then drill to **-10**. This new option saves cycle time.



Cutting, Copying, and Pasting Drill Segments

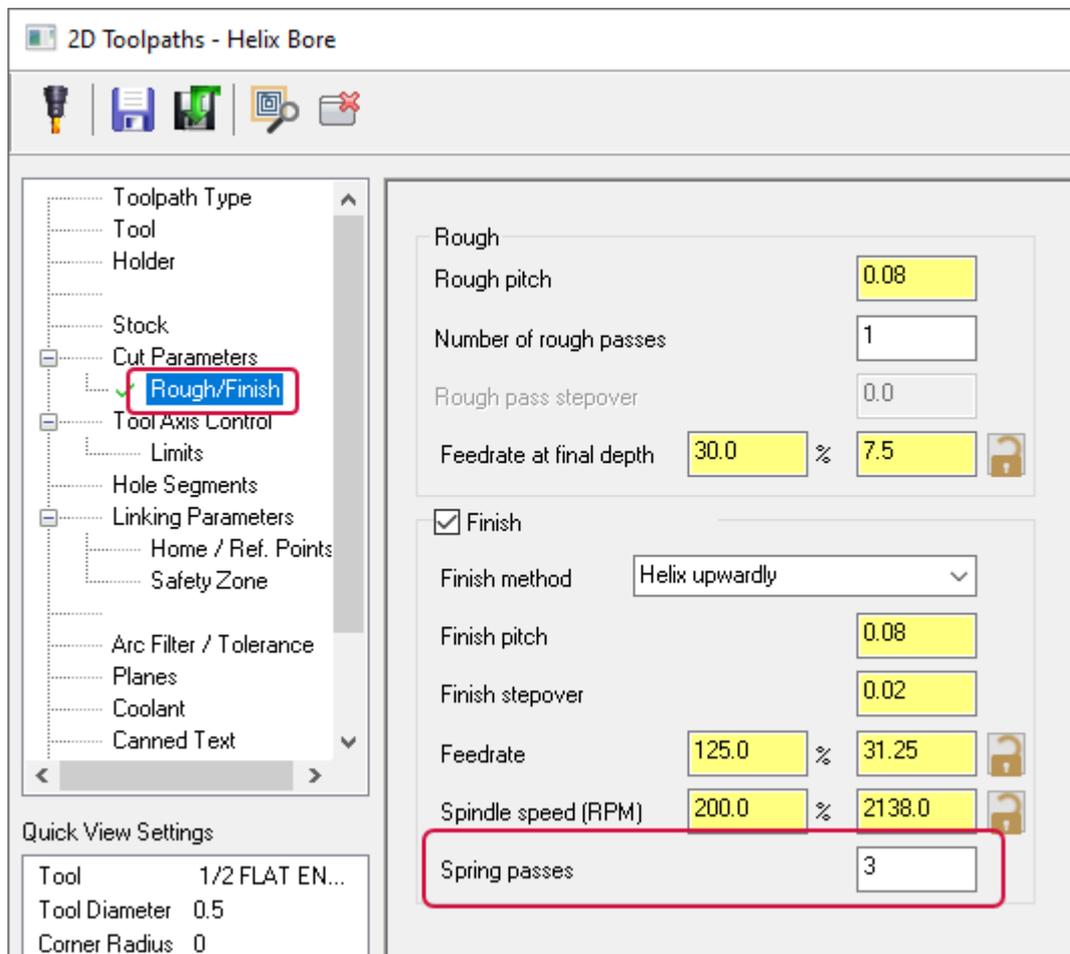
You can now cut, copy, and paste drill segments on the **Cut Parameters** page by using the right-click menu. These changes can reduce data entry time by letting you use existing segments as the basis for new ones.



Helix Bore Toolpath Enhancements

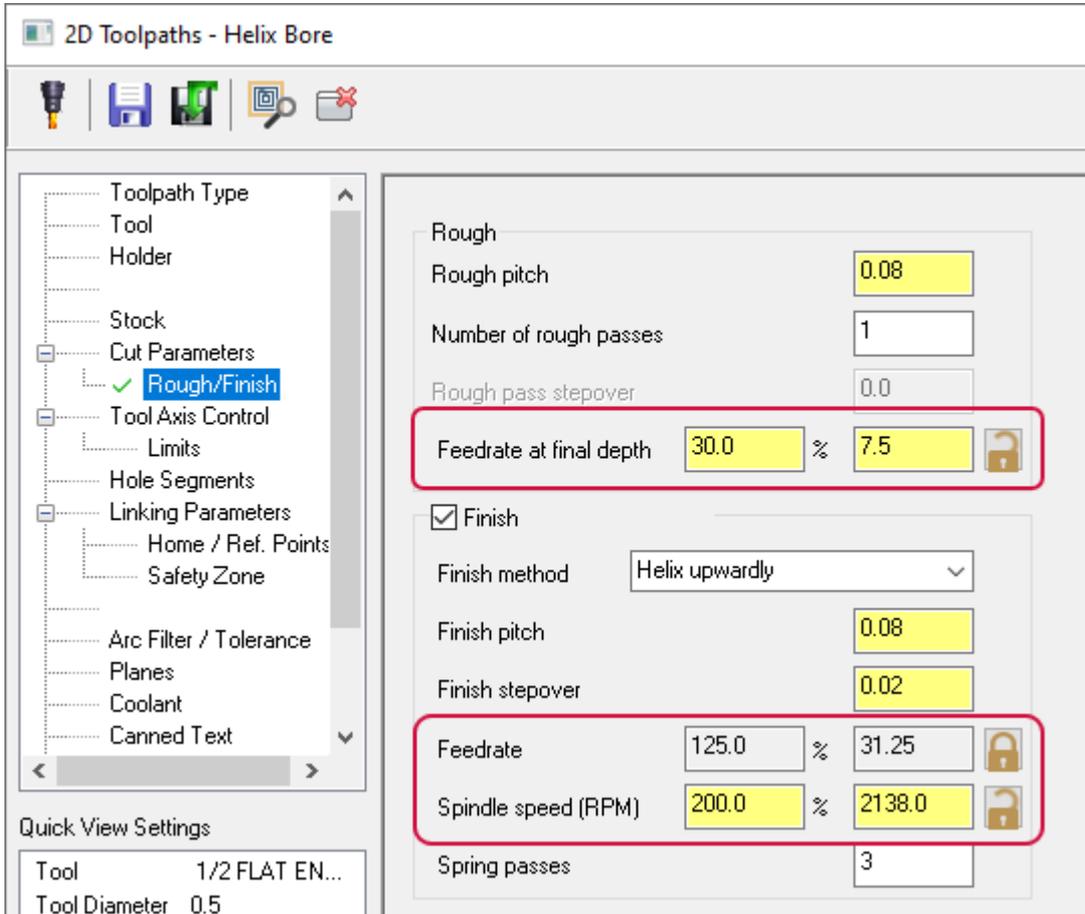
Adding Spring Passes to Helix Bore

You can now add spring passes to Helix Bore toolpaths. Spring passes create additional finish passes along the same path as the last finish pass. Add these spring passes by entering a value in the **Spring passes** field on the **Rough/Finish** page.



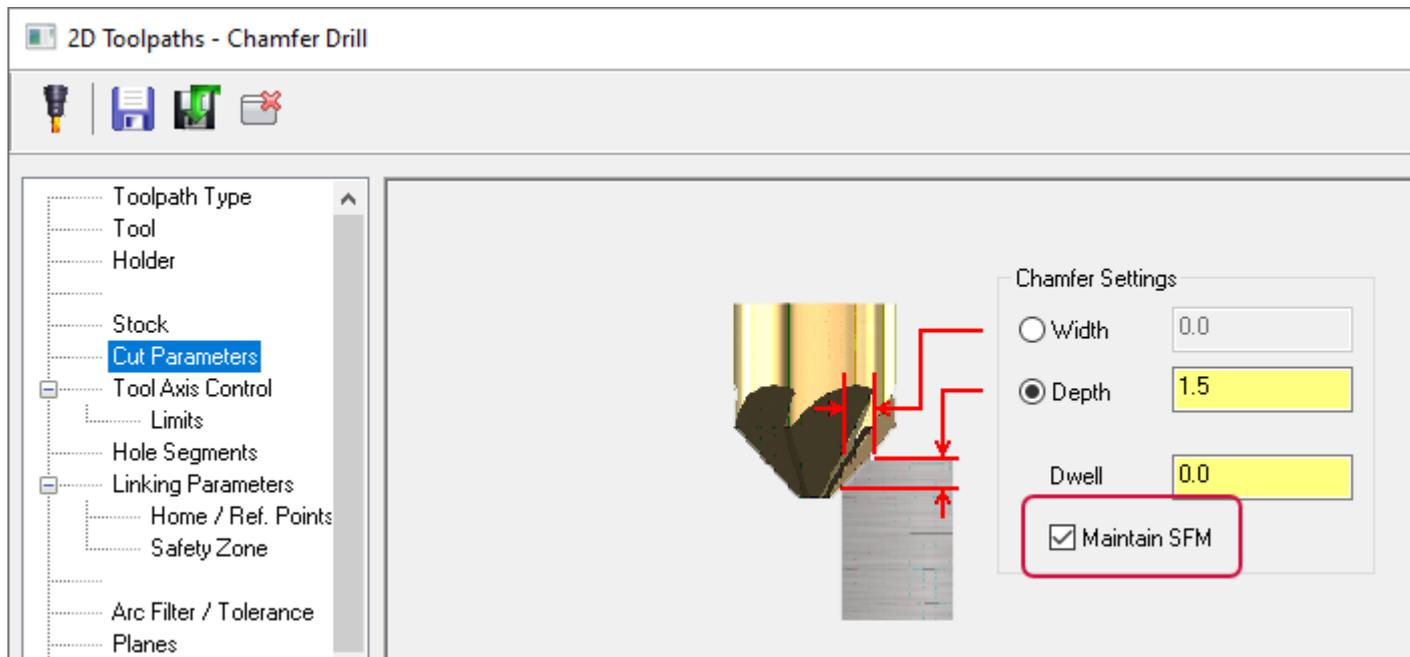
Locking Values for Helix Bore Toolpaths

You can now lock the **Feedrate at final depth**, **Feedrate**, and **Spindle speed** values on the **Rough/Finish** page for Helix Bore toolpaths. Locking these fields prevents them from being automatically updated when parameters change on the **Tool** page.



Maintaining CS/SFM in Chamfer Drill

The new **Maintain CS/SFM** option on the **Cut Parameters** page of Chamfer Drill toolpaths varies the spindle speed for each hole to maintain the cutting speed (CS) in surface meters per minute or surface feet per minute (SFM). This new option helps when machining different sized holes, as the CS or SFM are generally calculated based on the diameter of the tool. This option increases the spindle speed for smaller holes to maintain the proper CS/SFM.

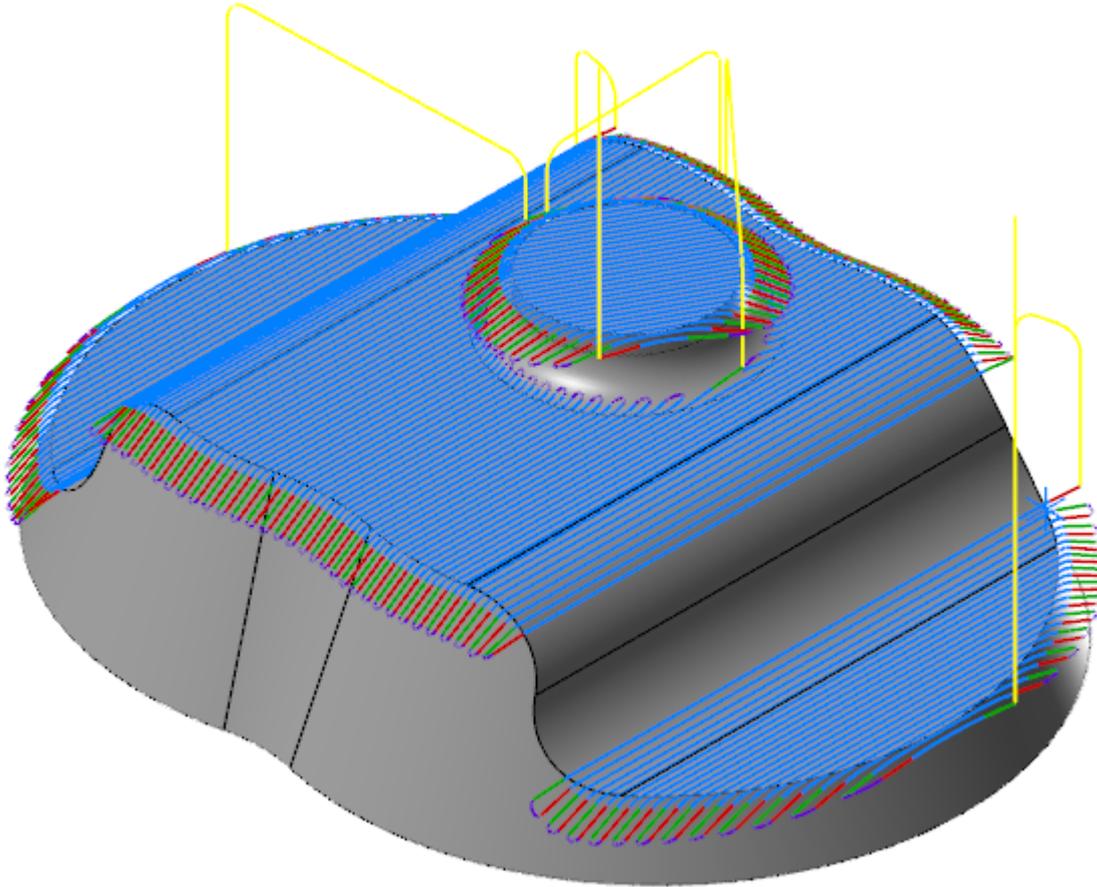


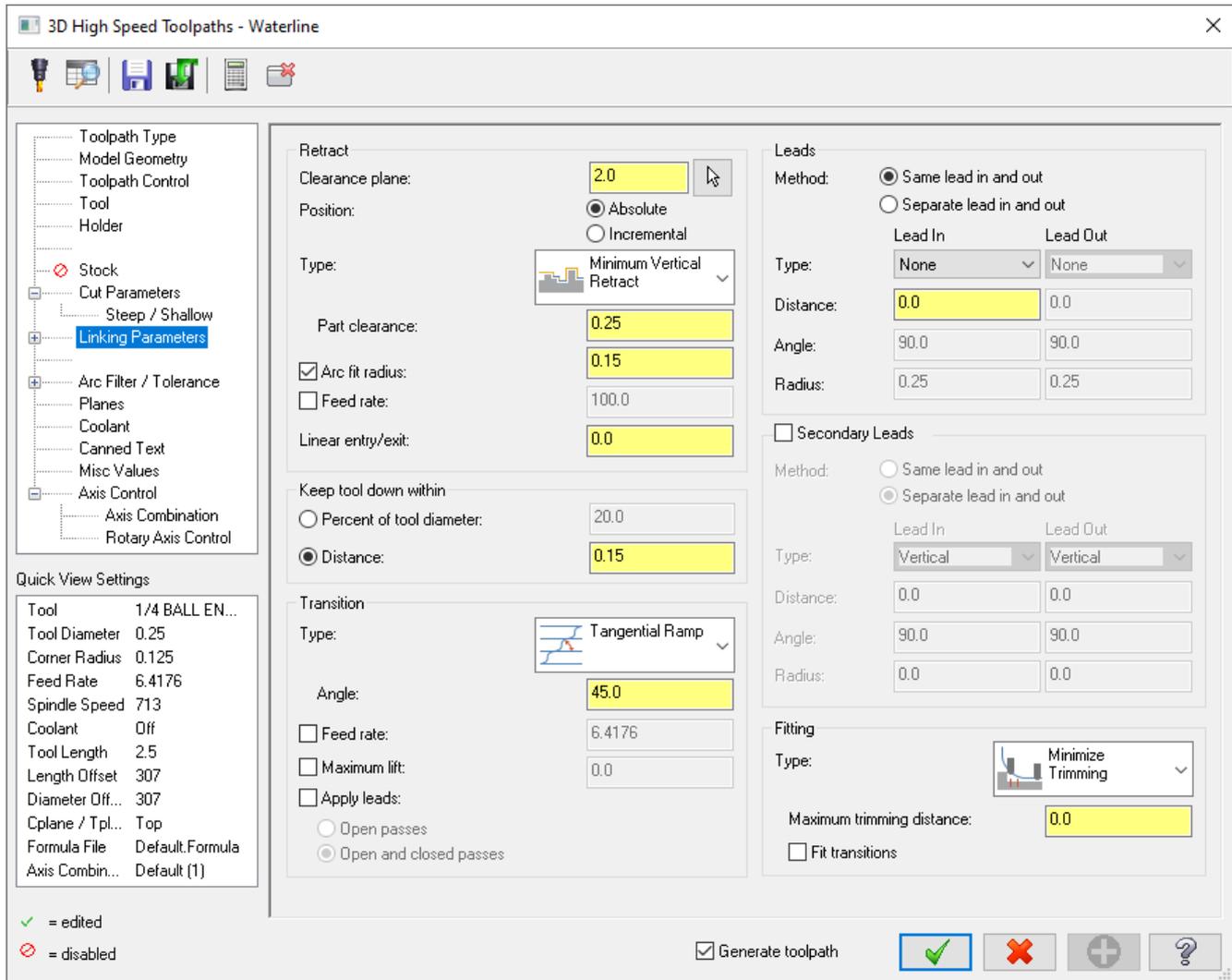
3D Enhancements

Listed below are enhancements to the 3D toolpaths.

More Powerful and Flexible 3D Linking

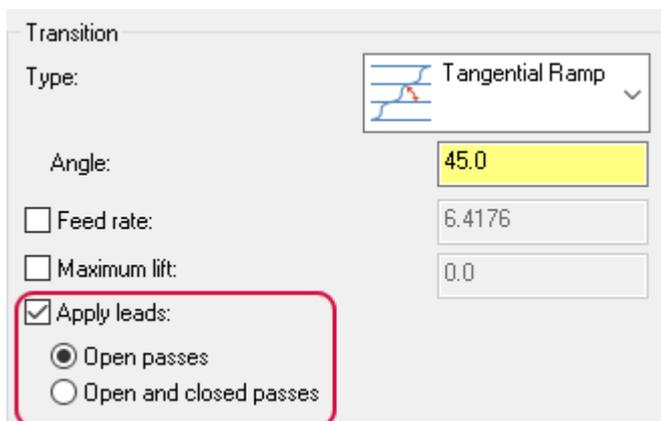
The **Linking Parameters** page for 3D high speed toolpaths (except for Dynamic OptiRough, Horizontal Area, and Area Roughing) has been redesigned and now includes new and improved functionality.

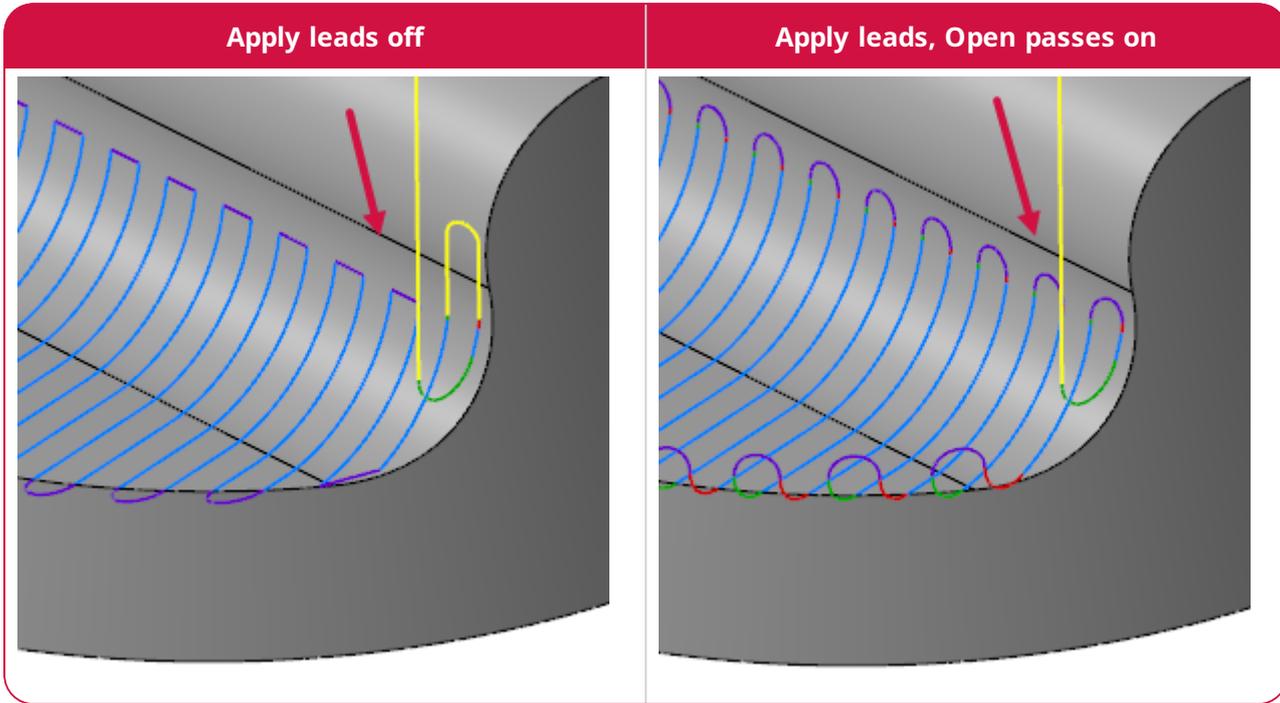




Adding Leads to Transition Moves

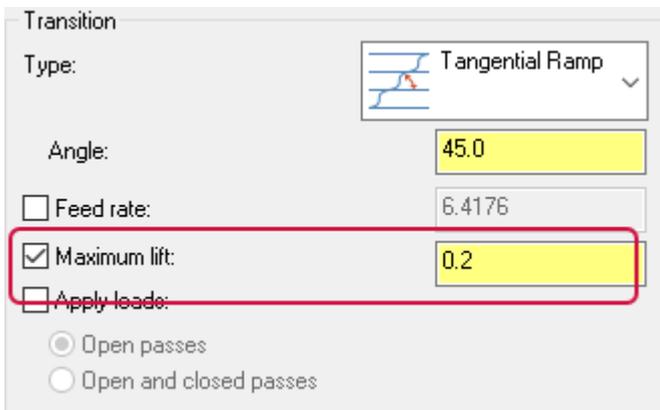
The new **Apply leads** option allows you to add leads to transition moves. For example, you can add a vertical arc entry or exit move to a pass and then use the smooth transition to connect raw passes. You can apply these leads to open passes or both open and closed passes.

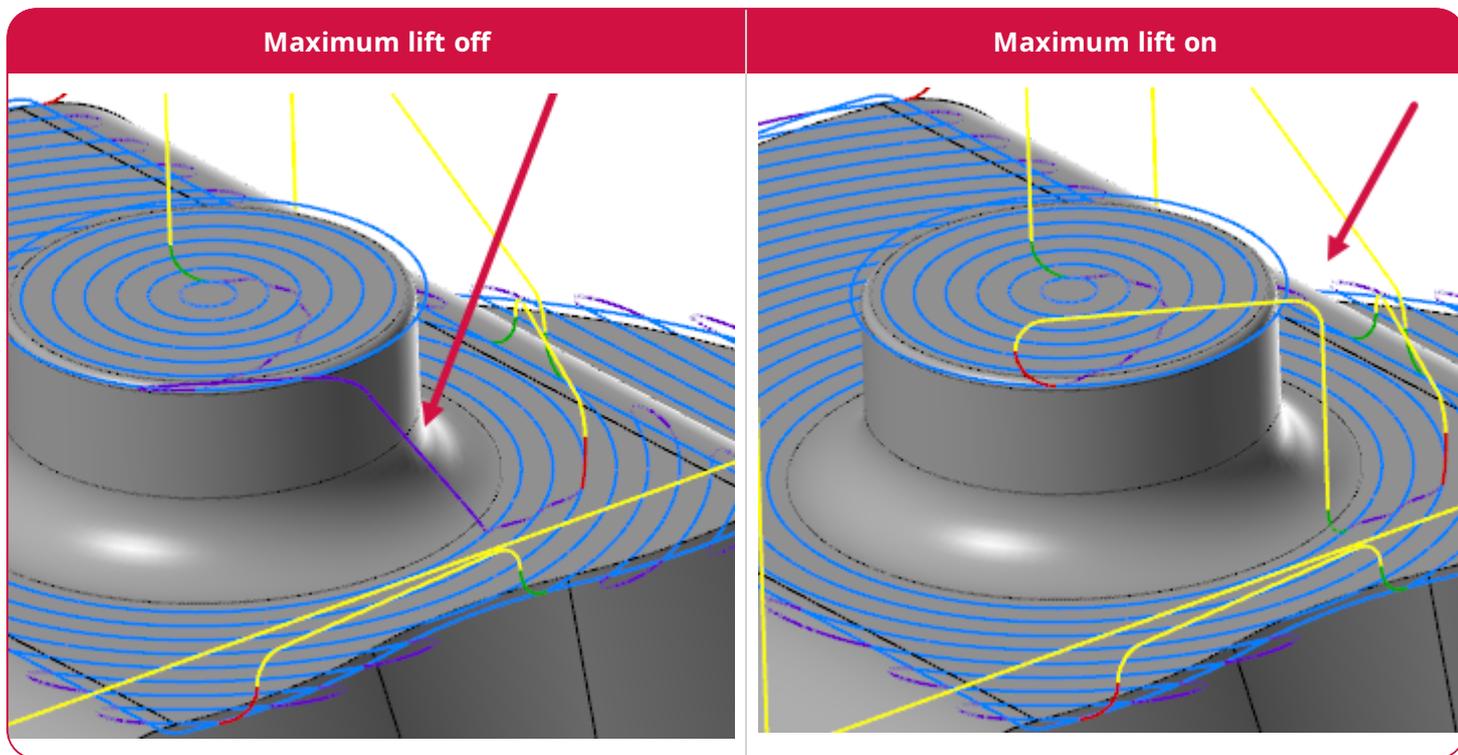




Avoiding Tall, Long, or Steep Transition Moves

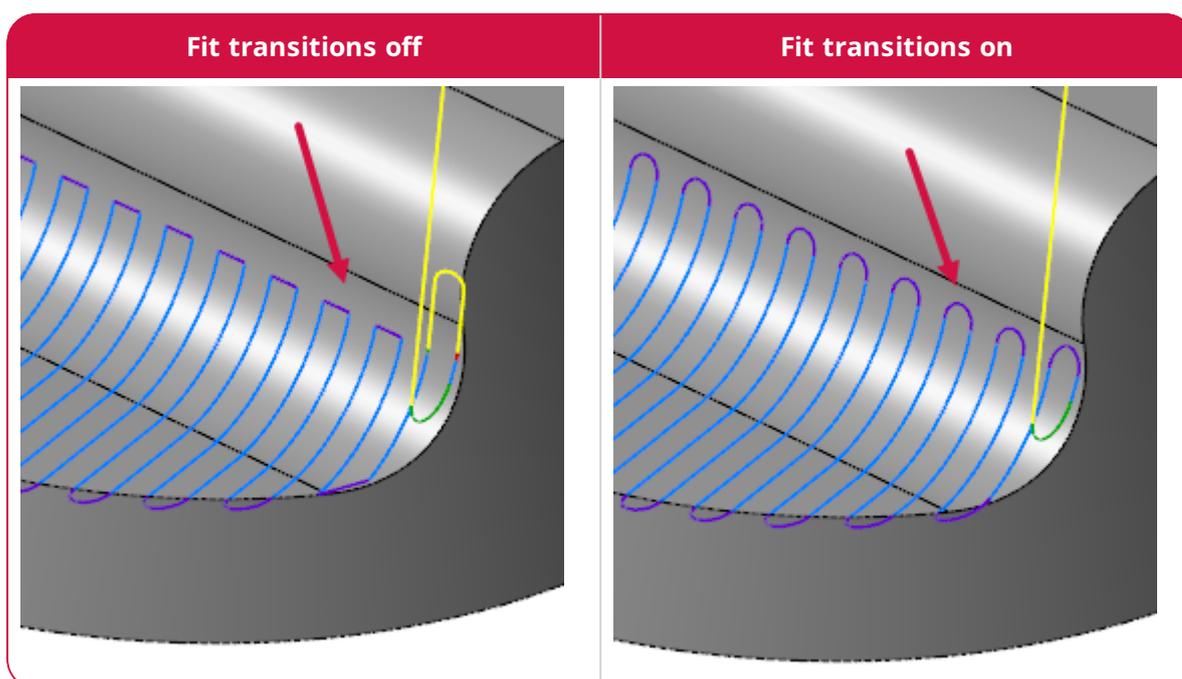
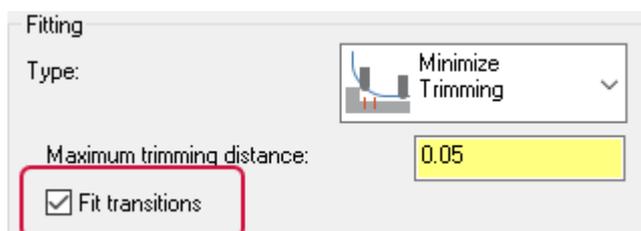
The new **Maximum lift** option helps to avoid long, tall, or steep transition moves. After the transition moves are calculated, **Maximum lift** analyzes the transition moves for a total height in Z and compares it to the **Maximum lift** distance. If the transition moves exceed the **Maximum lift** distance, the transition moves are replaced by a simple retract.





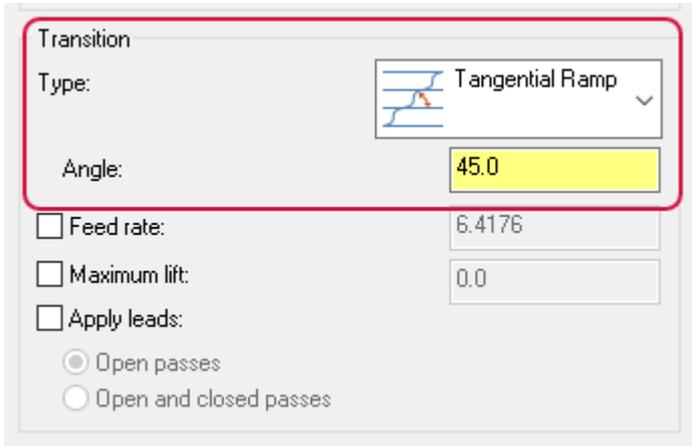
Trimming Passes to Fit Transition Moves

The new **Fit transitions** option trims the passes to safely fit the transition moves. In previous Mastercam releases, raw passes were only trimmed to safely accommodate the retract and lead connection moves, not transition moves.



Additional Changes to the Linking Parameters Page

The **Transitions** page has been removed and the relevant options moved to the **Transition** group in the **Linking Parameters** page. Use these options to configure the entry move that the tool will make as it transitions to new Z levels.



Transition

Type:  Tangential Ramp

Angle: 45.0

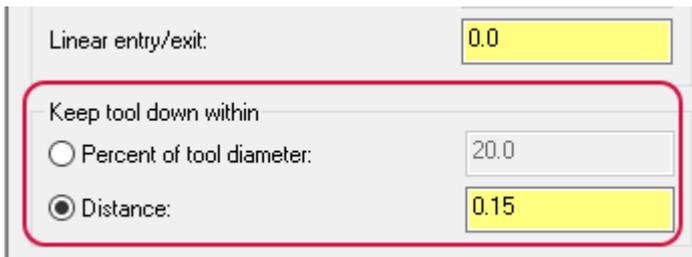
Feed rate: 6.4176

Maximum lift: 0.0

Apply leads:

- Open passes
- Open and closed passes

The **Keep tool down within** option has been moved from the **Cut Parameters** page to the **Linking Parameters** page. This option prevents the tool from retracting between passes.



Linear entry/exit: 0.0

Keep tool down within

- Percent of tool diameter: 20.0
- Distance: 0.15

The **Curl up** and **Curl down** options have been consolidated into a single **Arc fit radius** option in the **Retract** group. **Arc fit radius** is available for all retract types, and rounds the sharp moves found in all retract moves.



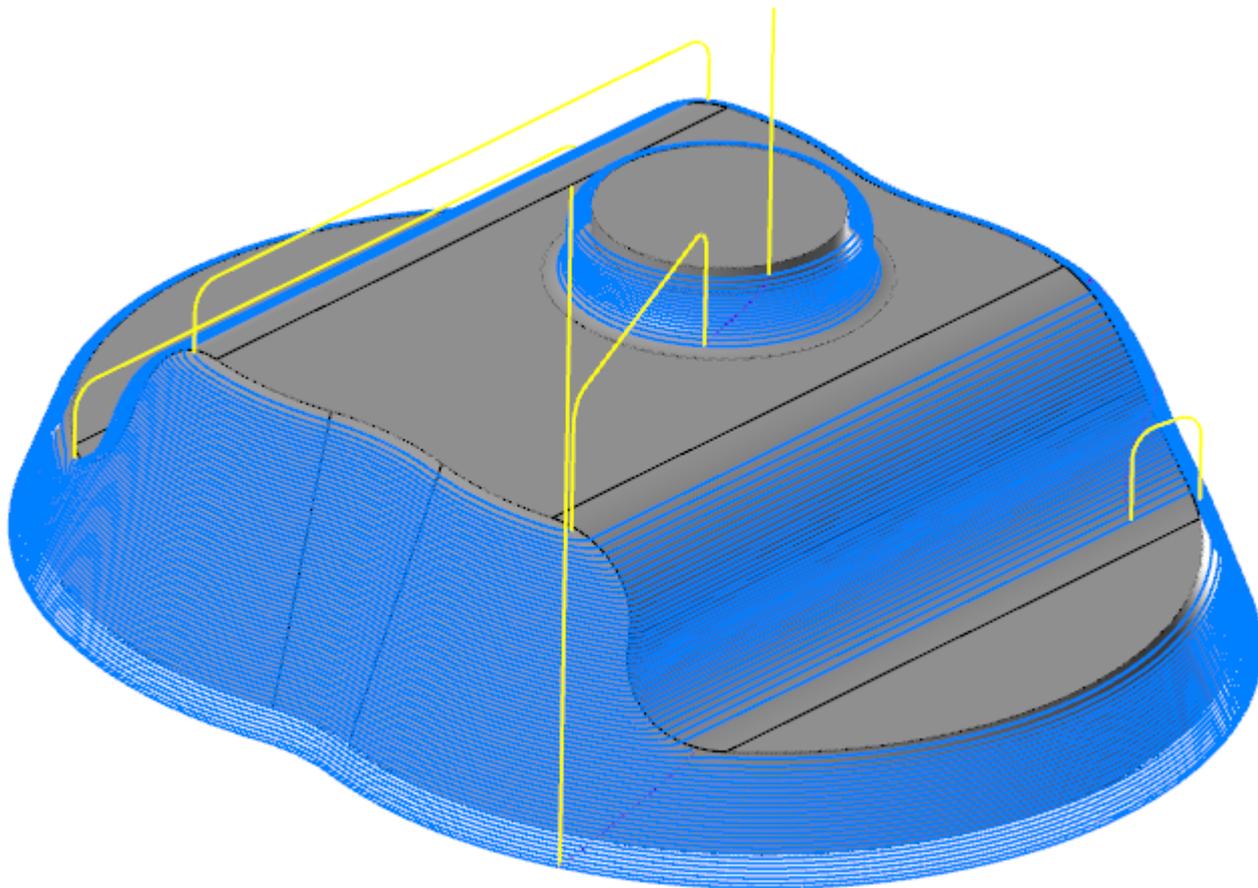
Type:  Minimum Vertical Retract

Part clearance: 0.25

Arc fit radius: 0.15

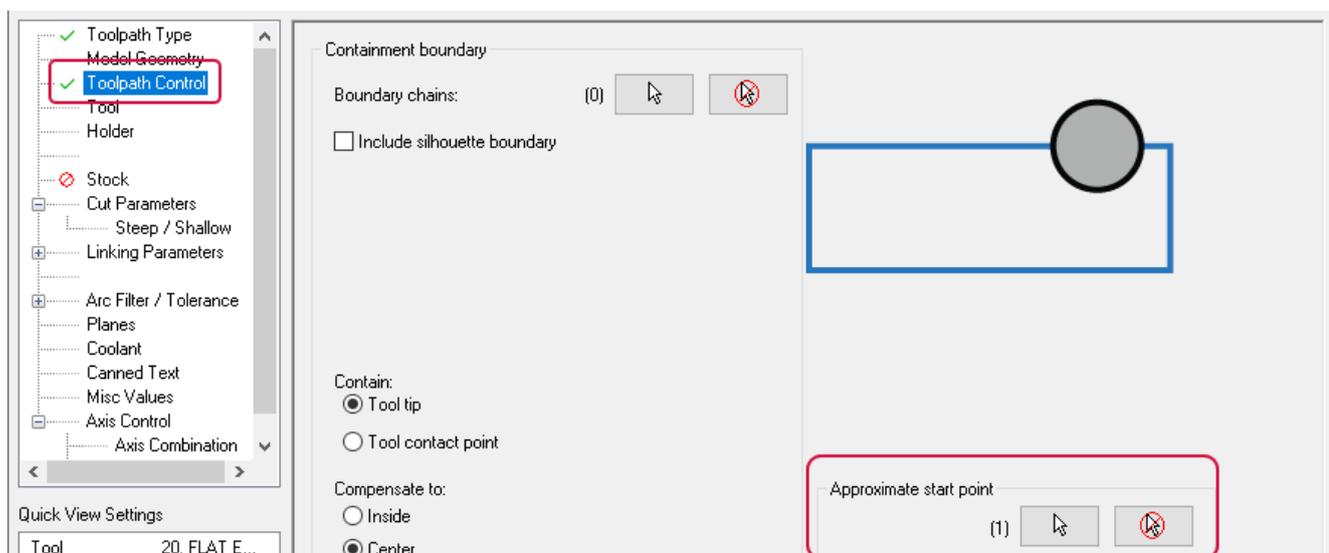
Feed rate: 100.0

Linear entry/exit: 0.0



Improved Approximate Start Point Selection

Mastercam includes two improvements to the **Approximate start point** option, located on the **Toolpath Control** page. **Approximate start point** now includes a count indicator to show that a start point is selected. In previous releases, this was not shown.



Additionally, for Raster and Radial toolpaths, Mastercam has improved the functionality of **Approximate start point** particularly when multiple machining areas are selected. Once an area has been machined the tool moves to the start point of the closest unmachined area.

Dragging and Dropping Geometry Groups

When dragging and dropping geometry from a 3D high speed toolpath to another 3D high speed toolpath, Mastercam checks both the source and destination toolpaths for a single group.

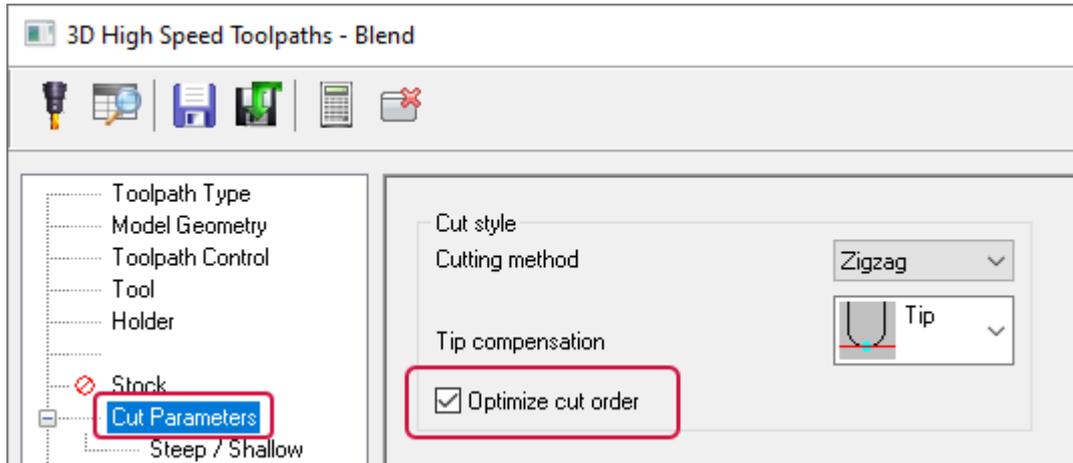
- If only one group is found, the geometry is added to or replaces that group.
- If you select **Add geometry and stock to leave values**, a new group is created in the **Model Geometry** page.
- If there is more than one geometry group in either the source toolpath or the destination toolpath, Mastercam creates a new group in the **Model Geometry** page as in previous releases.

3D High Speed Blend Toolpath Enhancements

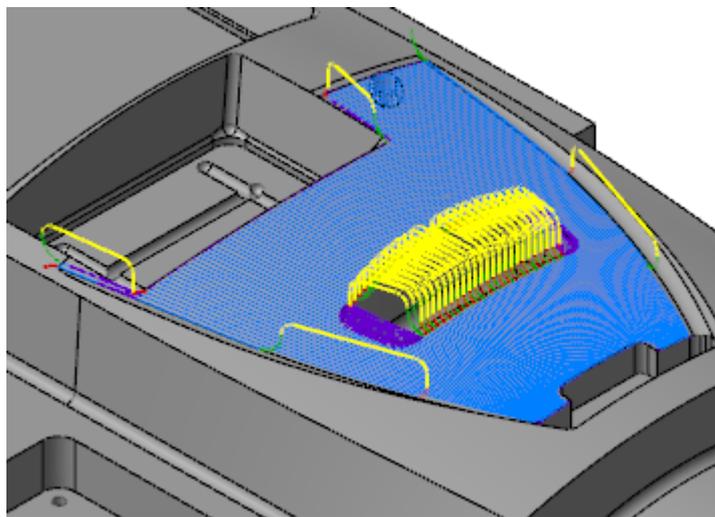
Listed below are the enhancements made to the 3D High Speed Blend toolpath, located in the **3D** gallery of the **Mill Toolpaths** contextual tab.

Optimizing the Cut Order

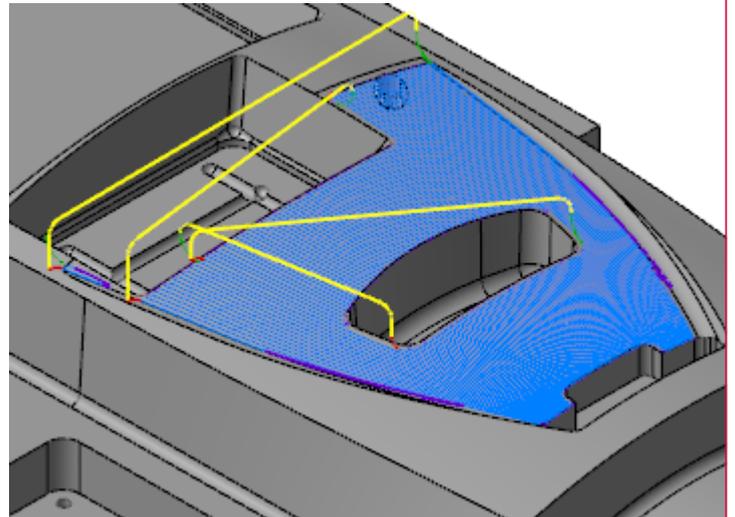
The **Optimize cut order** option is now available on the **Cut Parameters** page. This option keeps a tool within an area until all cuts in that area are completed.



Optimize cut order off

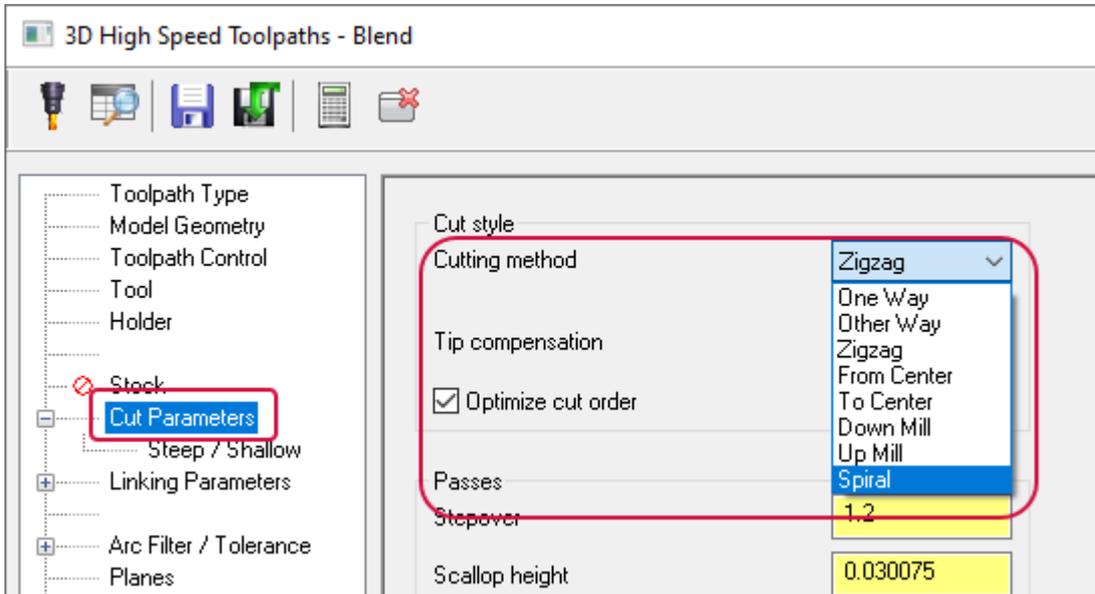


Optimize cut order on



Machining with Spiral Cutting

The 3D High Speed Blend toolpath now includes the **Spiral** cutting method. This new option is located on the **Cut Parameters** page, in the **Cutting method** drop-down.



The **Spiral** option directs the tool to machine in a continuous circular motion from one boundary to another.

3D High Speed Waterline Toolpath Enhancements

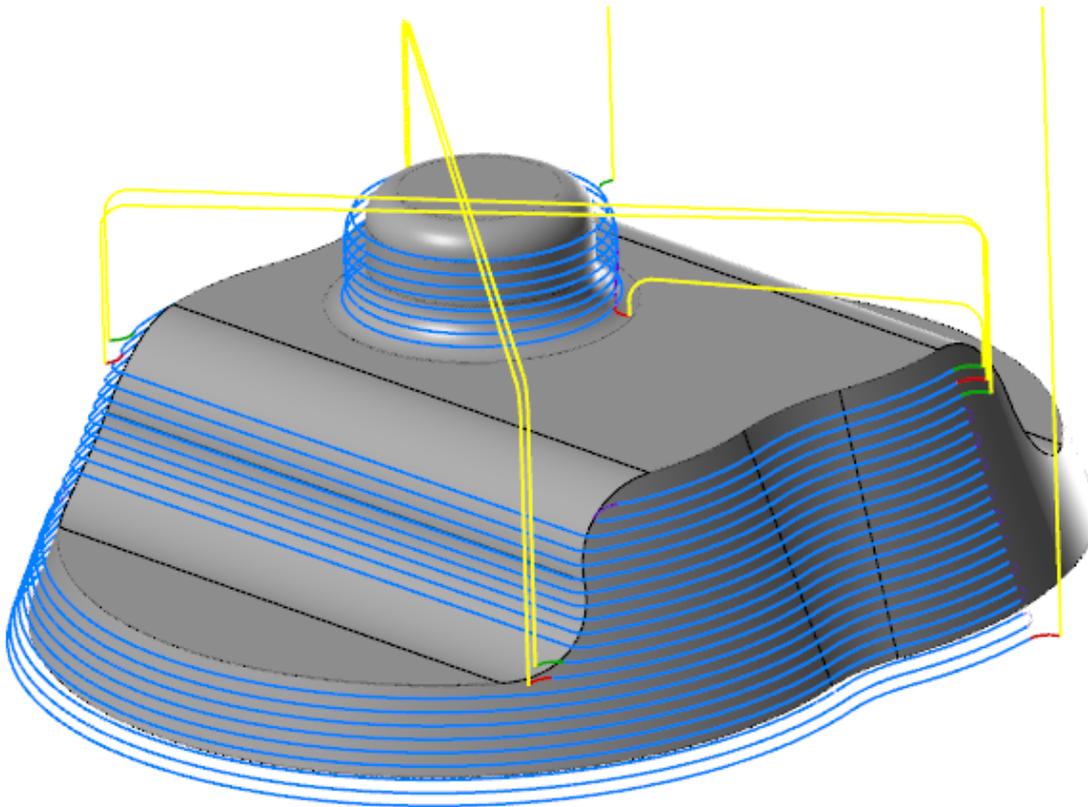
Listed below are the enhancements made to the 3D High Speed Waterline toolpath.

Machining with a Spiral Pattern and New Contour Directions

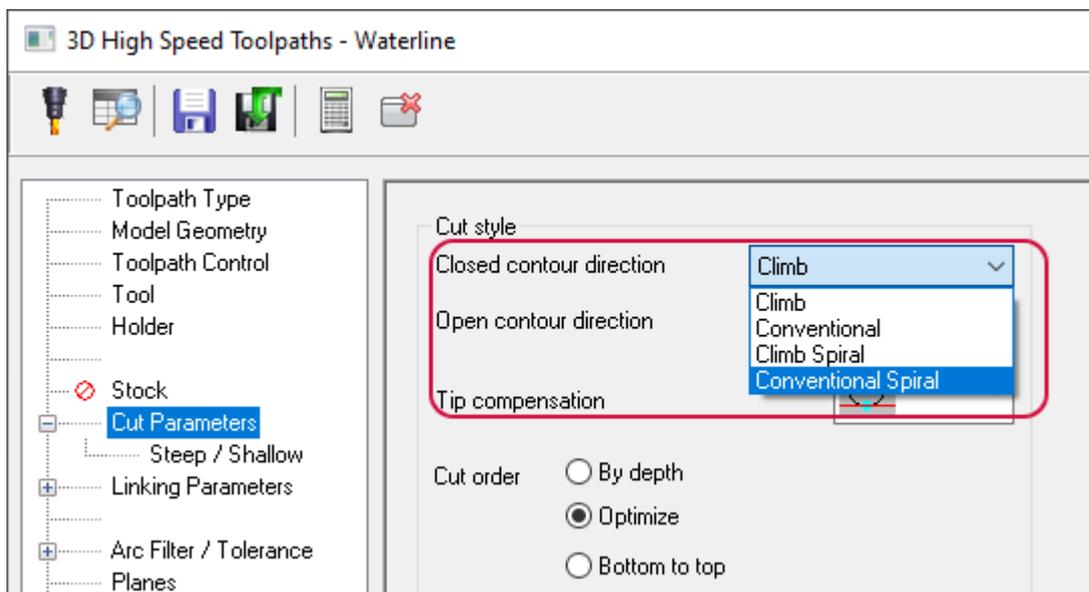
The **Cut Method** drop-down on the **Cut Parameters** page has been replaced with the **Closed contour direction** and **Open contour direction** drop-downs. These new drop-downs allow you create more efficient cutting motion and include reduced retracts on open contours.

- **Closed contour direction:** Sets the cutting direction for closed contours, creating continuous motion without the need for a retract or reversal of direction.
- **Open contour direction:** Sets the cutting direction for open contours.

The example below has **Closed contour direction** set to **Climb** and **Open contour direction** set to **Zigzag**.

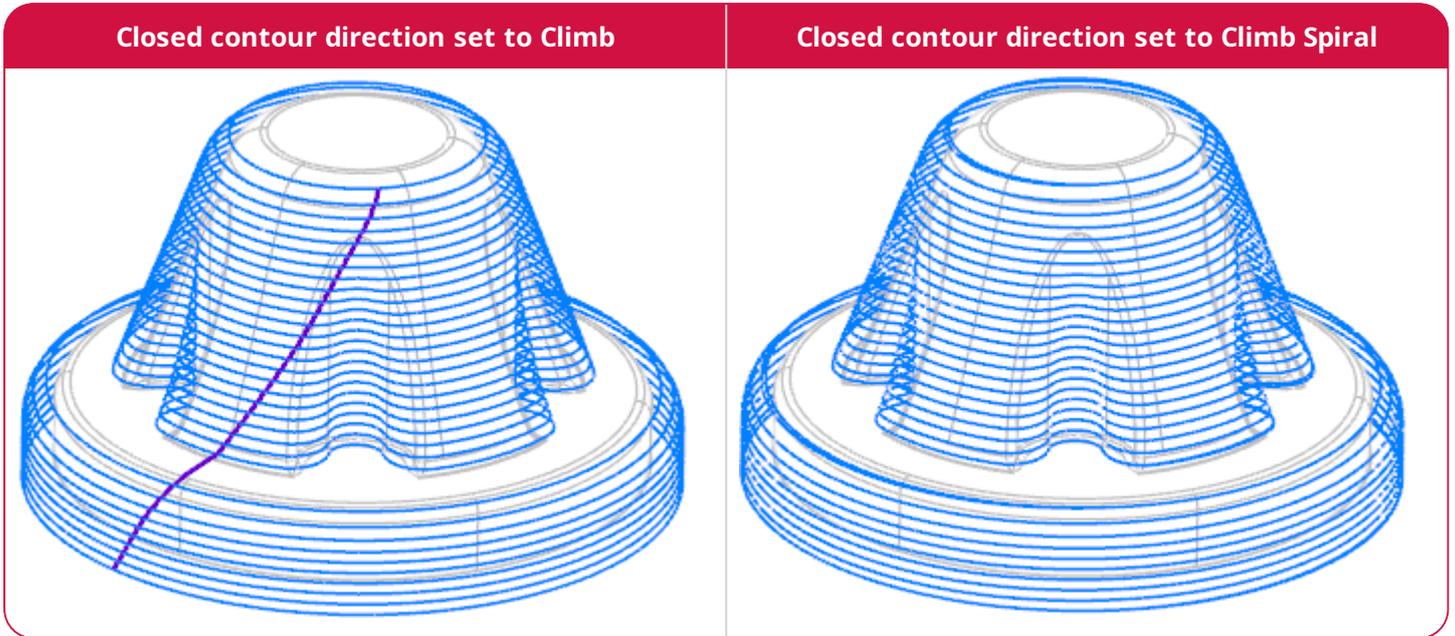


Waterline also includes two new **Closed contour direction** options, **Climb Spiral** and **Conventional Spiral**.



These new cut styles provide cleaner finishes, less machining motion, and fewer retracts. Spiral also helps to avoid creating marks on your part due to stepdown motion.

- **Climb spiral:** Cuts in one direction in a spiral pattern, with tool rotating in the opposite direction of the tool motion.
- **Conventional spiral:** Cuts in one direction in a spiral pattern, with the tool rotating in the same direction as the tool motion.



Machining Flat Areas

You can now determine how a Waterline toolpath handles flat areas in a part. Use the new **Critical Depths** group on the **Cut Parameters** page to further customize this handling.

- Stock
- Cut Parameters**
- Steep / Shallow
- Linking Parameters
- Arc Filter / Tolerance
- Planes
- Coolant
- Canned Text
- Misc Values
- Axis Control
- Axis Combination
- Rotary Axis Control

Quick View Settings

Tool	18 Spherical / ...
Tool Diameter	18
Corner Radius	9
Feed Rate	120
Spindle Speed	1000
Coolant	Off
Tool Length	100
Length Offset	1
Diameter Off...	1
Cplane / Tpl...	Top

Tip compensation Tip v

Cut order By depth
 Optimize
 Bottom to top

Stepdown

Stepdown

Add cuts

Minimum stepdown

Maximum profile stepover

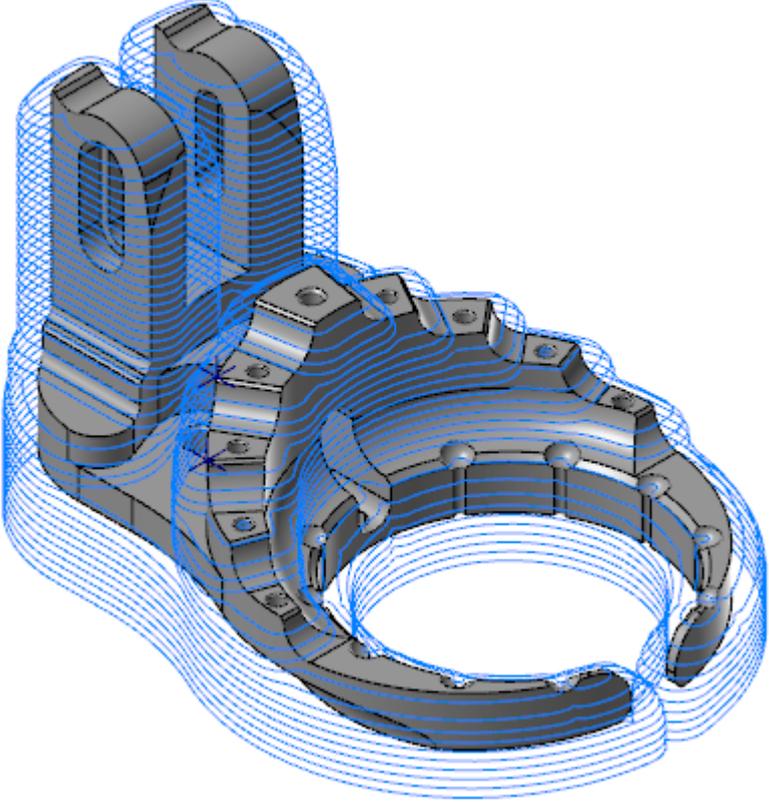
Critical Depths

Type Manual contact only v

Manual (2) v v

Extend

Use the **Type** drop-down options to determine the types of flat areas to be machined, such as **Flats only**, which allows you to machine only flat areas. You can also manually select the flat areas using the **Manual** option, or you can use **Extend** to expand out these flat areas. The following examples show the available option combinations.

Critical Depths options	Toolpath Result
No options set.	

Critical Depths options

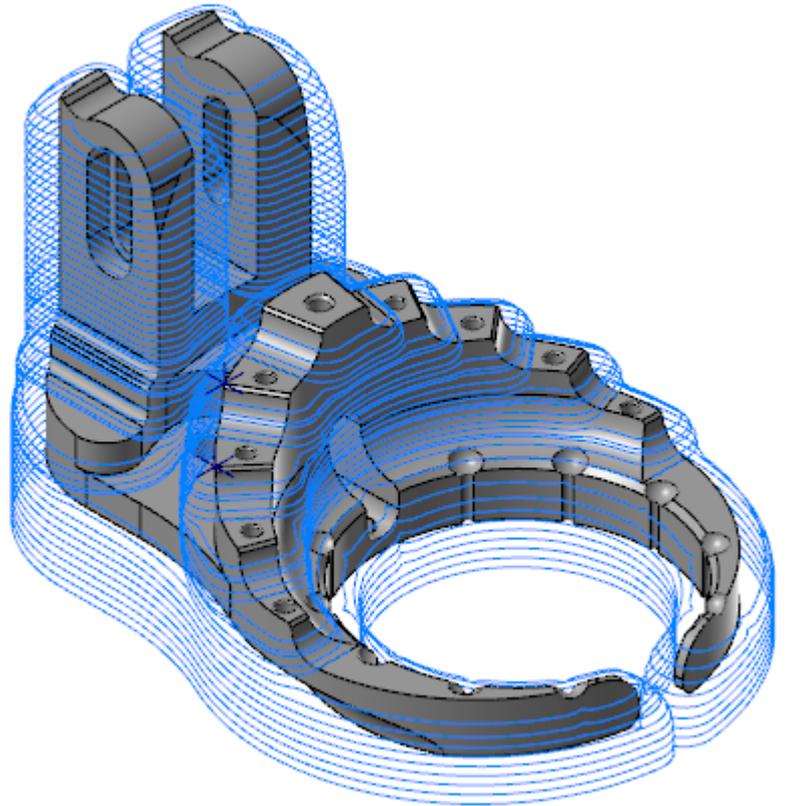
Toolpath Result

Critical Depths

Type Include flats ▾

Manual (0)  

Extend 0.0

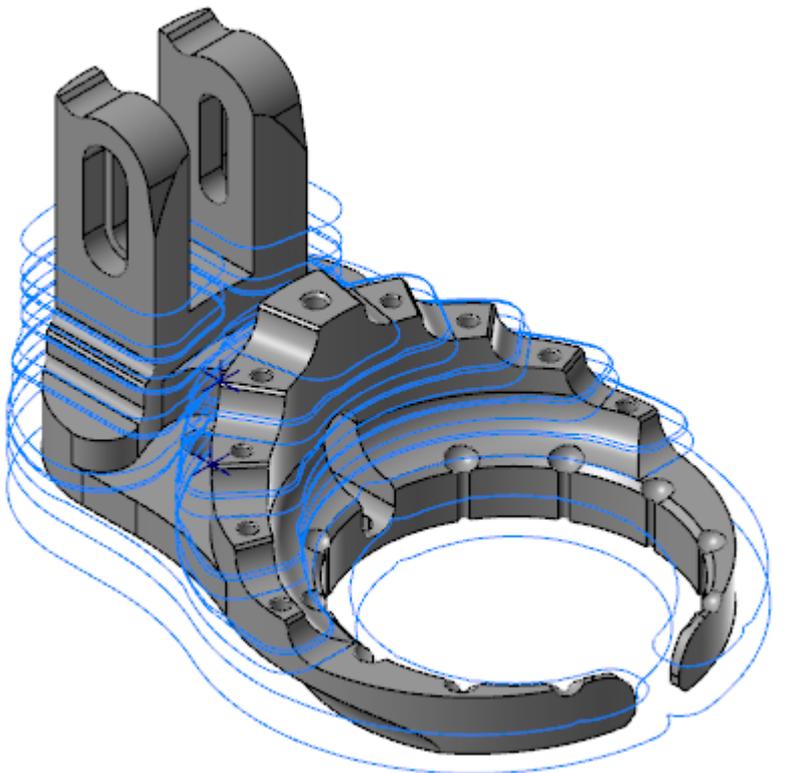


Critical Depths

Type Flats only ▾

Manual (0)  

Extend 0.0

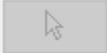


Critical Depths options

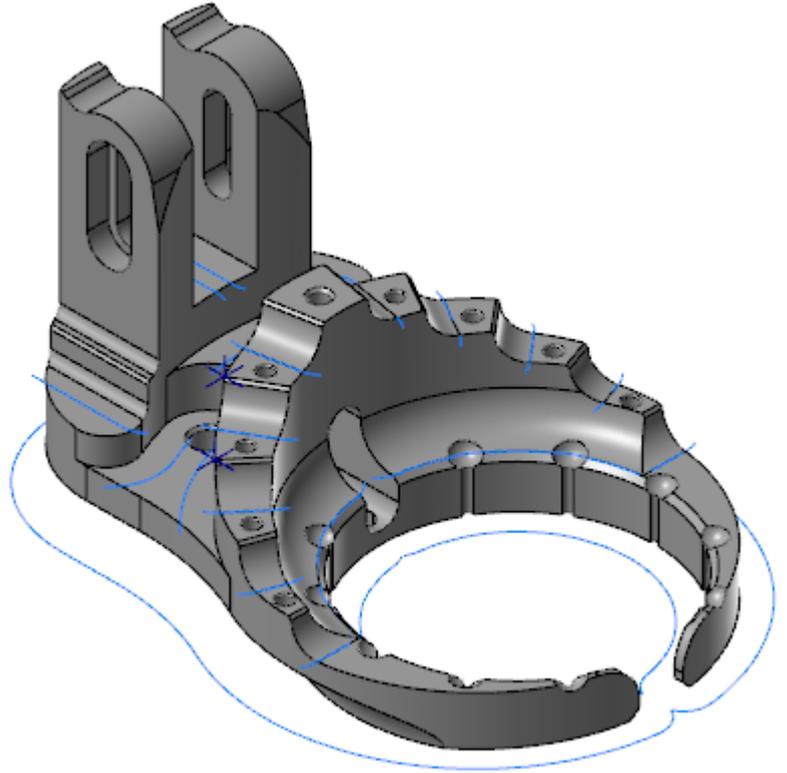
Toolpath Result

Critical Depths

Type Contact flats only ▾

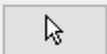
Manual (0)  

Extend 5.0

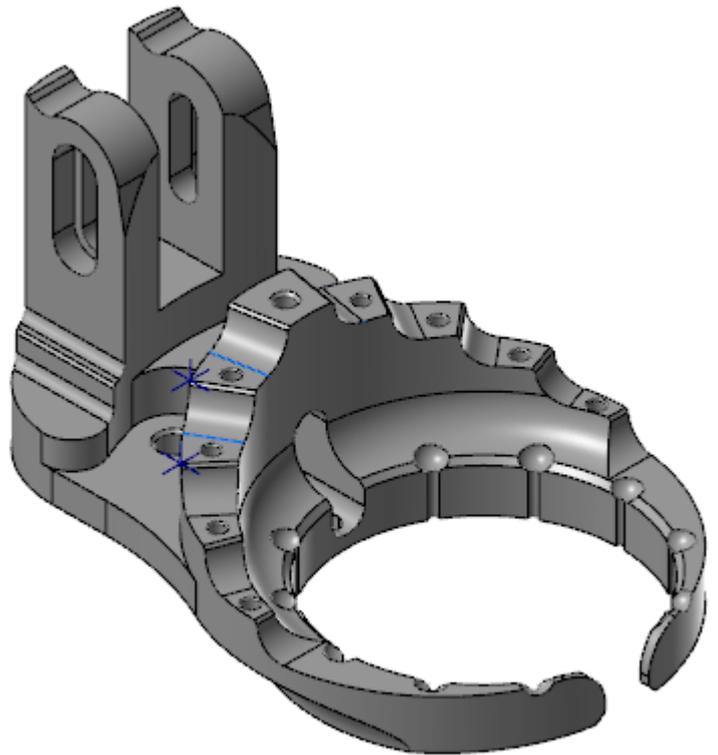


Critical Depths

Type Manual contact only ▾

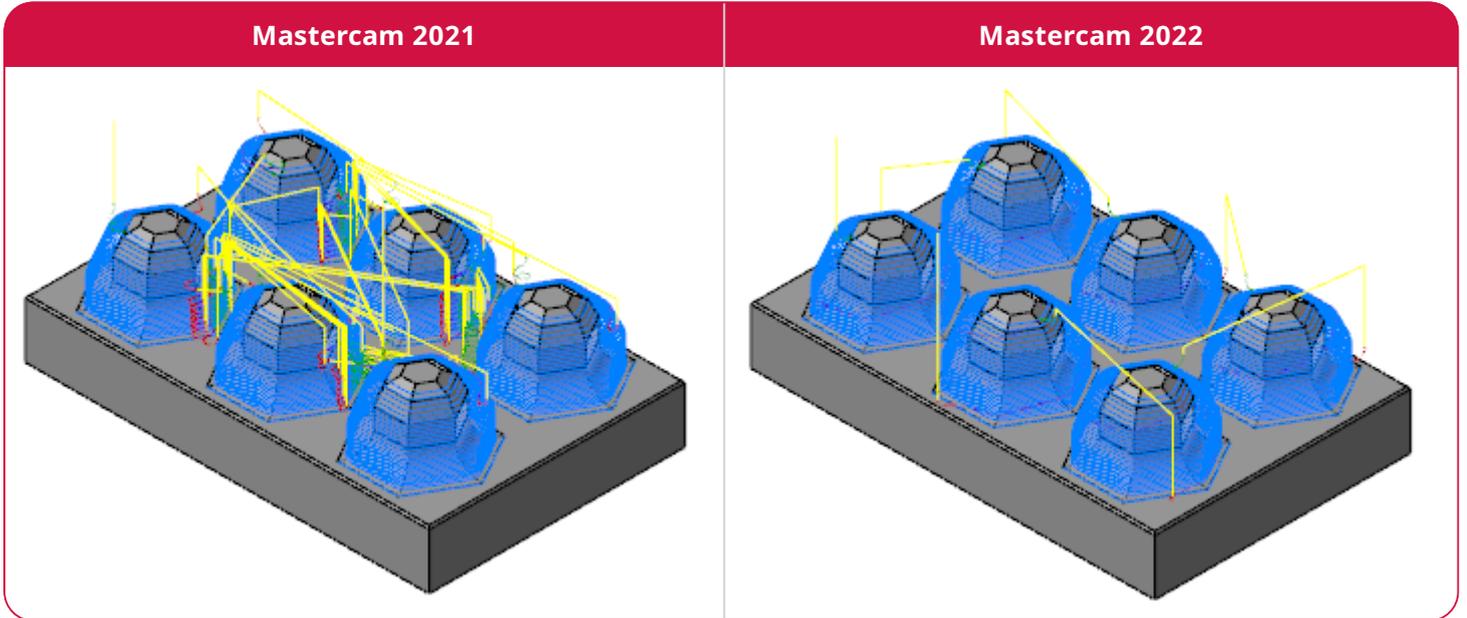
Manual (2)  

Extend 5.0



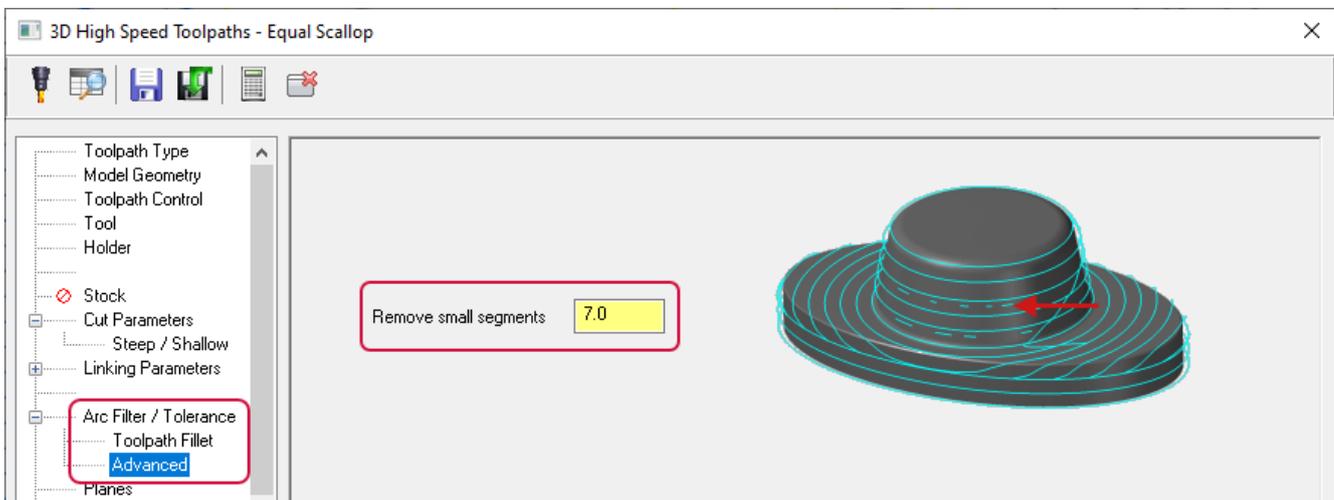
Optimizing the Cut Order

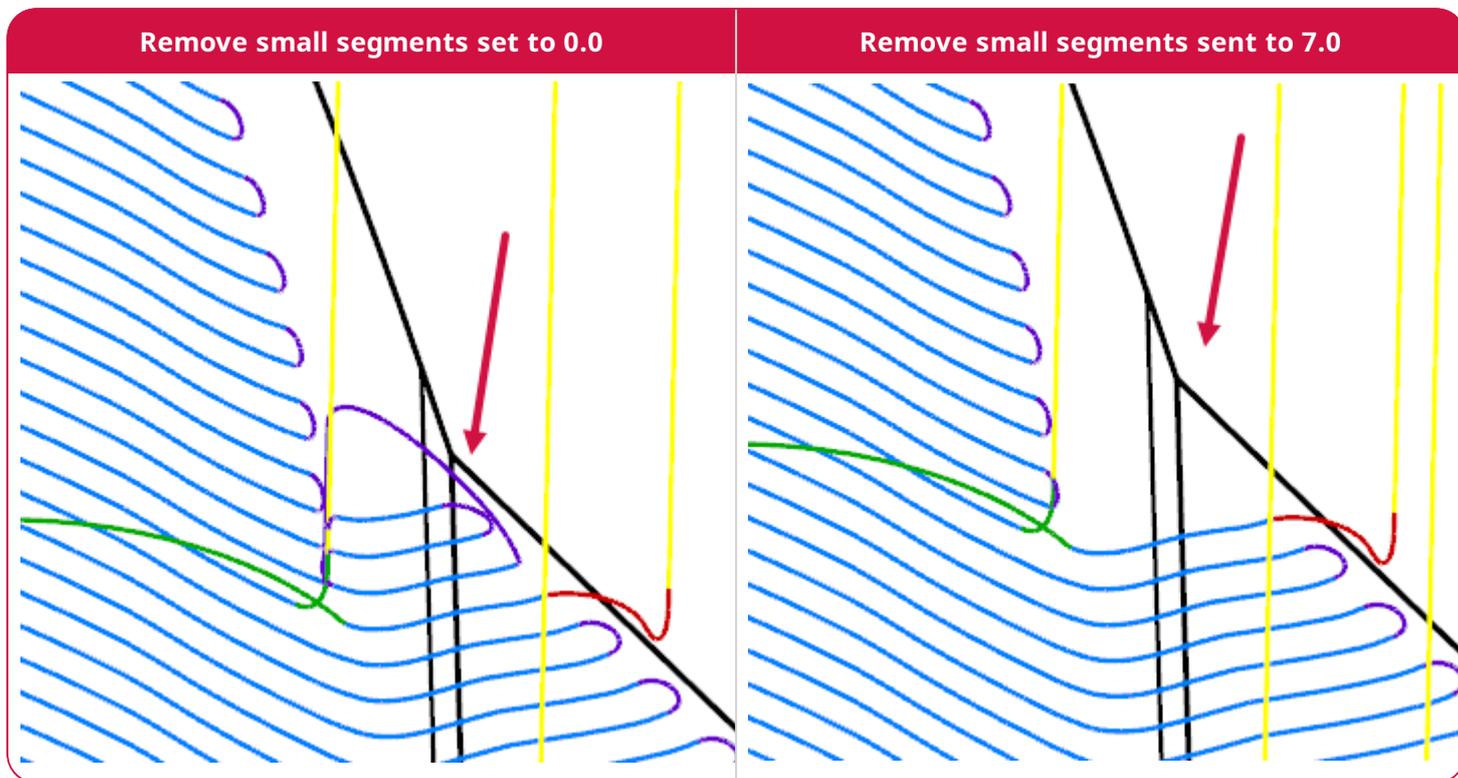
When machining bosses and features that are close to each other, Mastercam now optimizes the cut order for Waterline toolpaths.



Filtering Out Small Toolpath Segments with Equal Scallop

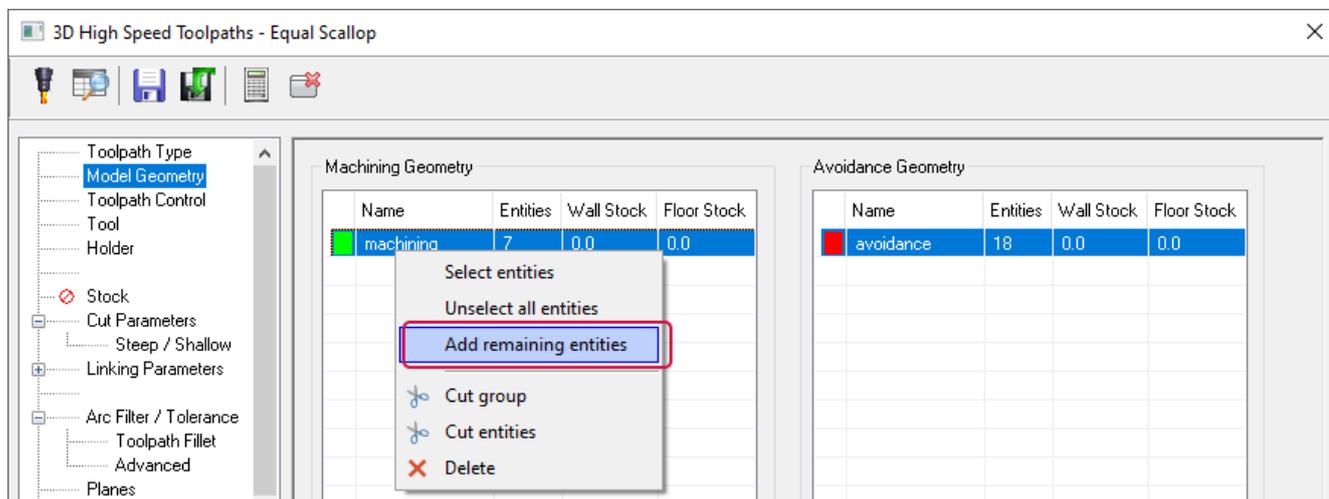
You can now filter out small toolpath segments using the **Remove small segment** parameter on the **Arc Filter/Tolerance Advanced** sub-page for Equal Scallop toolpaths. This option allows you to remove any unwanted motion that is smaller than the set value.



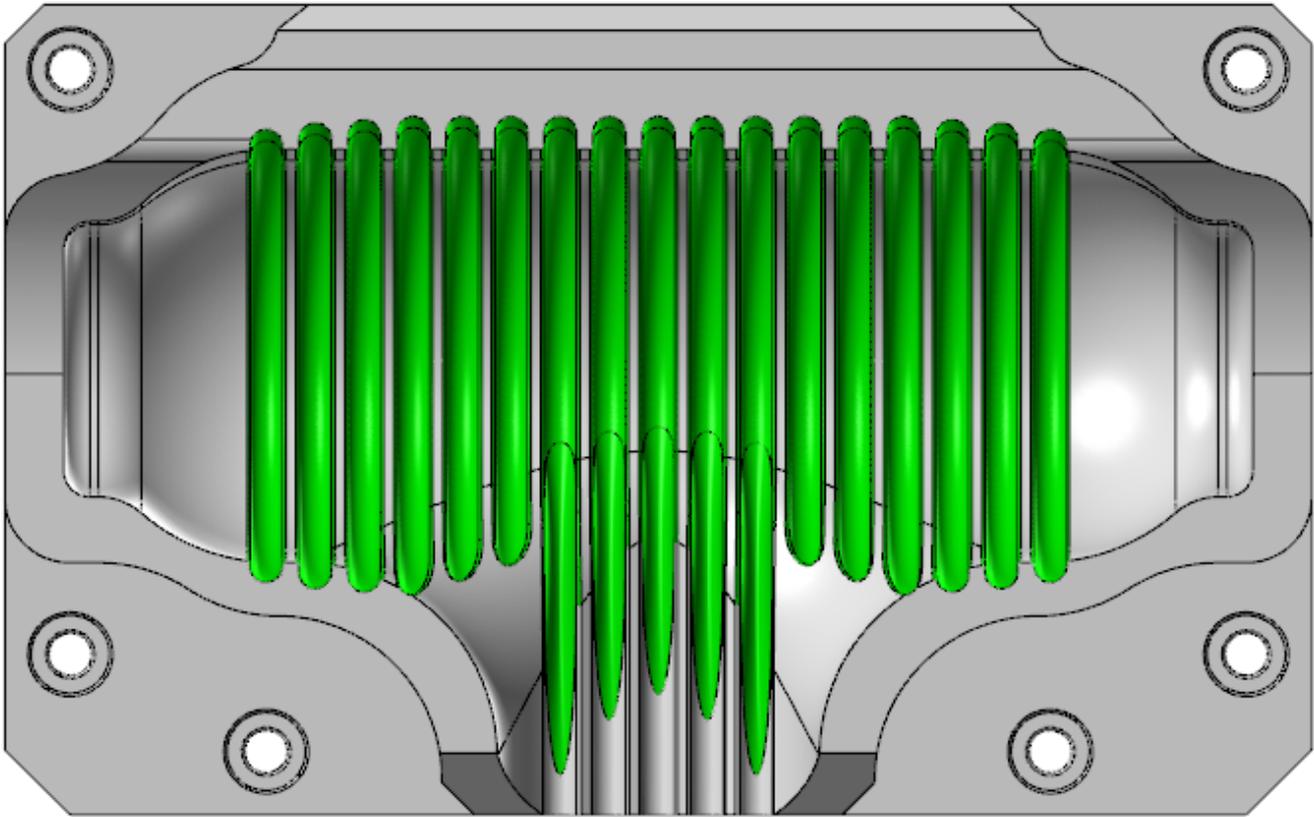


Selecting Machining, Avoidance, and Remaining Entities from the Right-click Menu

You can now right-click in the **Machining geometry** and **Avoidance geometry** tables of the **Model Geometry** page and use **Select Entities** to select the machining or avoidance geometry. You can also use the **Select Entities** button at the bottom of the page.



You can also select **Add remaining entities** to add any unselected entities to that group. This helps you save time, by automatically adding these entities instead of manually selecting them in the graphics window. The part below demonstrates the utility of this feature. It shows how to use **Add remaining entities** to select the individual slots rather than manually selecting them.



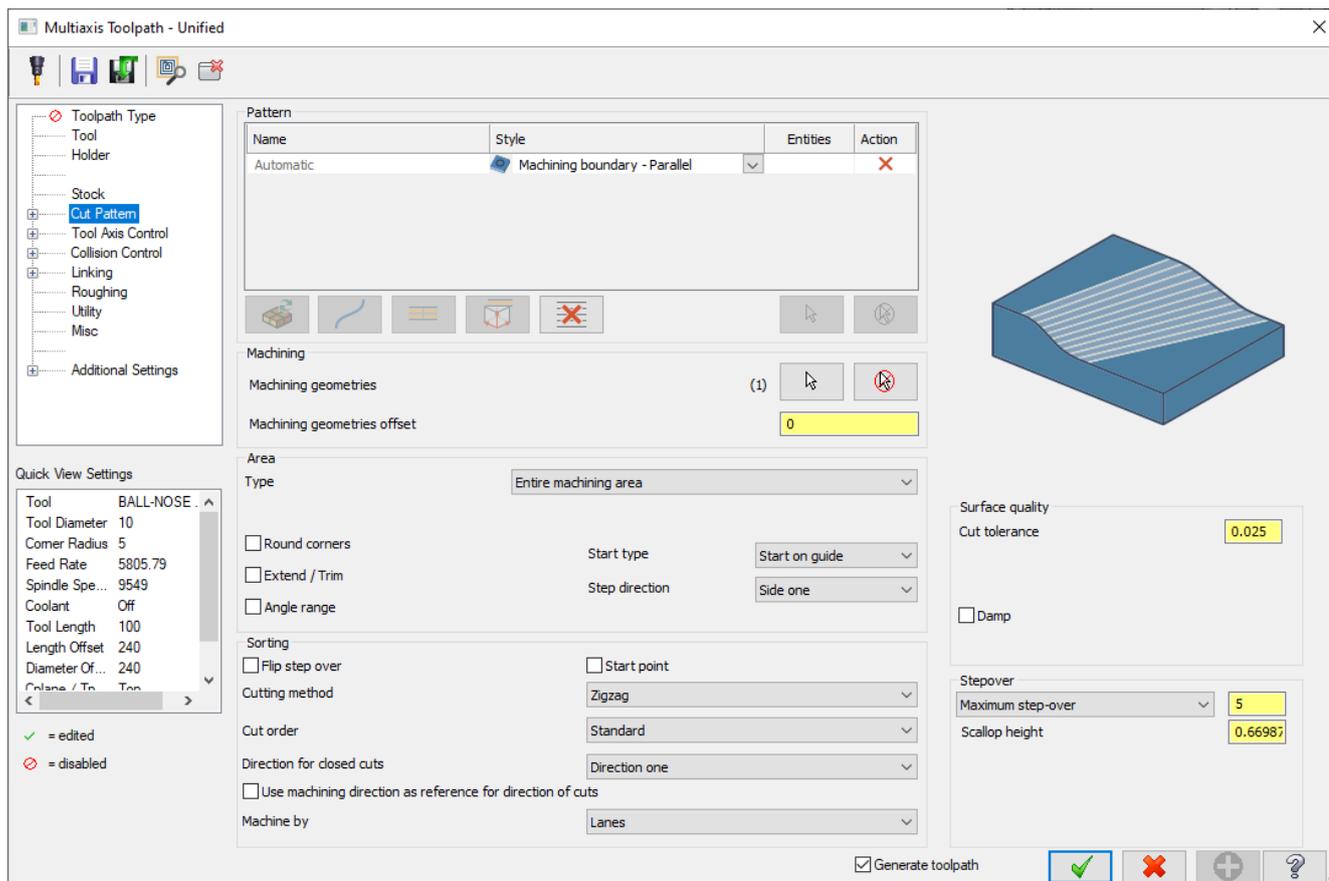
Multiaxis Enhancements

Listed below are enhancements to Multiaxis toolpaths.

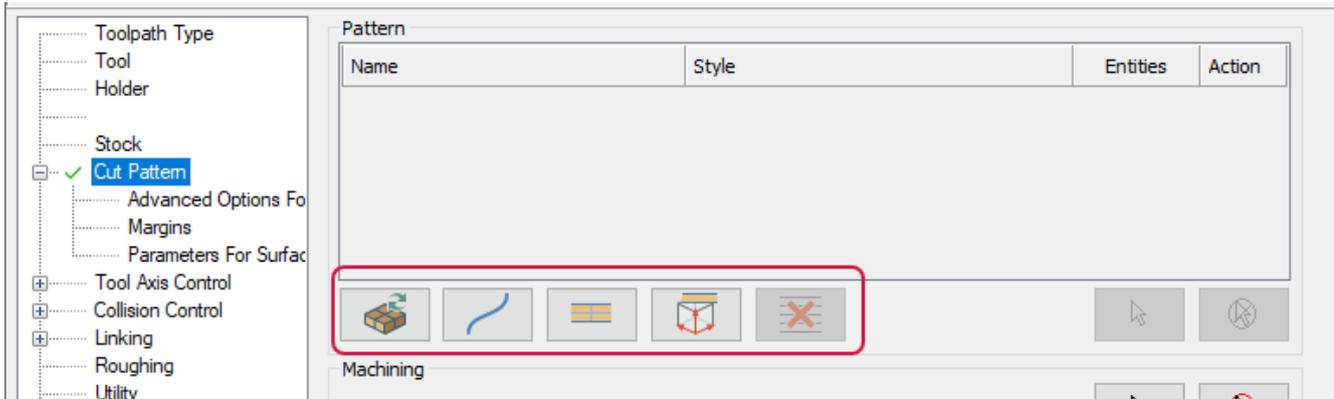
New Multiaxis Unified Toolpath

The new Multiaxis Unified toolpath allows you to select multiple pieces of input geometry to generate the toolpath pattern. Then, using those geometry choices, the toolpath picks the best algorithm to calculate the path. An example is using chains to influence the cut pattern, or two surfaces between which to morph.

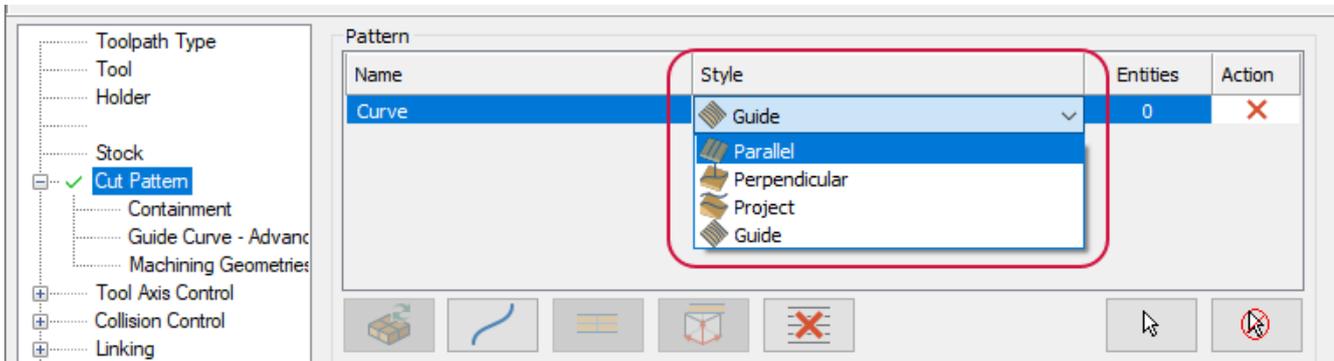
This single toolpath gives you access to all the surface-based toolpaths (Morph, Parallel, Along Curve, and Project), as well as the geodesic toolpath options (Automatic, Guides), in a single interface.



You can add or remove patterns, such as switching from a parallel to a chain pattern to a morph between two surfaces without starting the toolpath over. Choose from four drive patterns (shown below): **Automatic**, **curves**, **surfaces**, **plane**. The curve, surface, and plane patterns are similar to options in Parallel or Morph toolpaths, such as **Parallel to curve**, **Morph between curves**, **Parallel to surface**, **Morph between surfaces**, or **Parallel to a plane**.



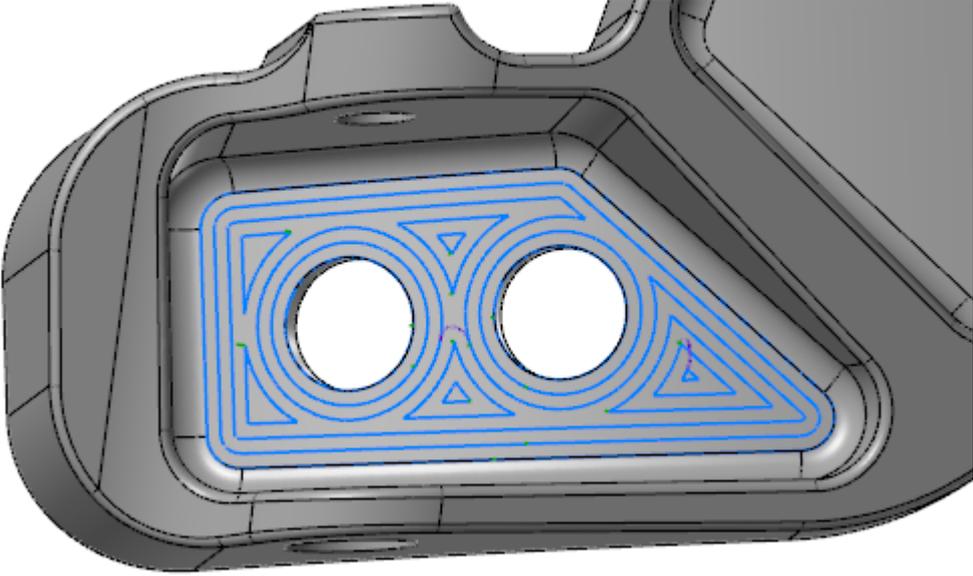
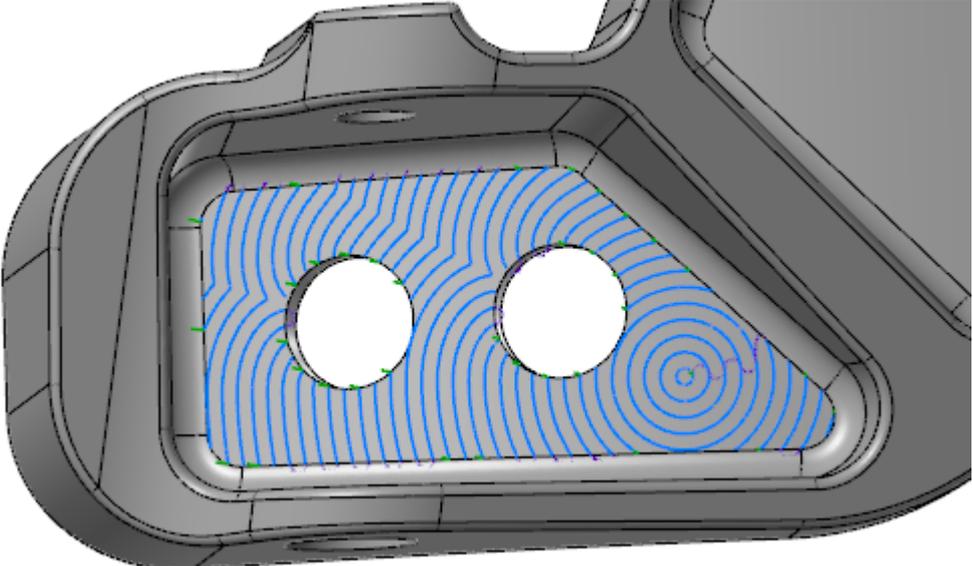
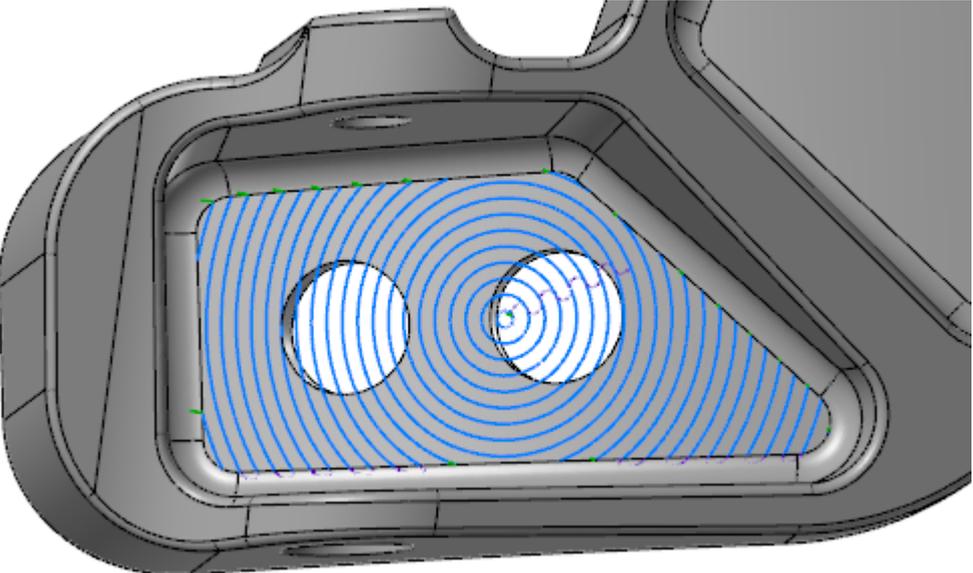
When you choose a pattern type, an entry appears in the **Pattern** table. You can have multiple entries of a pattern in the table, as shown below. Note that the **Style** column is a drop-down containing options to use with the pattern.



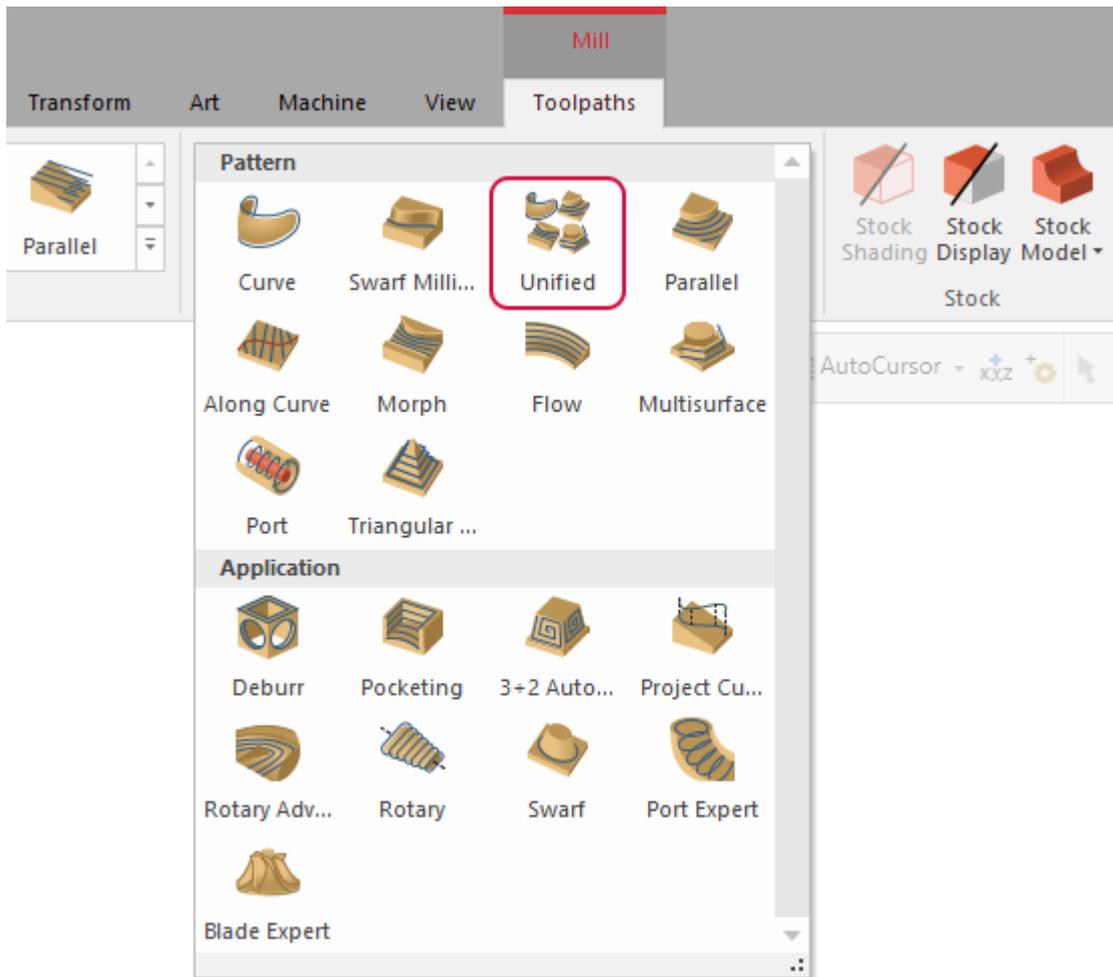
The Multiaxis Unified toolpath uses the new geodesic algorithm when the **Pattern** is set to **Automatic** or when **Curves** and **Surfaces** are set to **Guide**. **Geodesic** is defined as the shortest line between two points on a curved surface. Mastercam's geodesic algorithm eliminates the distortions you may get when generating a toolpath over such a surface.

The geodesic **Automatic** pattern takes the extents of the geometry you have selected and tries to find its boundary. It then creates, based on the **Style** you choose, a toolpath that is either parallel to the boundary or morphs between a central shape and the boundary.

Below are a few examples of the Unified Multiaxis toolpath.

Parameters	Toolpath Result
<p>Pattern set to Automatic.</p> <p>Style set to Machining boundary - Parallel.</p>	
<p>Pattern set to Automatic.</p> <p>Style set to Center - Parallel.</p>	
<p>Pattern set to Automatic.</p> <p>Style set to Center - Parallel.</p> <p>Fill holes selected on the Machining Geometries - Advanced Parameters page.</p>	

Access the Multiaxis Unified toolpath from the **Multiaxis** gallery on the **Mill Toolpaths** contextual tab.

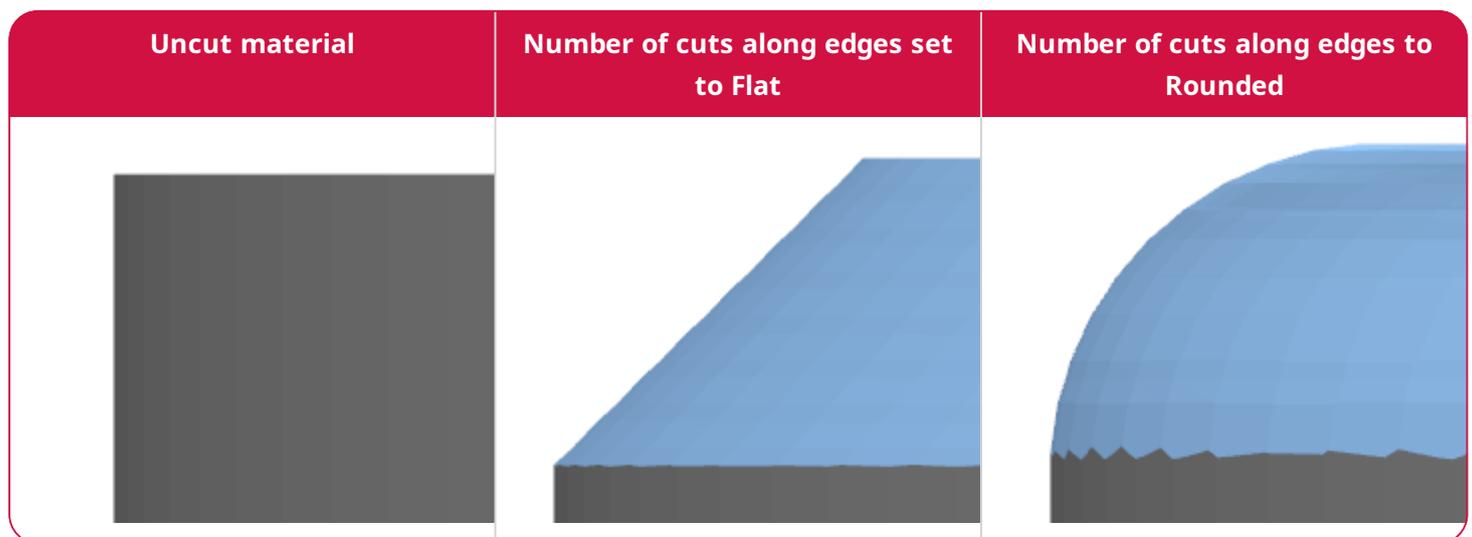
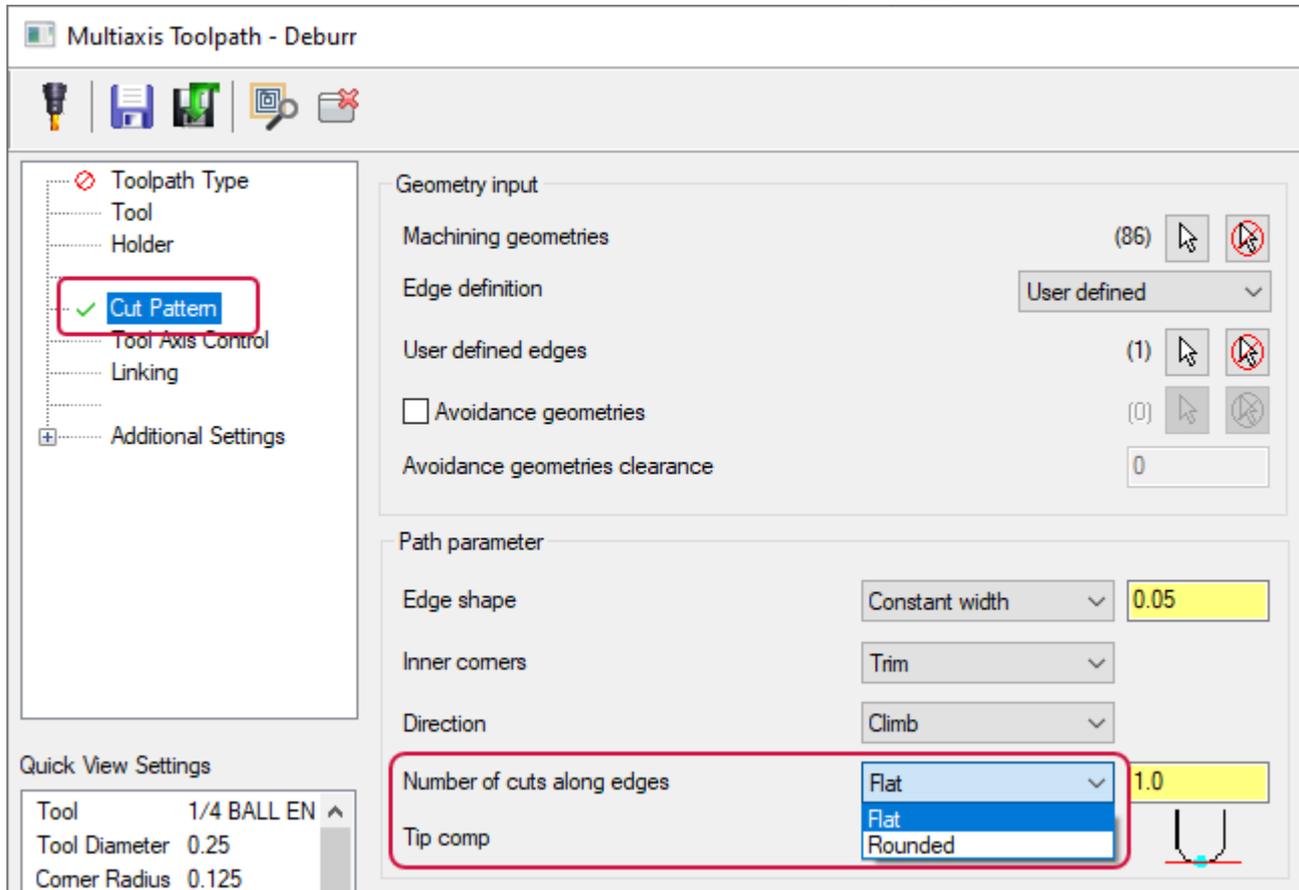


Deburr Toolpath Enhancements

Listed below are enhancements to the Deburr toolpath.

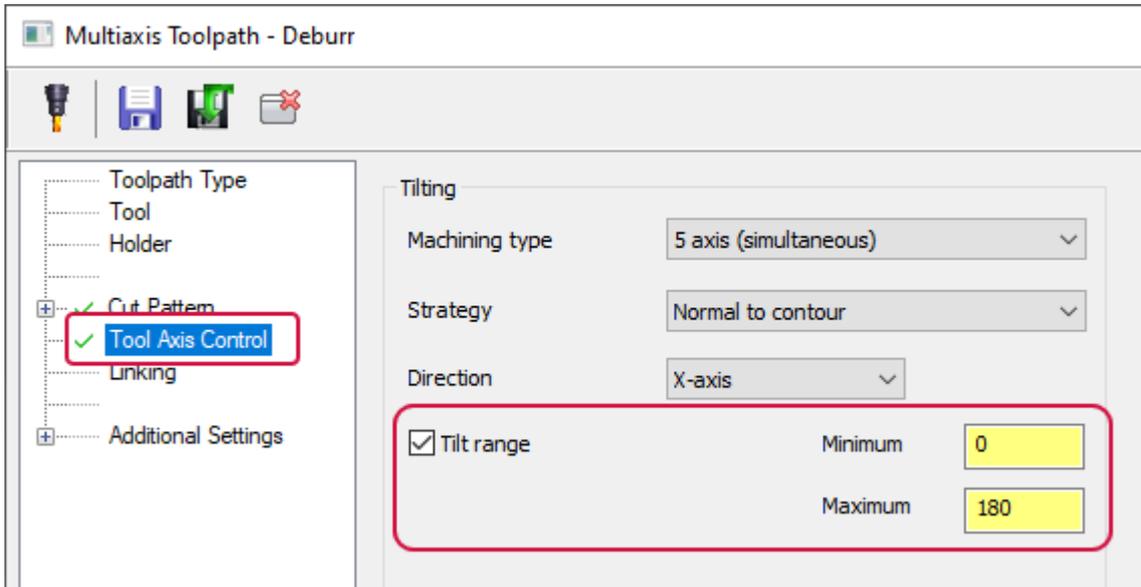
Performing Multiple Cuts

A new option, **Number of cuts along edges**, allows you to add multiple cuts to the Deburr toolpath, rounding or flattening edges, to create a quality edge on deburred parts.



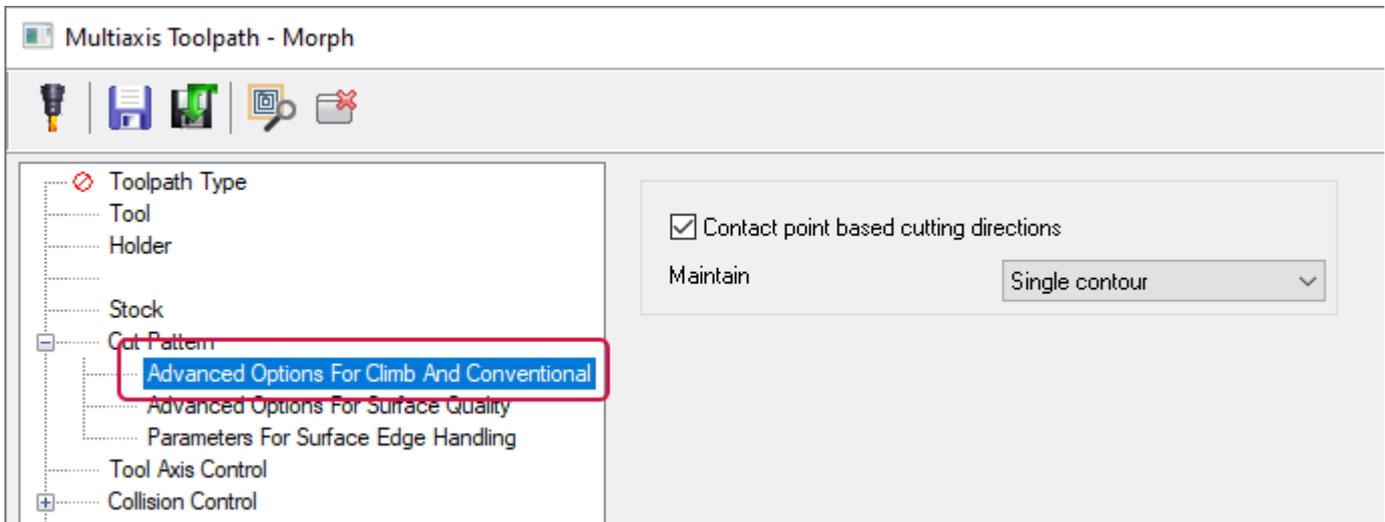
Limiting Tilt Motion

The new **Tilt range** option allows you to limit tilt when **Machining type** is set to **5 axis (simultaneous)** or **3+2 axis** on the **Tool Axis Control** page. Enter minimum and maximum values to control the tilt range of the tool.

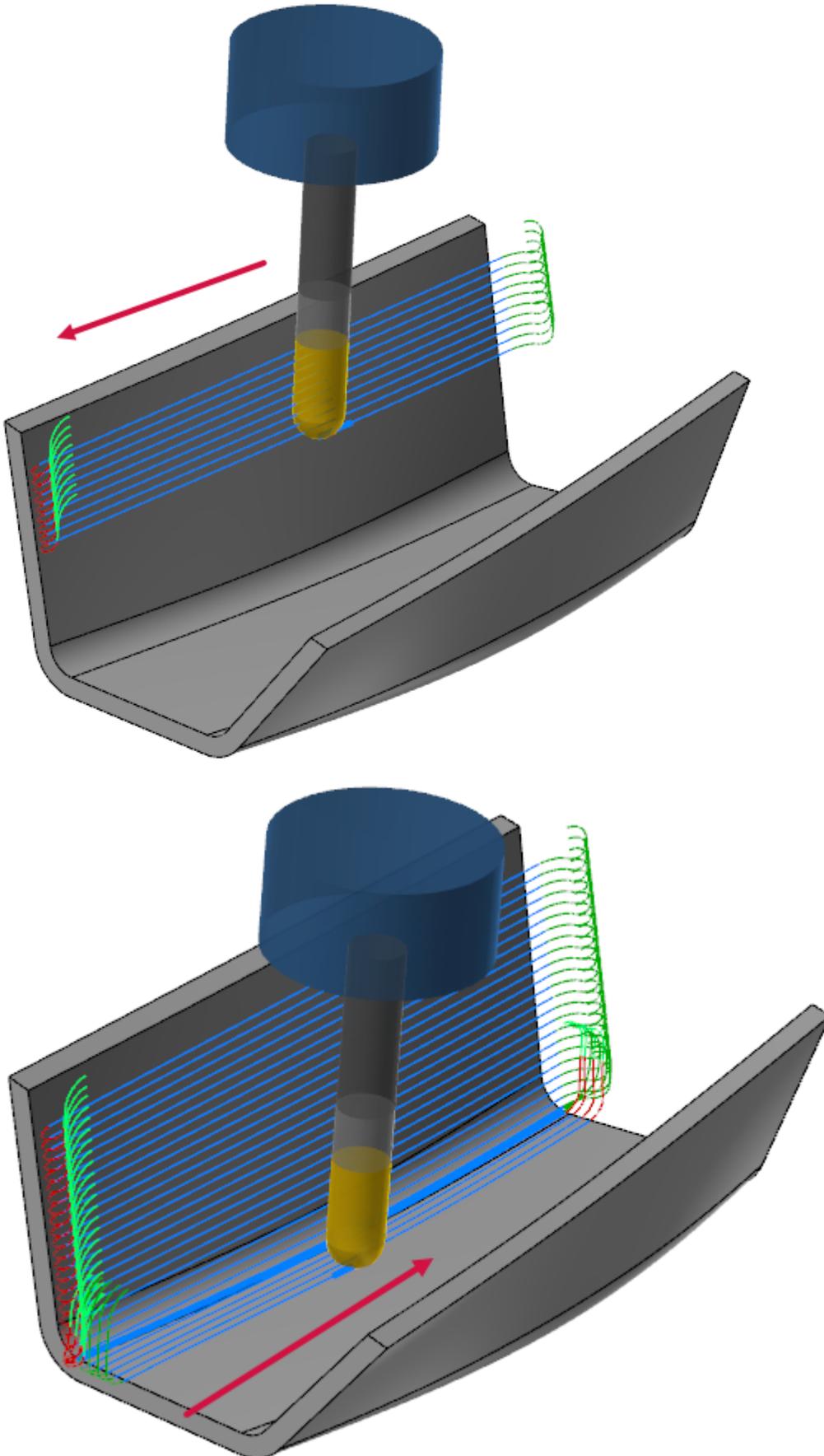


Controlling Climb and Conventional Cutting

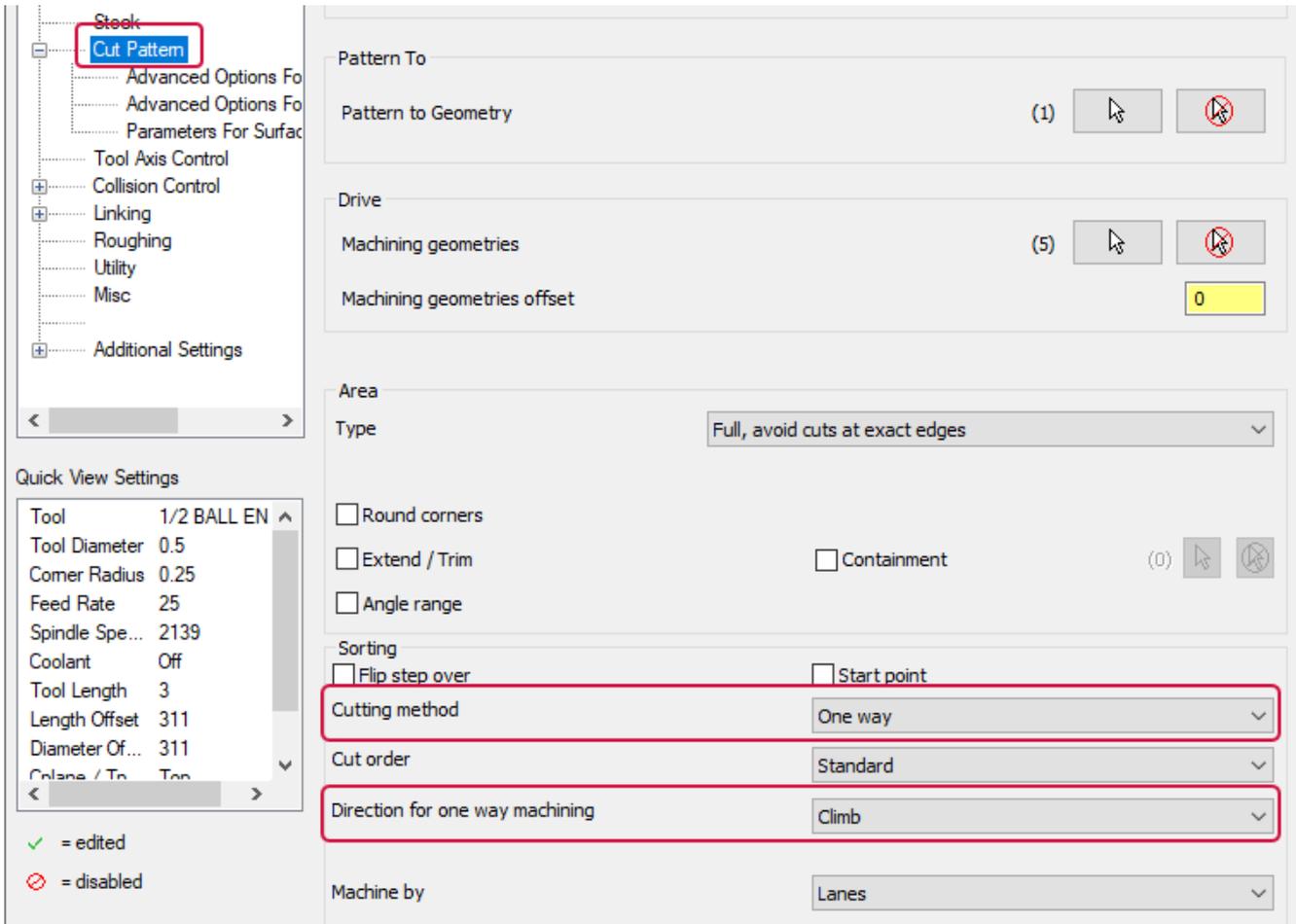
Settings on the **Advanced Options for Climb and Conventional** page for the Morph, Parallel, Unified, Project Curve, and Flow toolpaths now provide control of climb or conventional cutting while working with surfaces that may require using both sides of a tool.



Maintaining climb and conventional machining is critical for surface finish and tool life. It was previously impossible to maintain this while cutting a curved shape—for example, a U-shaped surface that would require the use of both sides of the tool.



To display the **Advanced Options for Climb and Conventional** page, set the **Cutting method** to **One way** and **Direction for one way machining** to **Climb** or **Conventional** on the **Cut Pattern** page for the applicable toolpaths.

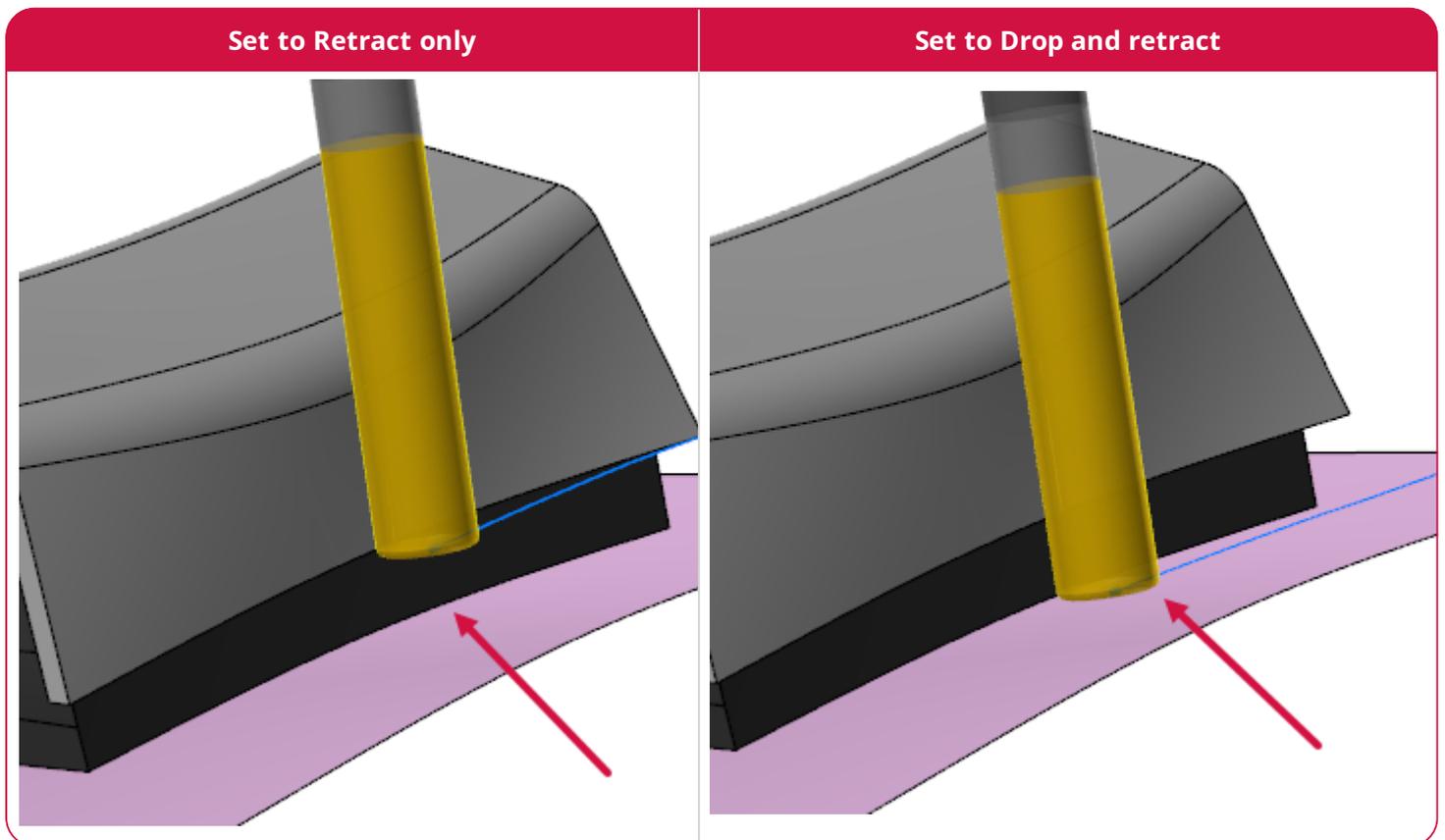
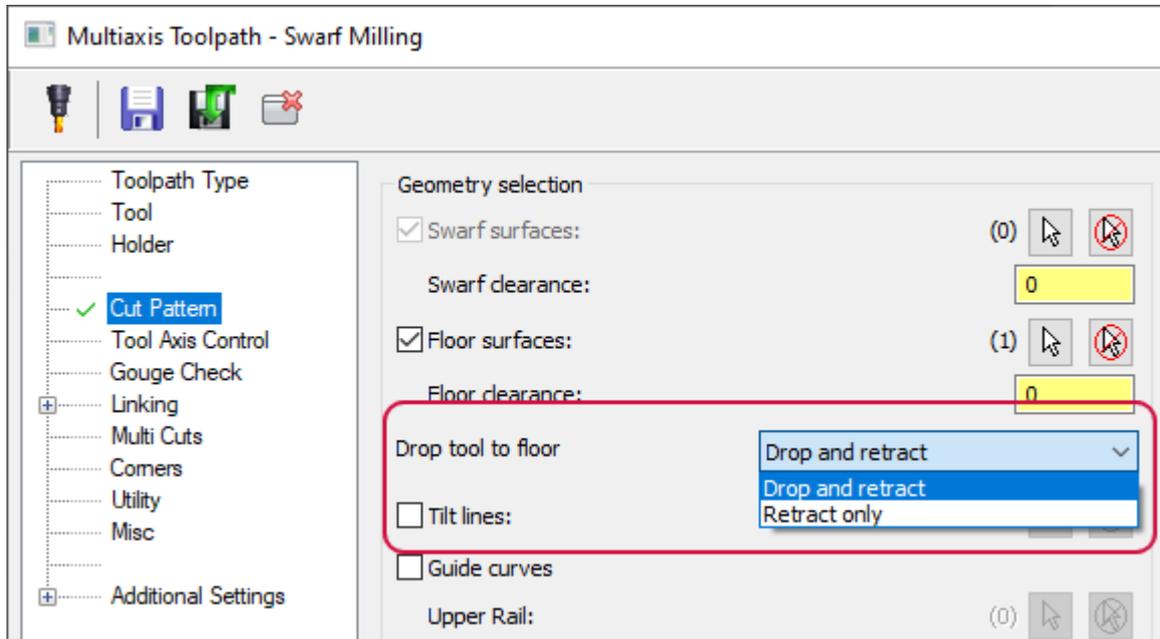


Swarf Milling Toolpaths Enhancements

Listed below are enhancements to the Swarf Milling toolpath.

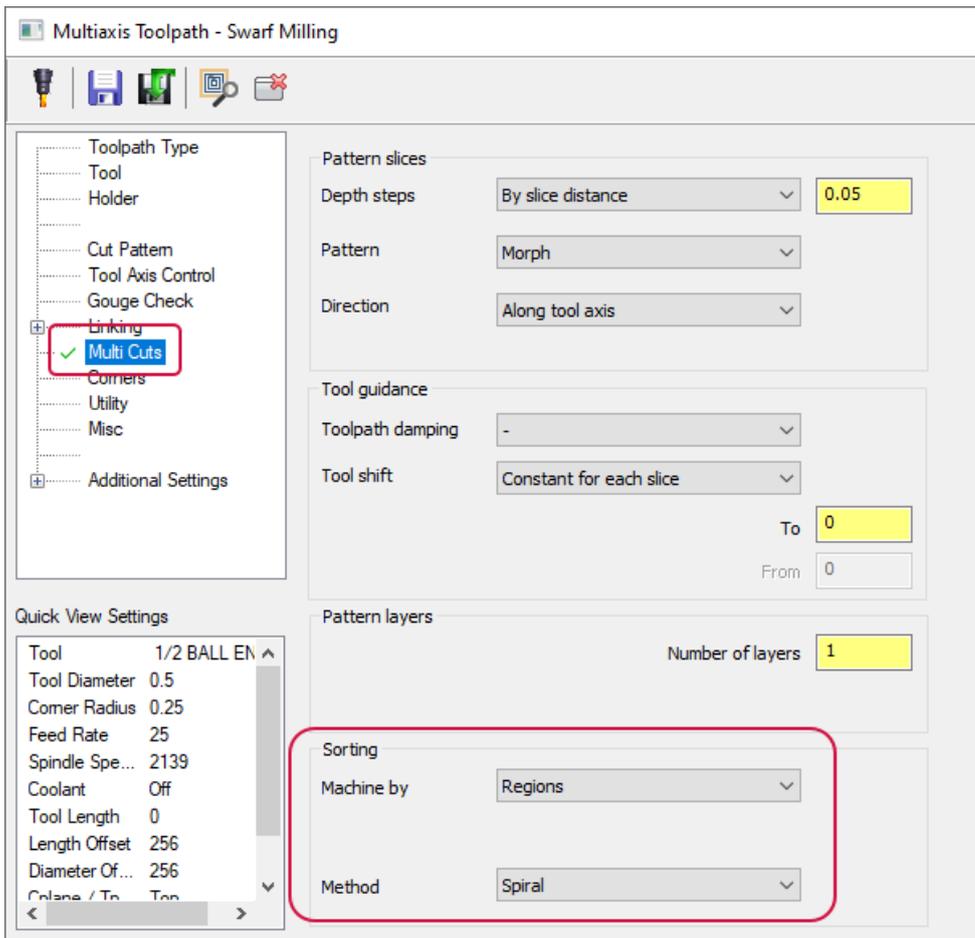
Dropping the Tool to the Floor

The **Drop tool to floor** option has been added to the **Cut Pattern** page. This new option controls whether Mastercam will drop the toolpath down to a floor or retract to the bottom of swarf surfaces.

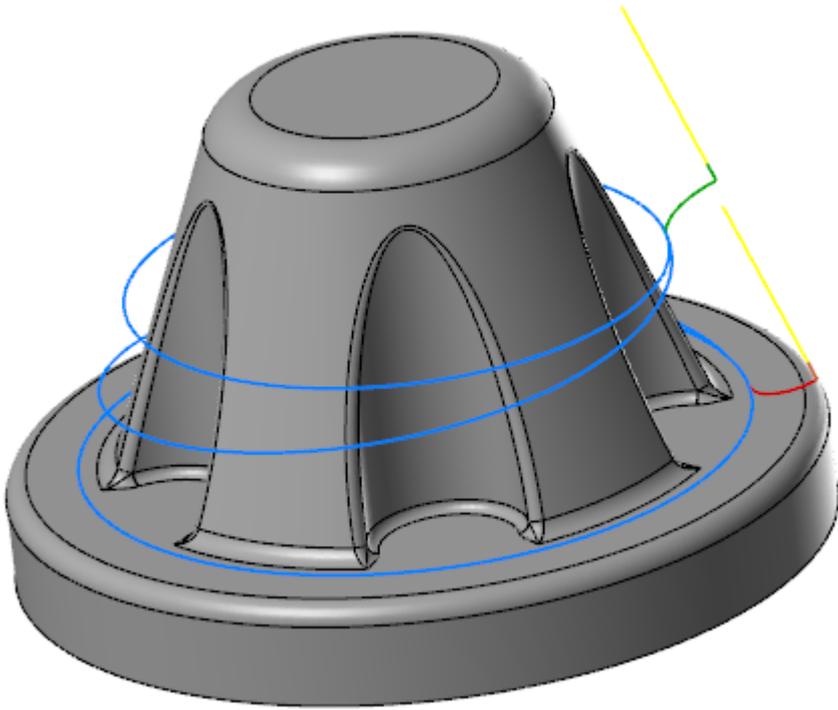


Sorting Spiral Multicuts

Spiral has been added to the sorting methods on the **Multi Cuts** page.

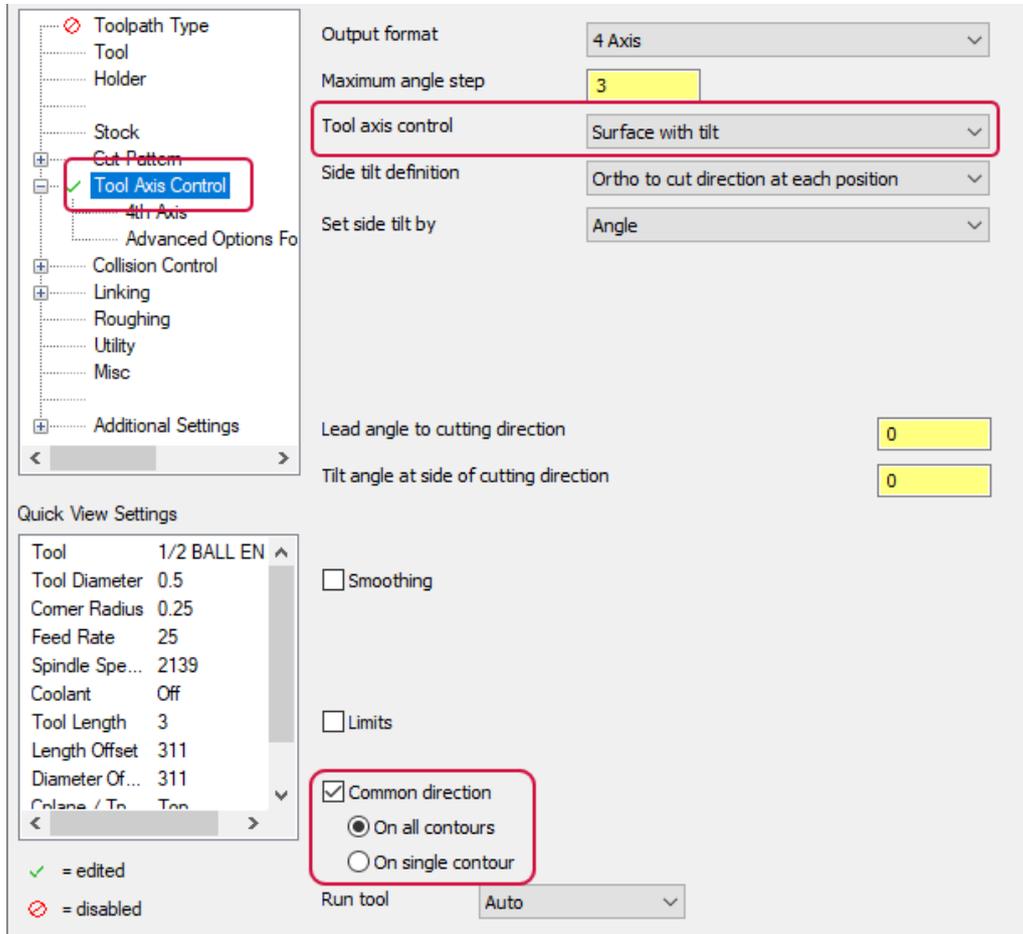


This new option allows multiple slices to be output as a spiral instead of single slices, providing a better finish across the part.



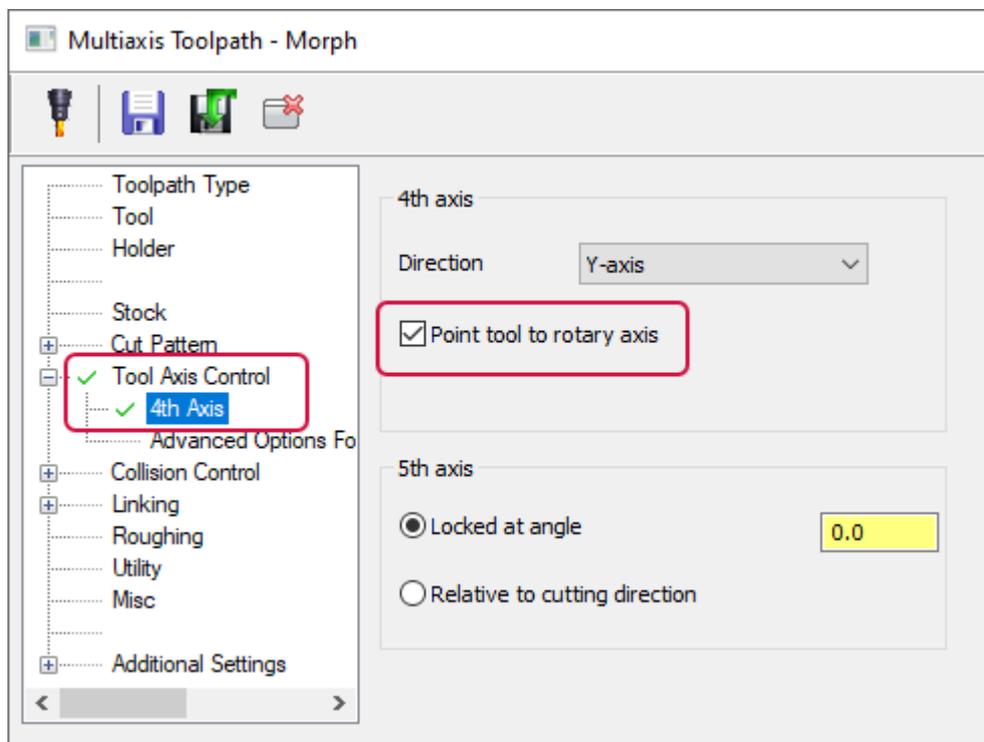
Maintaining a Single Tool Axis Orientation with Accelerated Finishing Tools

The **Common direction** parameter, which was previously only available when using a ball mill or end mill, is now available for Accelerated Finishing tools. **Common direction** finds and maintains a single tool axis orientation for the toolpath on all contours or a single contour, providing optimal tool axis control that minimizes or eliminates extra multi-axis movement. This option is found on the **Tool Axis Control** page for the Morph, Parallel, and Unified toolpaths when **Tool axis control** is set to **Surface with tilt**.



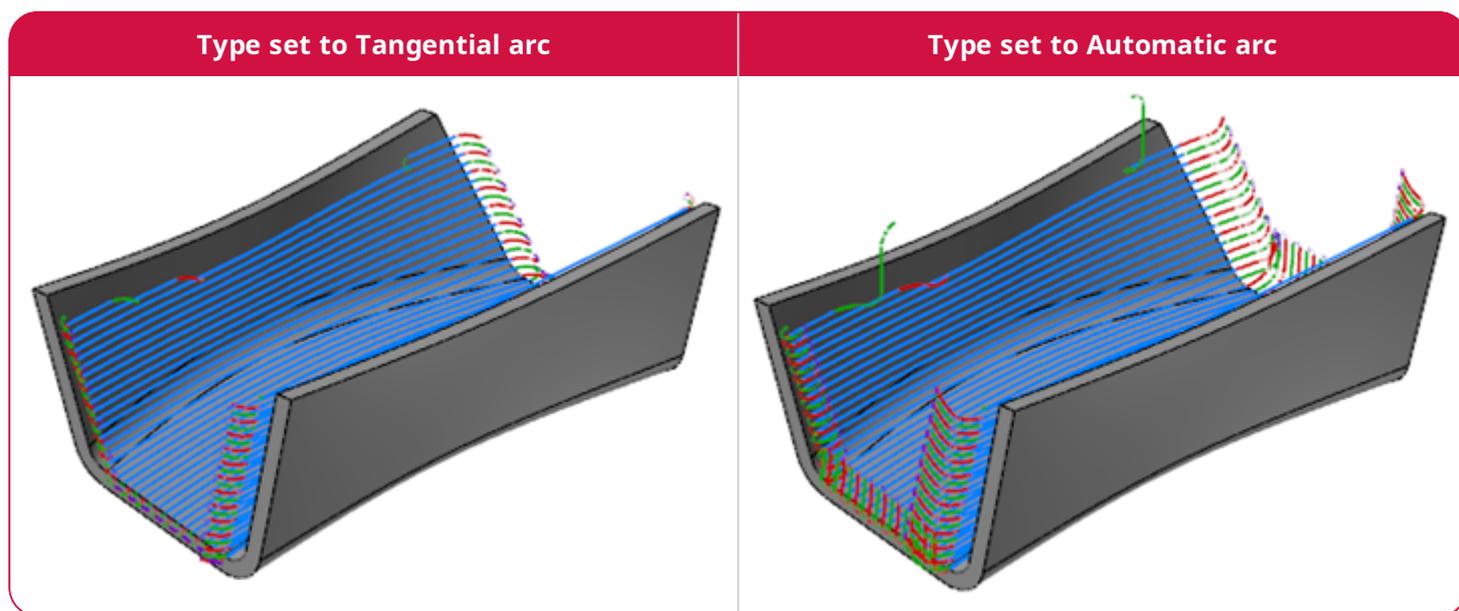
Rotating a Toolpath Around an Offset Point

The new **Point tool to rotary axis** option allows you to rotate a toolpath around an offset point instead of the toolpath being fixed to the axis itself. This leads to a better tool contact point. This new option is found on the **4th Axis** page, when **Output format** is set to **4 Axis** on the **Tool Axis Control** page of the Morph, Parallel, Unified, Along Curves, and Project Curves toolpaths.

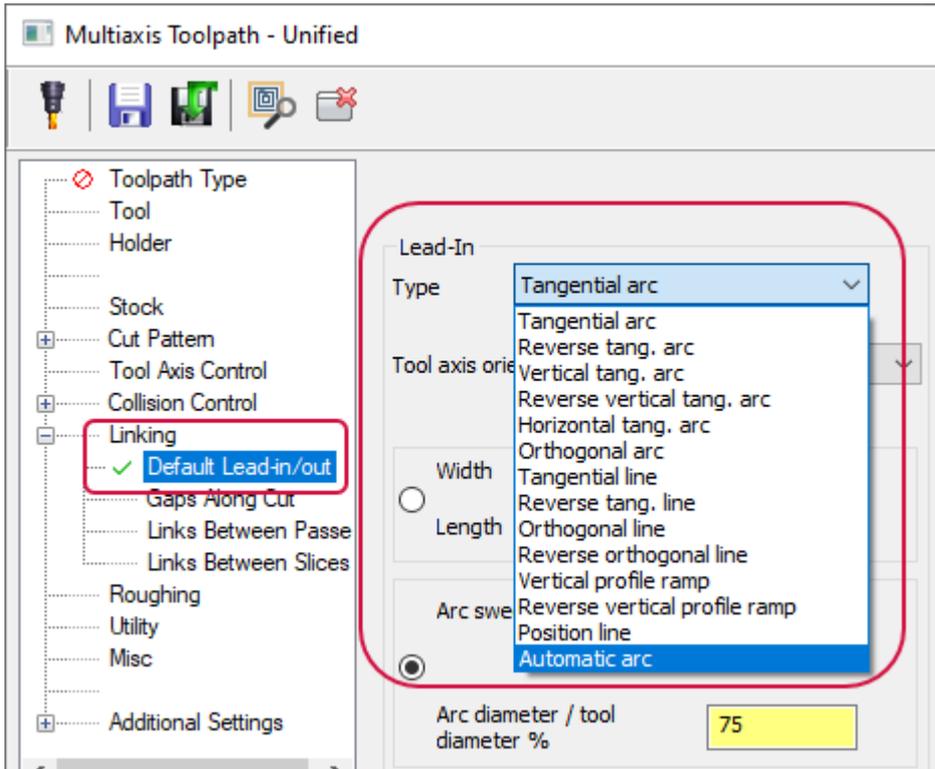


Automatic Arcs and Automatic Tool Axis Orientation for Lead In/Out Moves

The new **Automatic arc** lead-in type creates automatic arcs in locations where it can be difficult to determine the correct arc to fit. This option also makes it easier to add arcs to linking moves. When **Automatic arc** is selected, Mastercam chooses a **Type** that is best suited for your part.

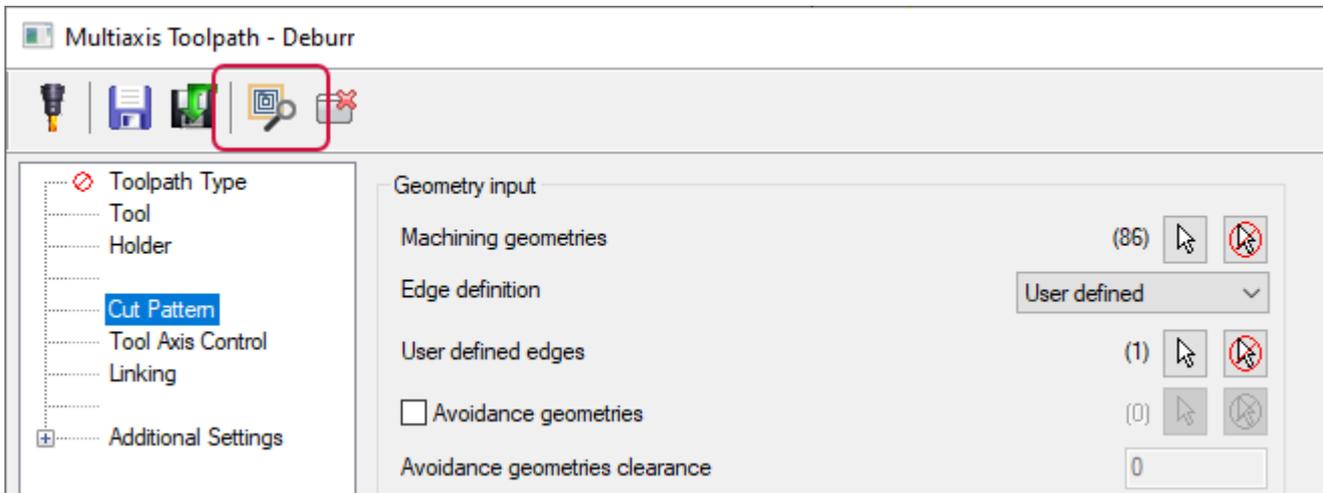


Automatic arc is available in the **Type** drop-down on the **Default Lead-in/out** page, under the **Linking** page for Morph, Parallel, Unified, Along Curves, and Project Curves toolpaths.



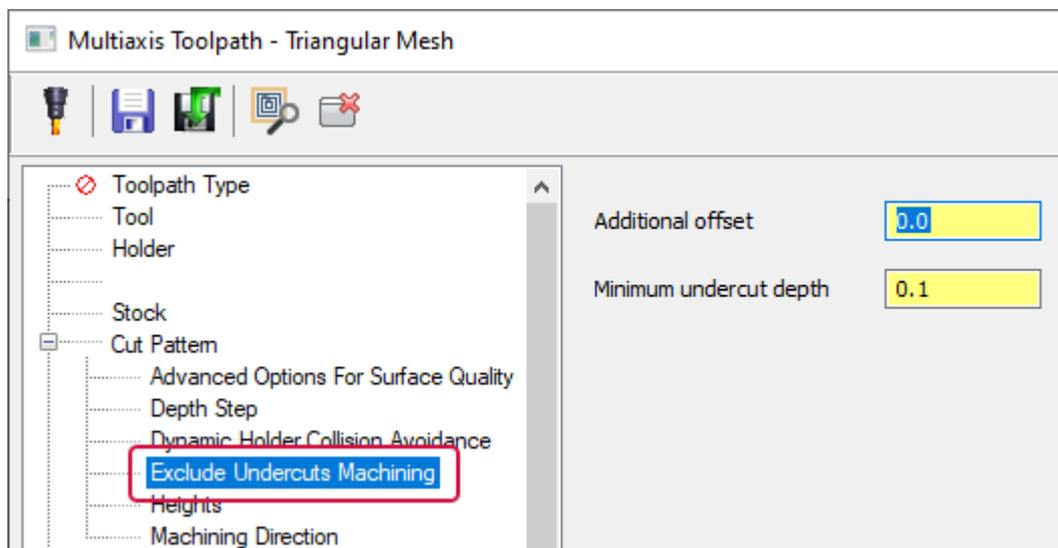
Previewing Multiaxis Toolpaths

You can now preview Multiaxis toolpaths using **Preview toolpath** located in the parameters dialog boxes. With this function, you can see the effects of toolpath settings without leaving the dialog box.



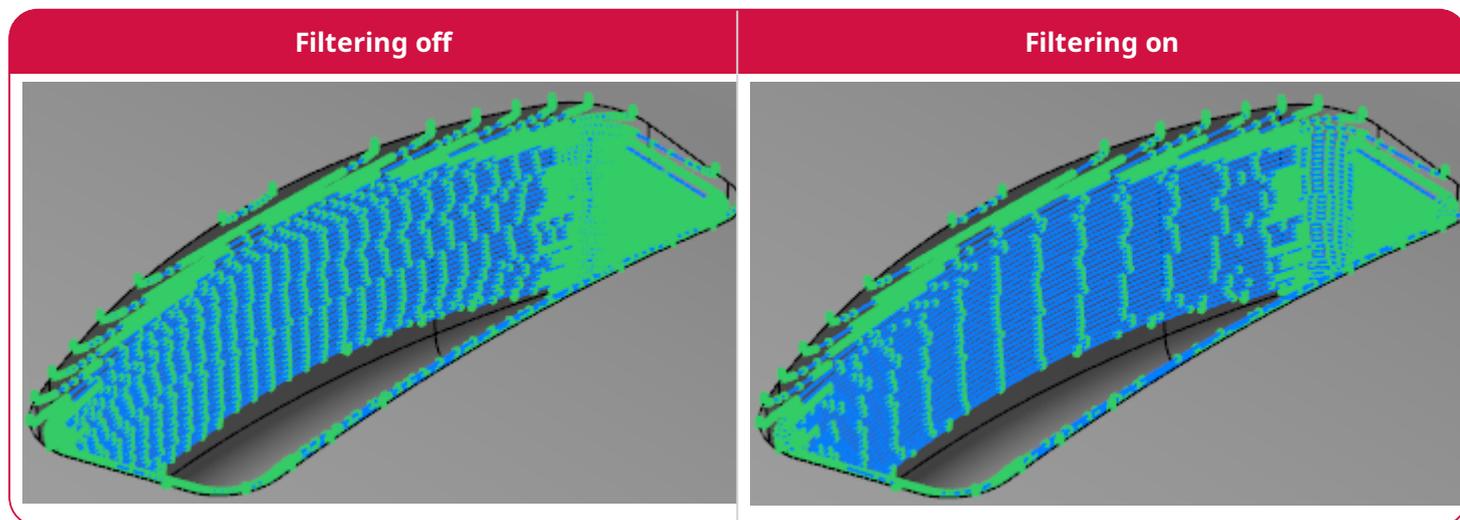
Expanded Options for Excluding Undercuts for Triangular Mesh

When you select **Exclude undercut areas** on the **Cut Pattern** page for Triangular Mesh toolpaths, Mastercam displays the new **Exclude Undercuts Machining** page in the tree list. The options on this page let you further refine the function's parameters.

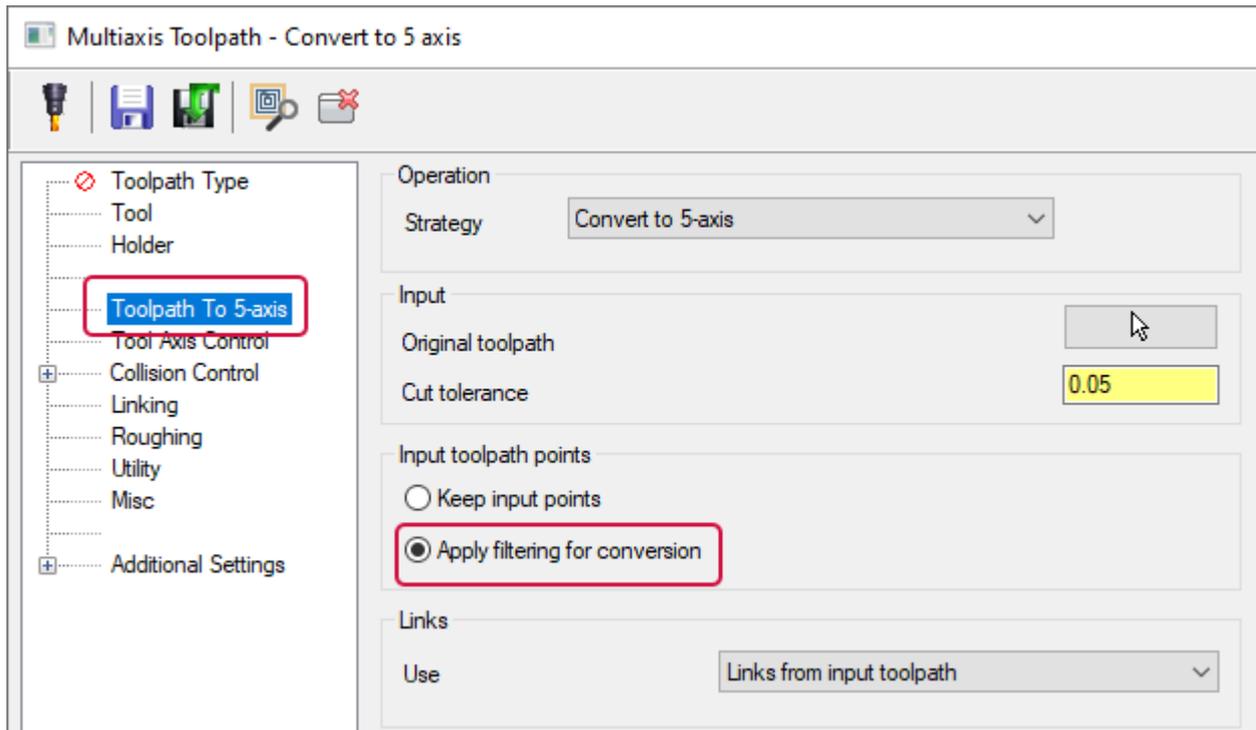


Filtering the Convert to 5-Axis Toolpaths for Size and Noise

The new **Apply filtering for conversion** option generates a more efficient toolpath by filtering incoming toolpaths to reduce toolpath size and noise. The images below have **Advanced Display, Endpoints** selected to better show the new functionality.



The **Apply filtering for conversion** option is available on the **Toolpath To 5-axis** page for Convert to 5-axis toolpaths.

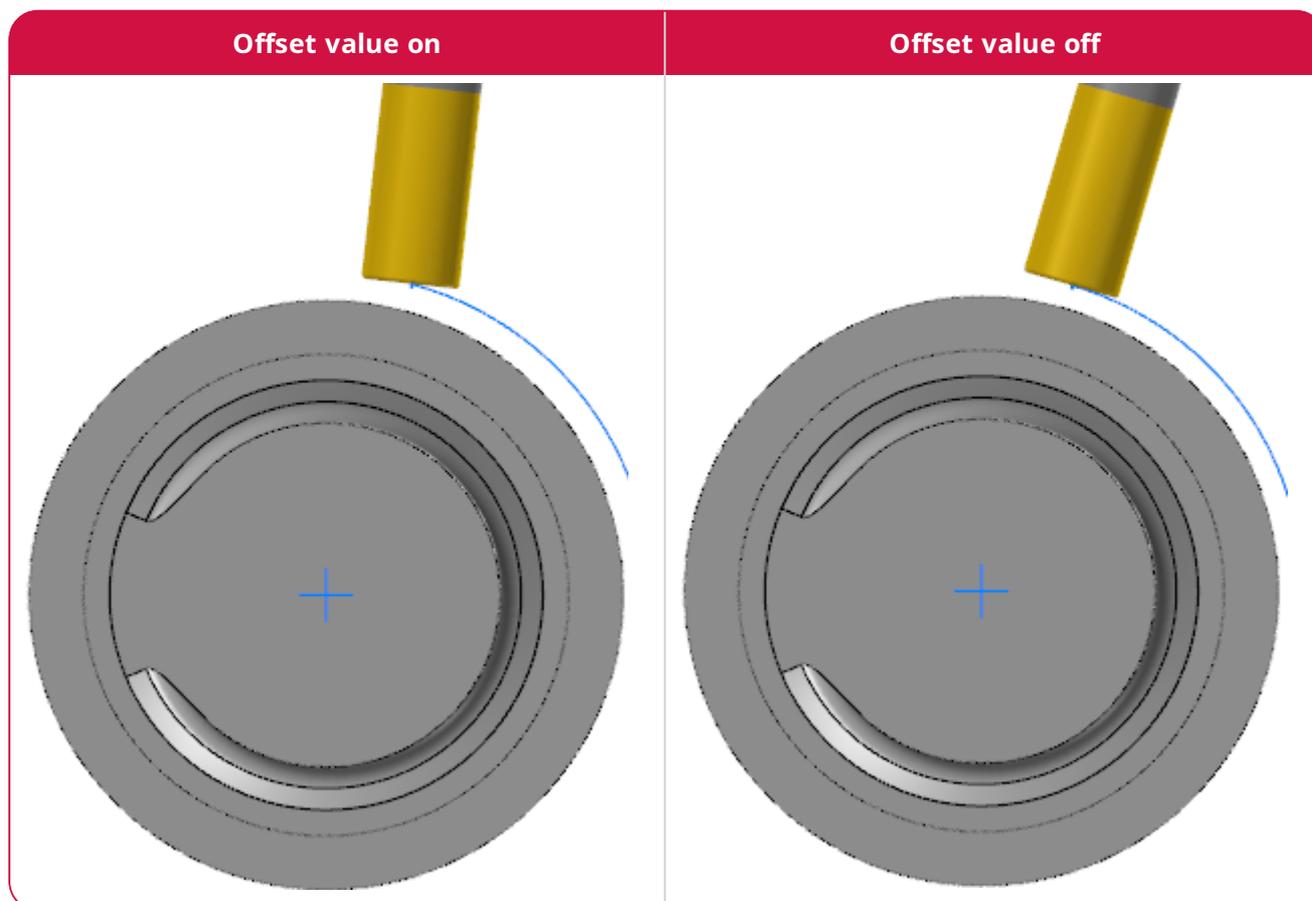
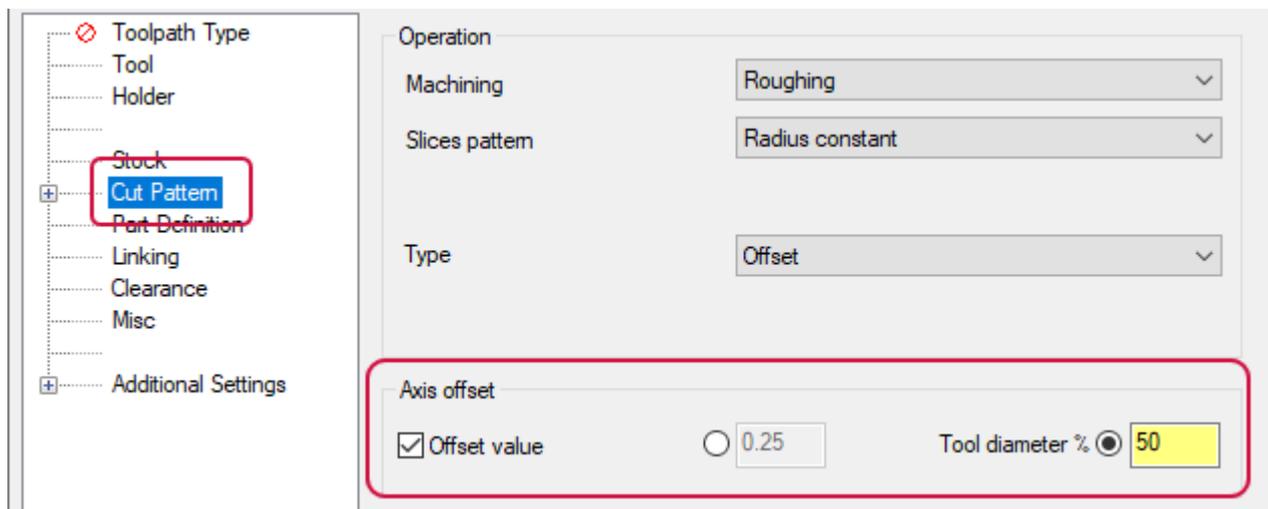


Rotary Advanced Toolpath Enhancements

Listed below are enhancements to the Rotary Advanced toolpath.

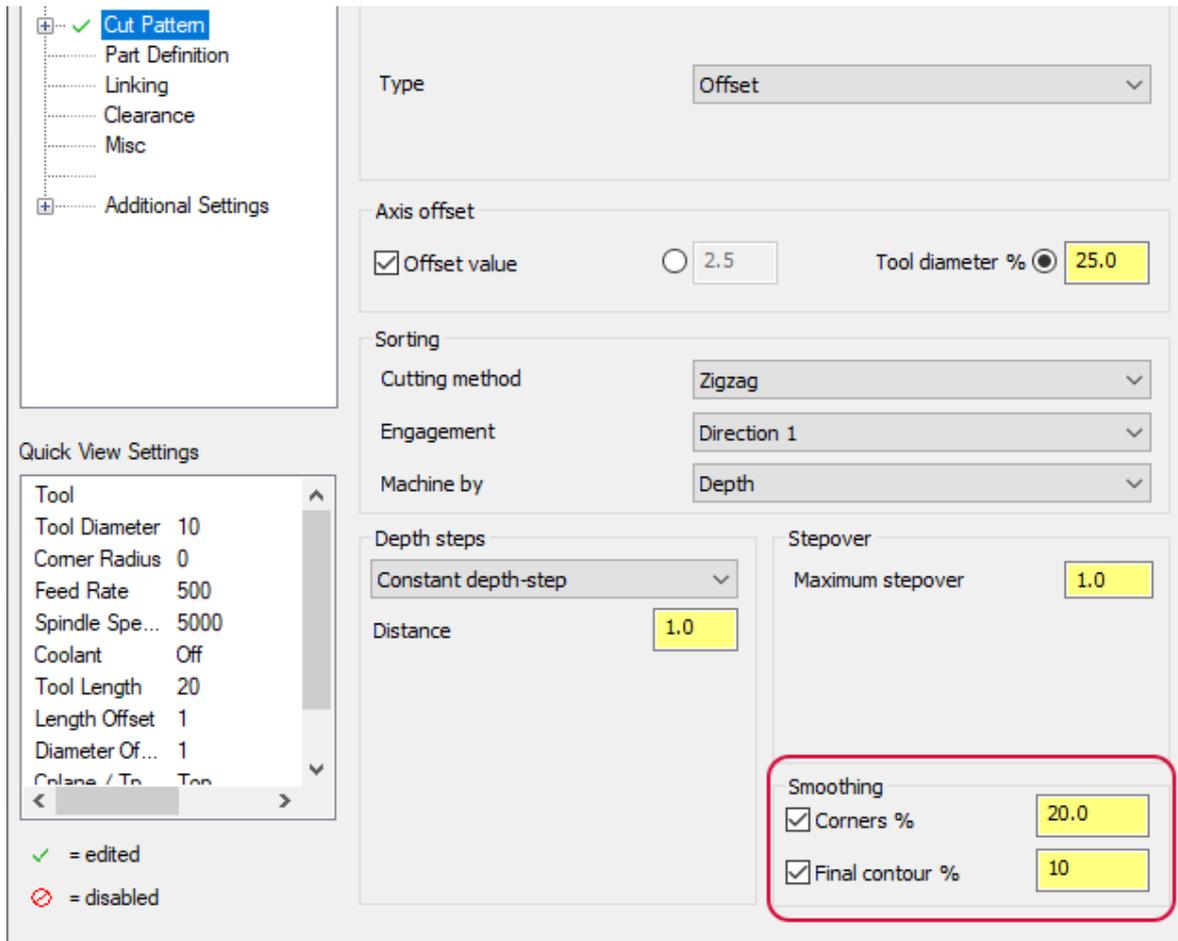
Shifting a Tool Off-Center

The **Axis offset** option for Rotary Advanced toolpaths allows you to shift the tool off-center to maintain a better contact point, which extends tool life and improves the machining quality. This option is located on the **Cut Pattern** page.



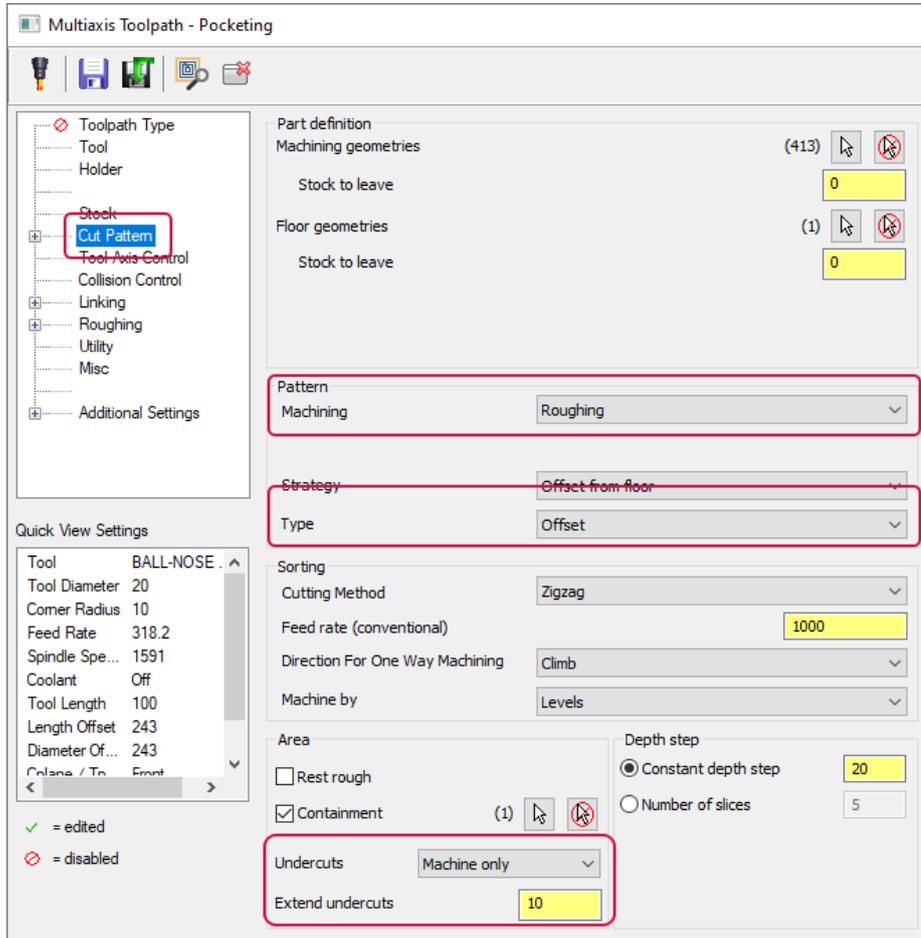
Rounding Sharp Corners

The new **Smoothing** options on the **Cut Pattern** page of Rotary Advanced toolpaths rounds sharp corners using a percent of the stepover.



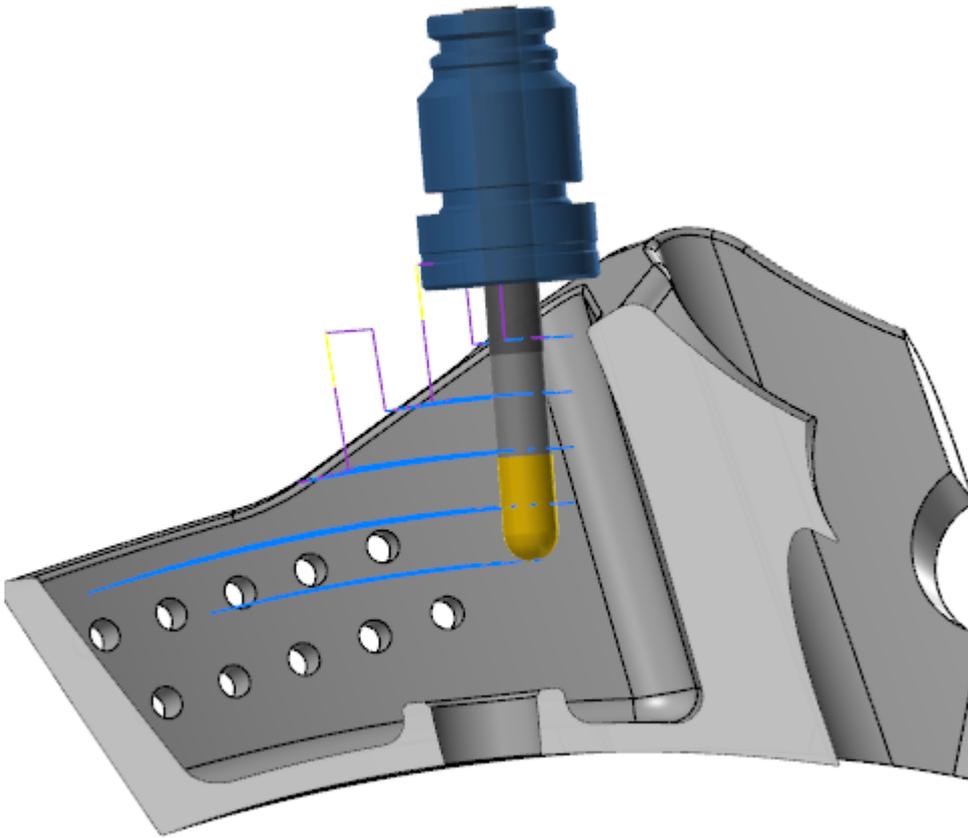
Blending to Undercut Areas for Multiaxis Pocketing

Use the new **Extend undercuts** option to blend a Multiaxis Pocketing toolpath to undercut areas easier. To see this option, set the following on the **Cut Pattern** page:

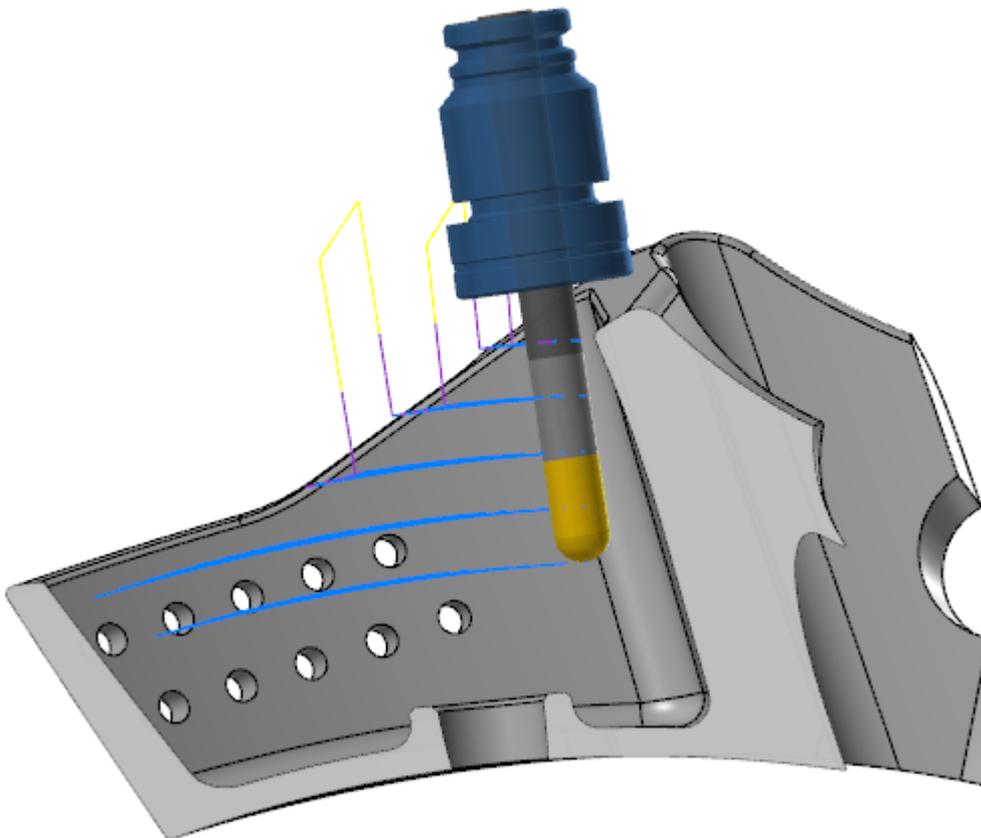


- Set **Machining** to **Roughing**.
- Set **Type** to **Offset**.
- Set **Undercuts** to **Machine only**.

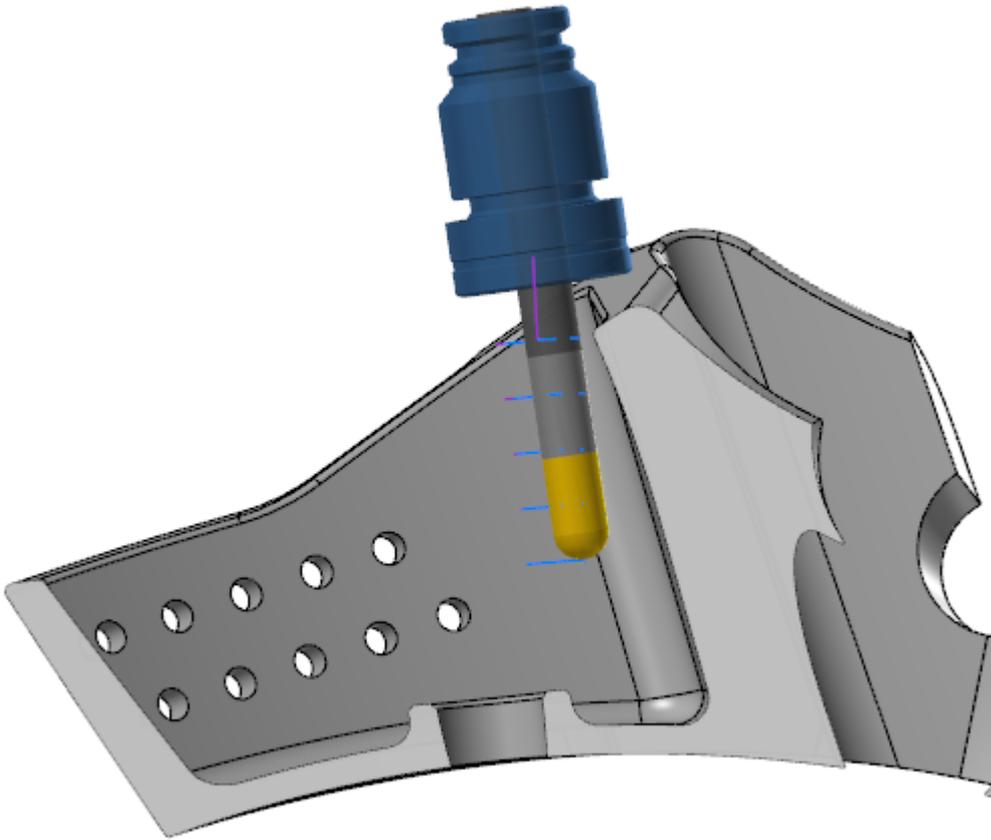
The example below shows a Multiaxis Pocketing toolpath with **Undercuts** set to **No**.



The following example shows the same toolpath, with **Undercuts** set to **Yes**.



The following example shows the same toolpath, **Undercuts** set to **Machine only** and **Extend undercuts** set to **10**.

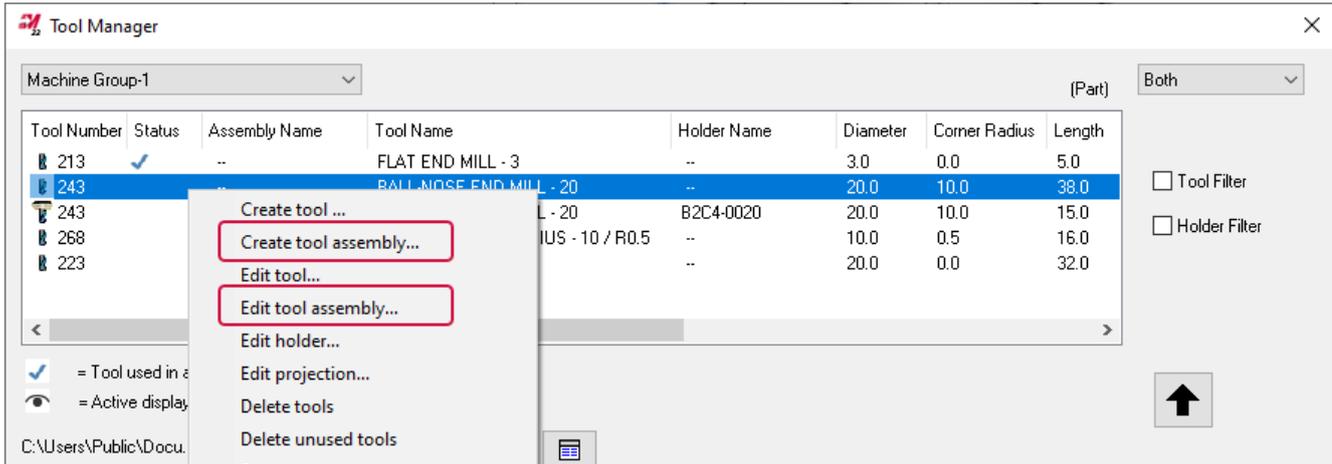


Tooling Enhancements

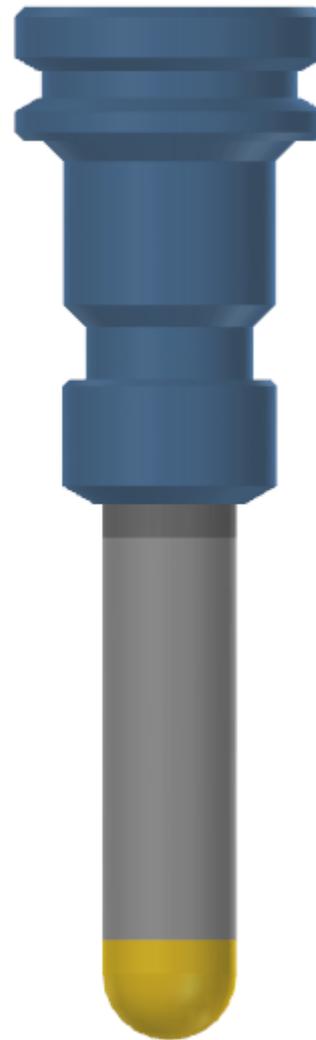
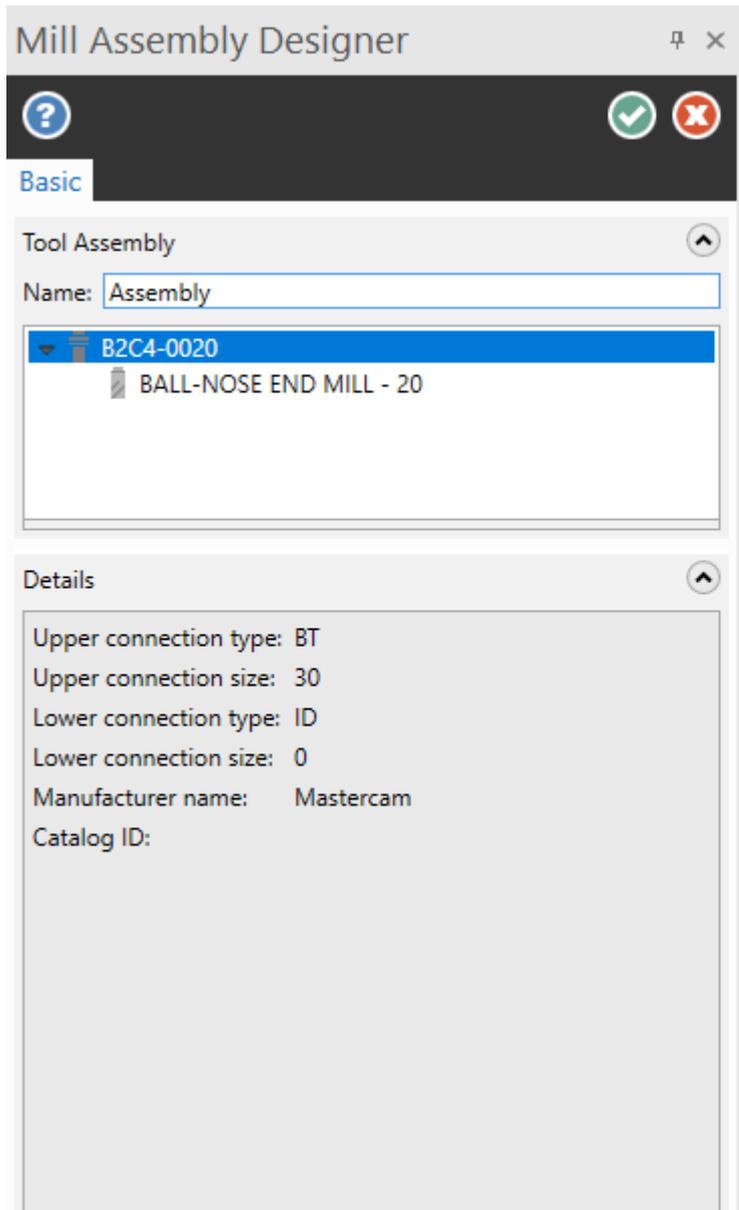
Listed below are enhancements made to mill tools.

Building Mill Tool Assemblies

In Mastercam 2022, you can create and edit multi-holder Mill tool assemblies in the Mill Tool Manager within Mastercam. You can import tools and holders from a library and/or create components from scratch when building your tool assemblies. When you select **Create tool assembly** or **Edit tool assembly** from the right-click menu in the Tool Manager, the **Mill Assembly Designer** function panel displays.

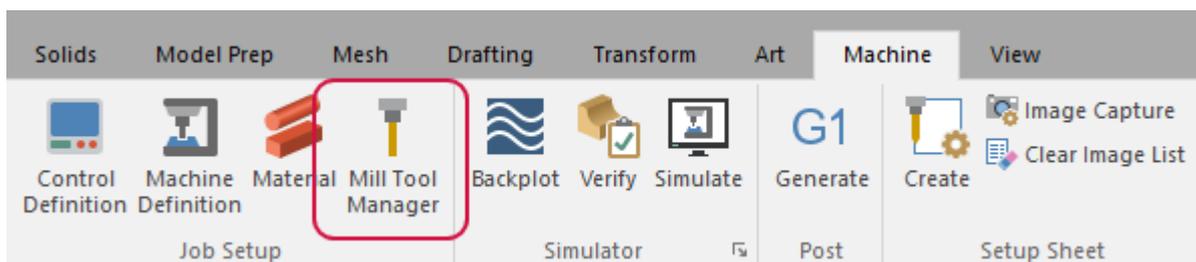


The new workflow allows you to stack multiple holder components from different libraries into one assembly. You can save geometry to a level or file.



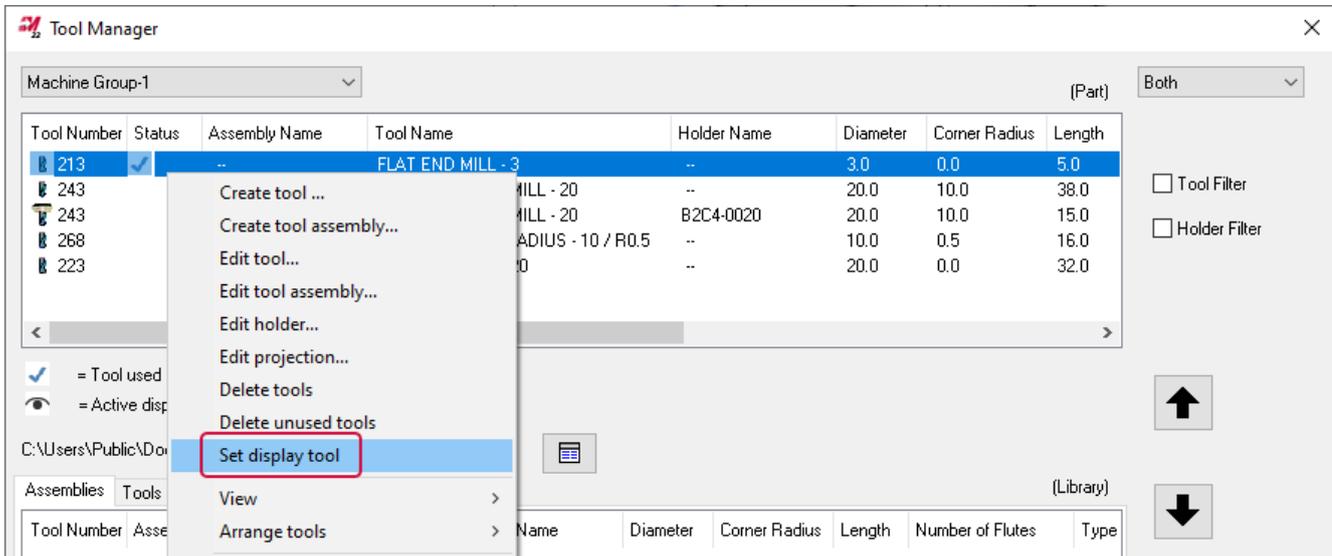
Using the Mill Tool Manager in Design

Prior to this release, the Mill Tool Manager was only available with an active Mill or Lathe license. In Mastercam 2022, you can now access the Mill Tool Manager with just a Design license. This is accomplished by the addition of design groups that you can use to create, edit, update, and delete tools that have not yet been assigned to a Mill machine group. Tools can be moved to and from the design group, either to a machine group in the part tool list or to a library. Select **Mill Tool Manager** on the **Machine** tab to manage your tools.



Setting Your Active Display Tool

Mastercam 2022 introduces the concept of a display tool. When turned on, a display tool shows the selected tool during functions not directly connected to a toolpath. When you choose your display tool, a visual indicator appears in the part tool list.



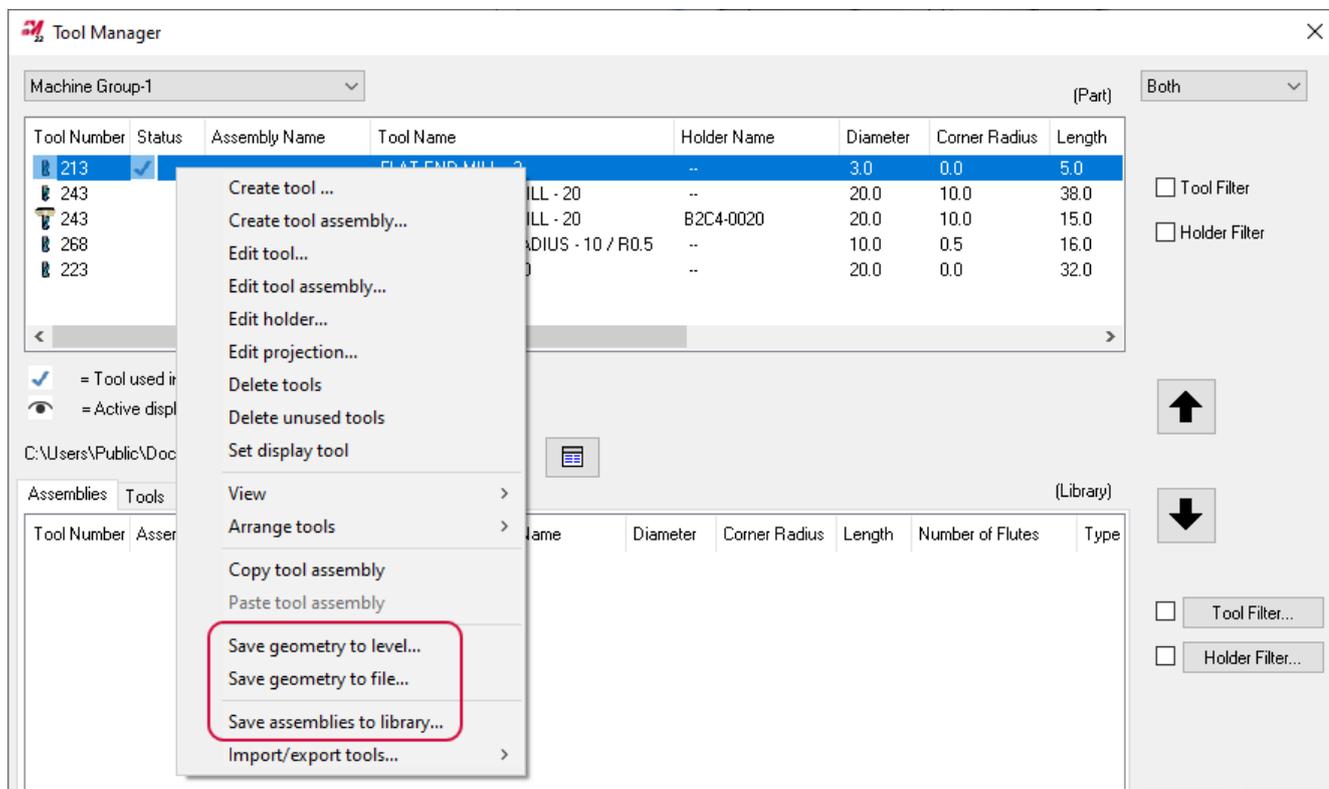
Note that only one display tool may be active at any given time. The display tool can be set only in the upper tool list of the Mill Tool Manager or the Lathe Tool Manager.

NOTE

This new feature works directly with the new **Show Tool** function. You can find out more information about **Show Tool** in "[Viewing Your Tool Assembly During Part Programming](#)" on page 144.

Exporting Tool Components

You can now export tool components to a level, file, or library from the Mill and Lathe Tool Managers. Right-click an item in the list and select an option from the menu. These options save the current tool position as wireframe and/or solid geometry to a specific level using specified entity attributes. This geometry can then be used to create tool assemblies or 3D tools.



TURNING ENHANCEMENTS

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

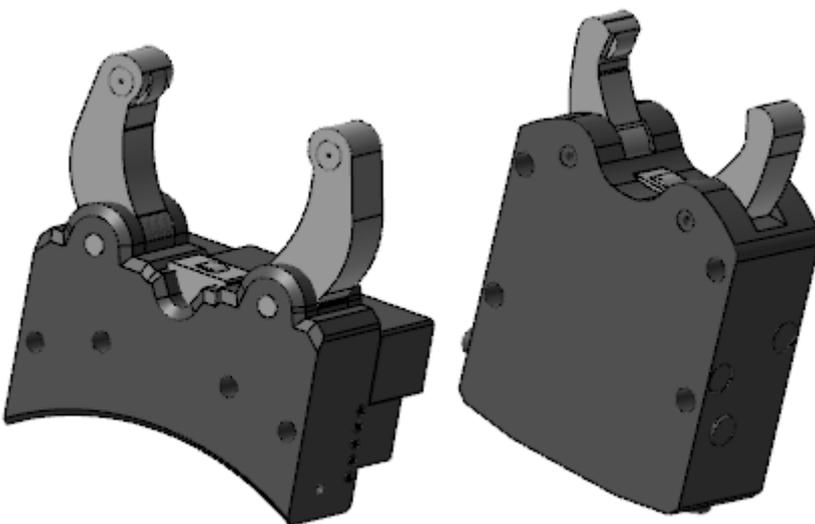
NOTE

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

Working with Mill-Turn Steady Rests

Mastercam 2022 introduces steady rest support for Mill-Turn. This lets you precisely position and simulate steady rests throughout their machining process, adding part support capability to turning, milling, and part-handling operations. This includes:

- New component types that you create for turret-mounted or standalone steady rests. Sub-component types for arms and pistons let you quickly create components with a modular approach.
- Enhanced component libraries that organize steady rests in the same libraries used for tool locators, chucks, jaws, and collets. This lets you easily select and load from a single library all of the components needed to configure your machine.
- Dedicated operations for each type of steady rest let you precisely program each individual movement. You can create syncs to specific steady rest events.
- Part handling strategies let you pre-program sequences of steady rest operations or add steady rest operations to existing part handling strategies.
- All steady rest operations and movements—including the movements of pistons and arms—are fully supported in simulation. You can also define and customize the boundary used for collision checking and turn it on or off individually for each steady rest operation.



Mill-Turn's steady rest support goes far beyond what has traditionally been available in Mastercam Lathe. However, even Lathe users without a Mill-Turn license can use single-stream .machine files to access Mill-Turn's steady rest capabilities.

Creating Steady Rest Components

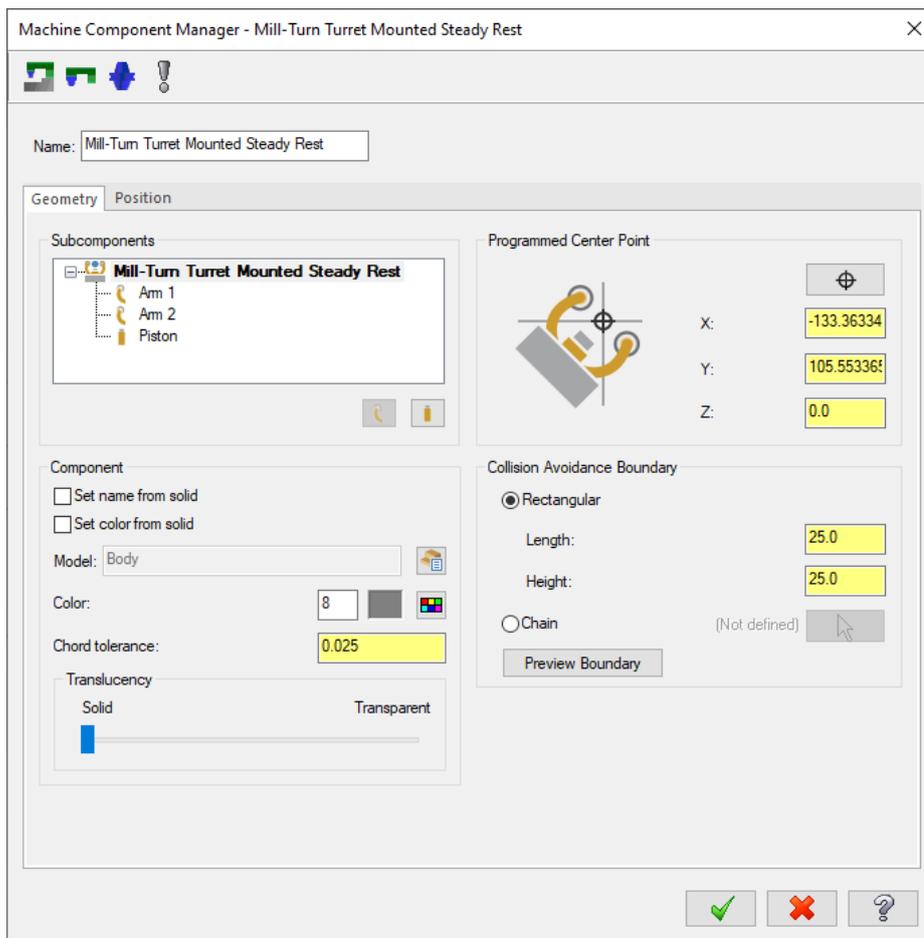
Mastercam's modular component architecture lets you efficiently create different types of steady rests. These include:

- Arm-and-piston steady rests.
- Ring-based steady rests.

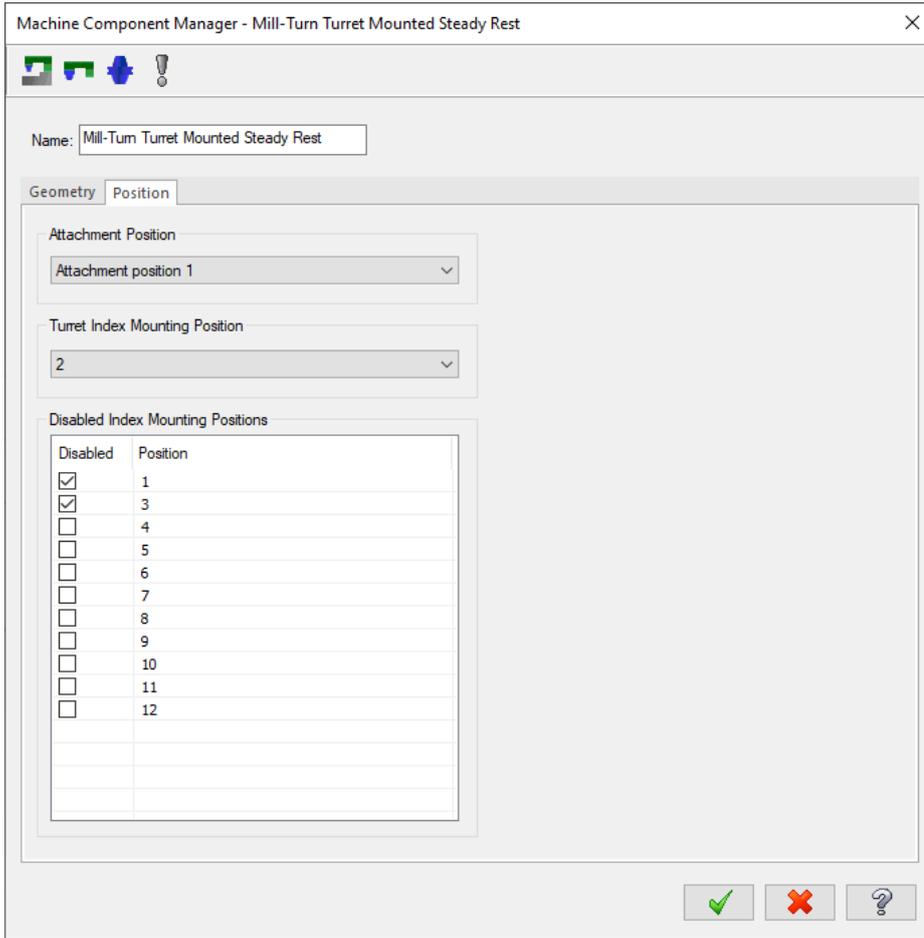
Arms and pistons are defined as individual sub-components. This means that arm-and-piston motions are individually represented and accurately simulated.

- Turret-mounted steady rests can be created by all Mastercam users. They are stored in component libraries, from where they can be selected for individual parts in Job Setup and easily reused in different machines.
- Standalone steady rests are defined in the machine definition. They can only be created by licensed machine developers. These steady rests are not mounted on a turret, but directly on the machine.

Use the **Geometry** tab to organize the sub-components and solid models in a tree. Create, edit, and remove sub-components as needed. You can also define the programmed point and collision boundary.



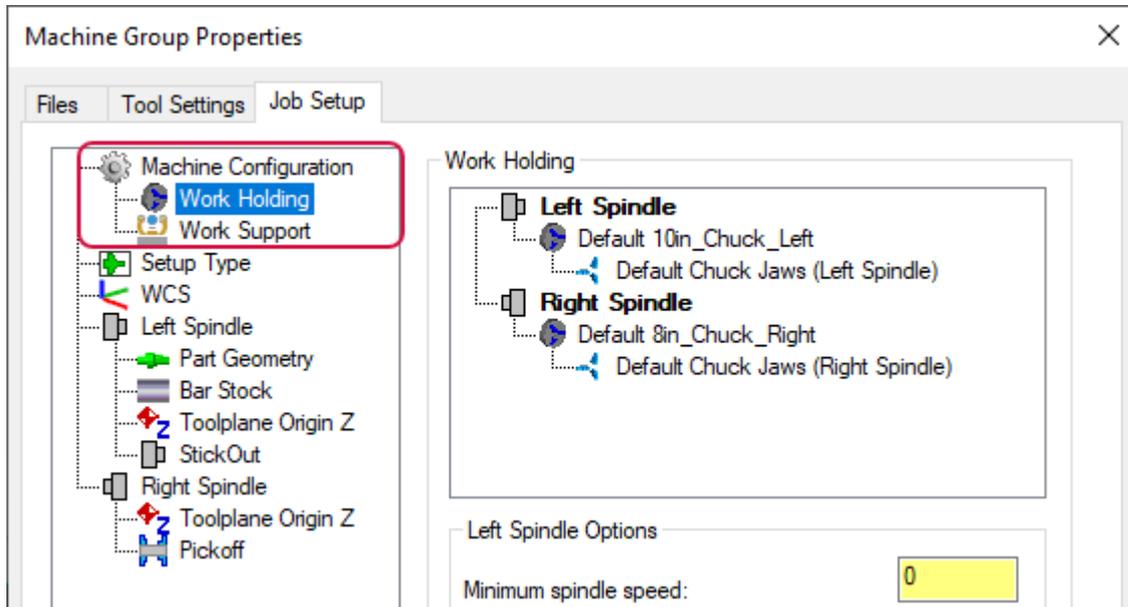
Use the **Position** tab to describe how turret-mounted steady rests connect to the turret. Steady rests use the same attachment positions as tool locators. Select the index position on which the steady rest will be mounted, then tell Mastercam if any additional index positions need to be disabled.



Standalone steady rests use the **Position** tab to specify in which data stream the steady rest commands will be output and whether or not you are able to remove or dismount the steady rest during Job Setup.

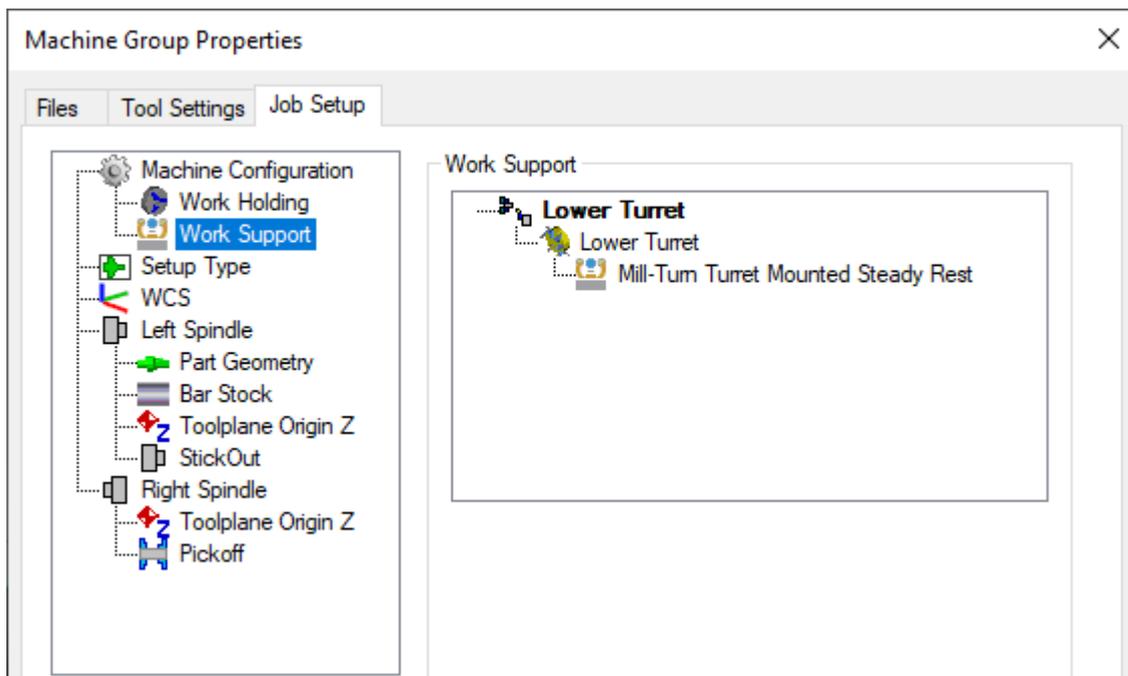
Selecting Steady Rests in Job Setup

The new **Work Support** page in Job Setup allows you to select a steady rest for your part. The **Machine Configuration** page from previous versions of Mastercam has been split into separate **Work Holding** and **Work Support** pages.



The **Work Holding** page is used to select and configure chucks, jaws, and collets. Mastercam only displays the new **Work Support** page if your machine includes a steady rest or a turret to which a steady rest can be added. For example, if your machine has only a B-axis head, you will not see the **Work Support** page.

The **Work Support** page lets you select either a turret-mounted steady rest from a component library or a standalone steady rest from the machine definition (if one has been defined). Standalone steady rests are listed in a **Steady Rest Group**. Turret-mounted steady rests are available in a **Turret Group**.



Right-click **Steady Rest (Empty)** and choose **Select new steady rest** to select a steady rest from the current component library. You can also open a different component library. Once a steady rest has been selected, you can right-click on it to edit its properties if desired.

Selecting a standalone steady rest activates additional Job Setup options:

- You can choose to use the collision detection boundary from the component definition.
- You can also choose to remove the standalone steady rest, if one has been defined as removable. Deselecting this option means that the steady rest will not be programmable, and will not appear in simulation or interfere with any other components.

Steady Rest Support in Part Handling Strategies

Mastercam provides two types of operations for programming your steady rests:

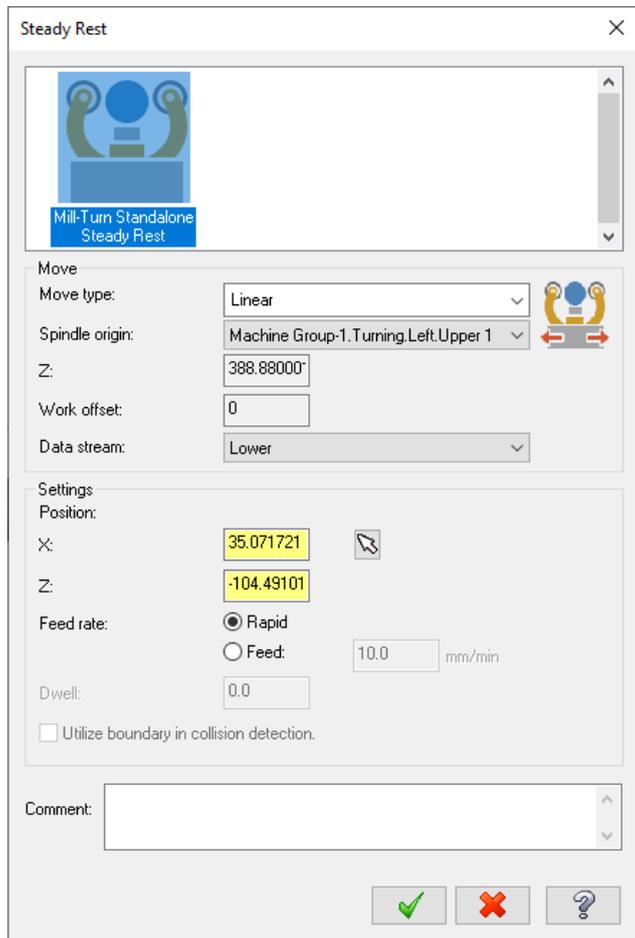
- Steady rest operation
- Steady rest point operation

The New Steady Rest Operation

Steady rest operations are used to program standalone steady rests. Four actions are supported:

- Clamp
- Unclamp
- Linear move
- Retract

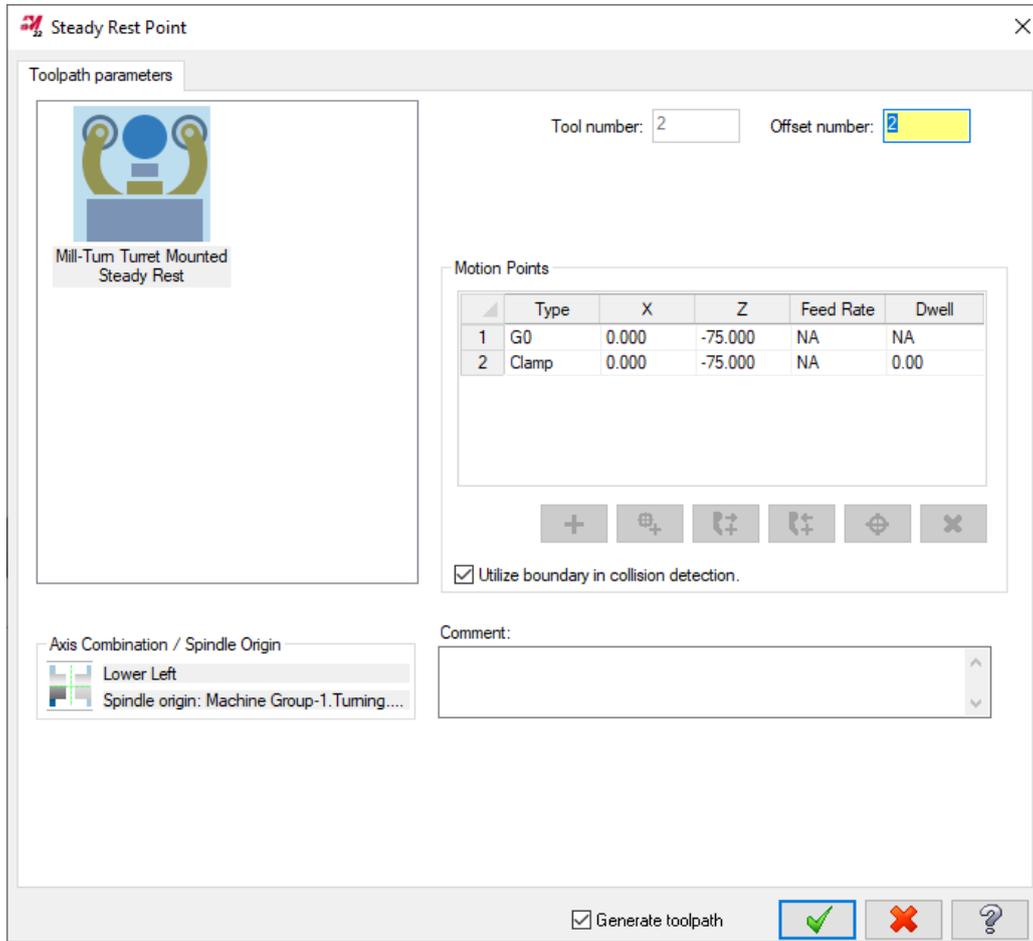
Each individual event is represented by a separate operation. This gives you maximum flexibility for how the steady rest actions interact with your other operations. For example, after clamping the steady rest on your part, you can program the steady rest to move while the part is being machined, following the tool or turret. You can also create syncs to individual steady rest events in the Sync Manager.



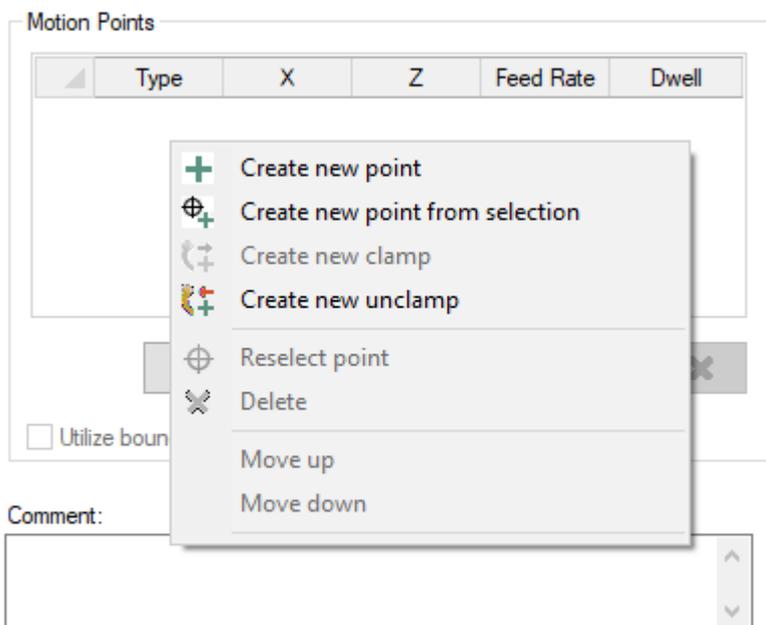
The options and settings that are available are specific to the selected **Move type** and the capabilities of your machine.

The New Steady Rest Point Operation

Steady rest point operations are used to program turret-mounted steady rests. These operations use a grid to organize the points and other steady rest actions, like a Center Point operation.



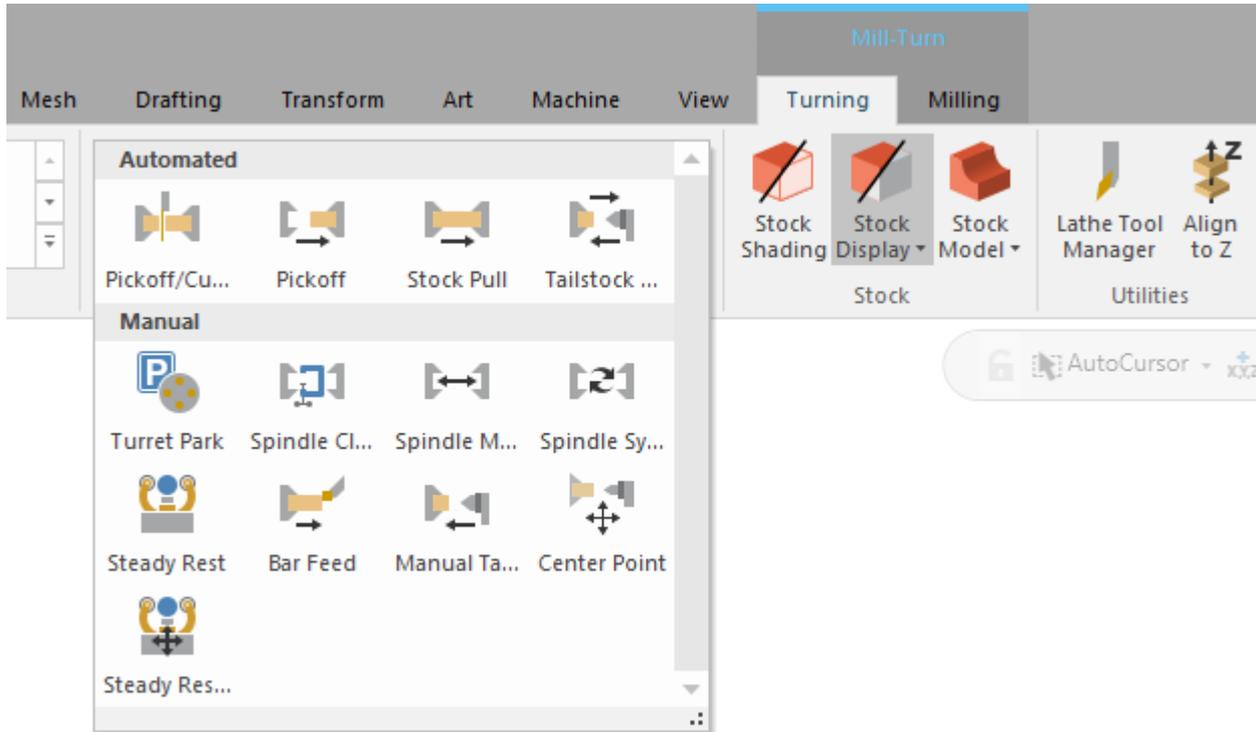
Right-click in the grid or use the buttons below it to add new points or clamp/unclamp events to the operation.



As you add points and events, Mastercam filters the available options to only those that make sense. For example, once you clamp the steady rest, the options to add new points are disabled.

Adding Steady Rest Events to Part Handling Strategies

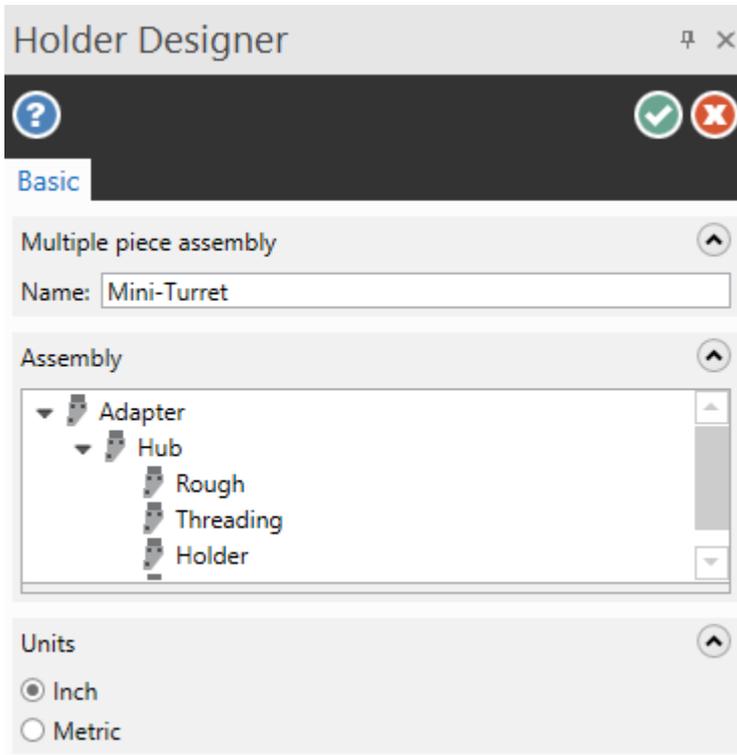
Mastercam 2022 allows you to incorporate steady rest events into the part handling operations for your machine. Part handling operations let machine developers precisely control the interaction between spindles, chucks, turrets and other components so that you can create detailed part handling applications with a single click. Once machine developers add strategies to the .machine file, you can select them from Mastercam.



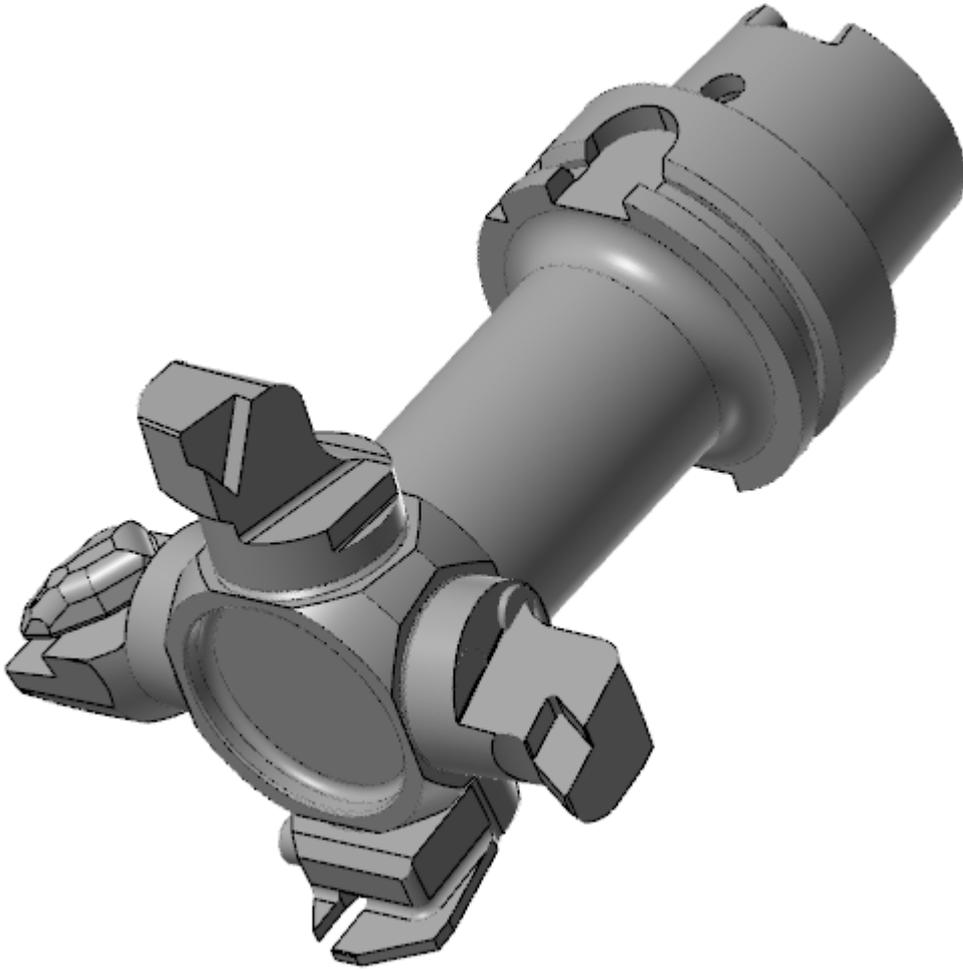
Machine developers have complete flexibility to add steady rest events to these strategies. For example, you can create separate pickoff-cutoff strategies with and without steady rest events. Machine developers can create strategies for either turret-mounted or standalone steady rests. Strategies for turret-mounted steady rests will automatically use the specific steady rest that the user selects during Job Setup.

Using Multiple Adapters in 3D-Turning Tooling

In Mastercam 2022, you can now define reusable holder assemblies in the **Holder Designer**, which is accessible through the Lathe Tool Manager **Holder** tab. Use the **Holder Designer** to create collections of adapters, extensions, and holders, and store them as an assemblies. You can also access the **Holder Designer** by clicking **Select from library** for a holder component, right-clicking in the **Browse for item** dialog box, and choosing to create or edit a holder.

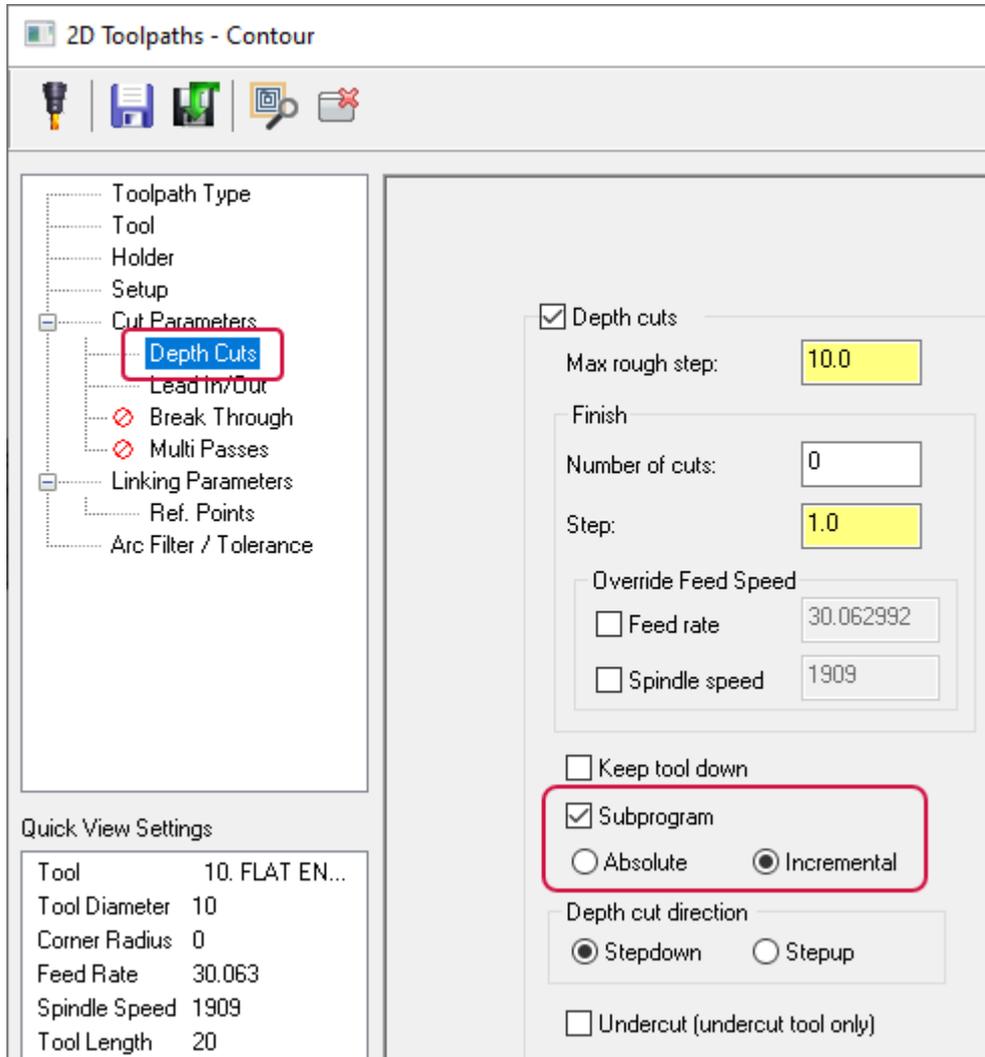


You can save time and avoid errors, by creating reusable assemblies of modular components. These assemblies can be stored in a library and used for the creation of full 3D Lathe Tool assemblies.



Subprograms Supported in Mill-Turn

The subprogram support found in other Mastercam products has been extended to Mill-Turn. You can create subprograms for depth cuts and hole-making operations such as drilling. These are the same options, in the same locations, that are available in Mill.

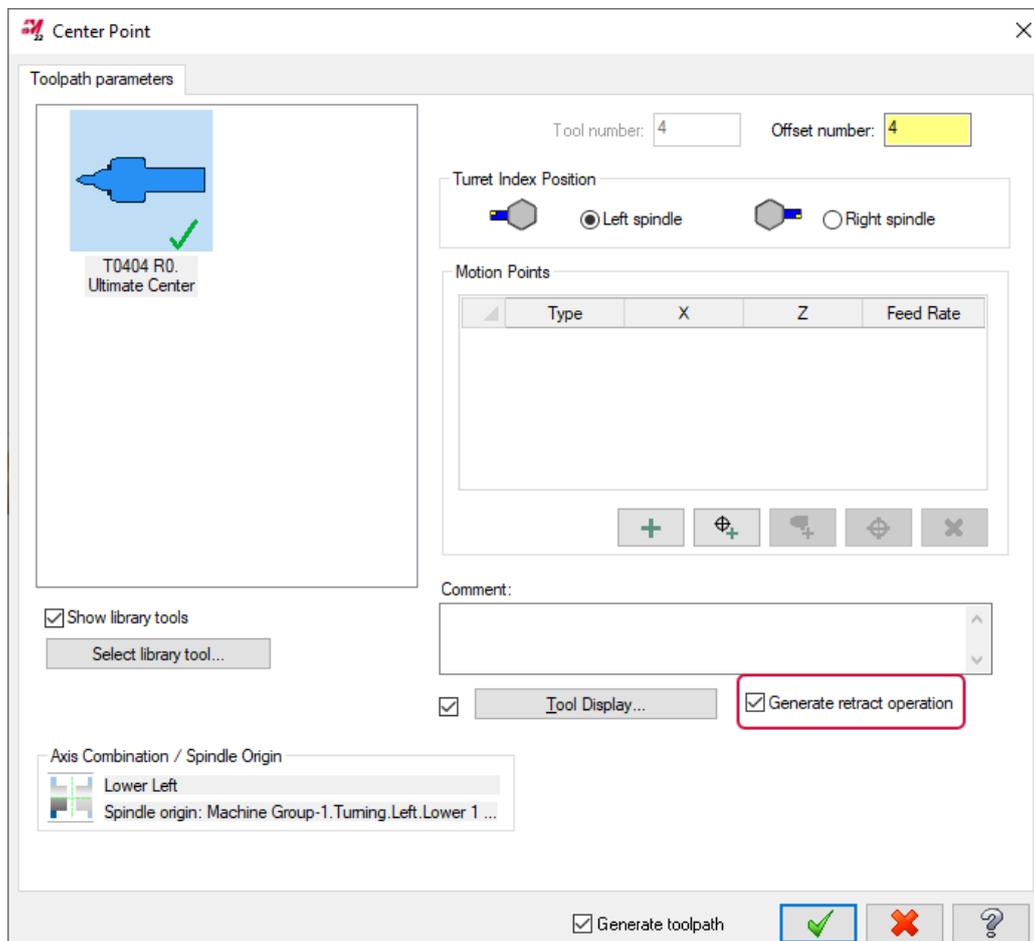


The subprogram options are only available when they are supported by the selected machine environment. Machine environment developers can enable or disable subprogram support as necessary for each individual machine environment. This is similar to other Mastercam products, where subprogram support is indicated in the control definition. Options available to machine developers let them completely customize subprogram numbering, how they are output, and which options to expose to end-users in their machine environments.

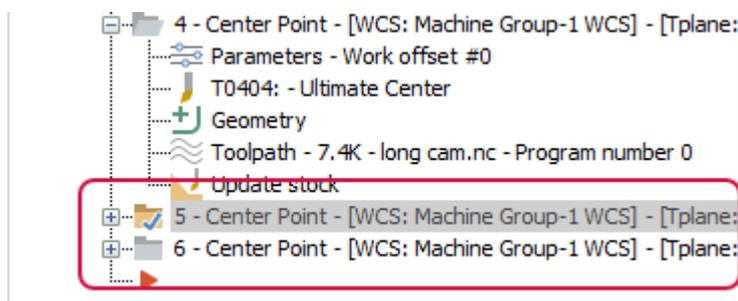
Note that transform and nested subprograms are not supported.

Automatically Retracting to Center

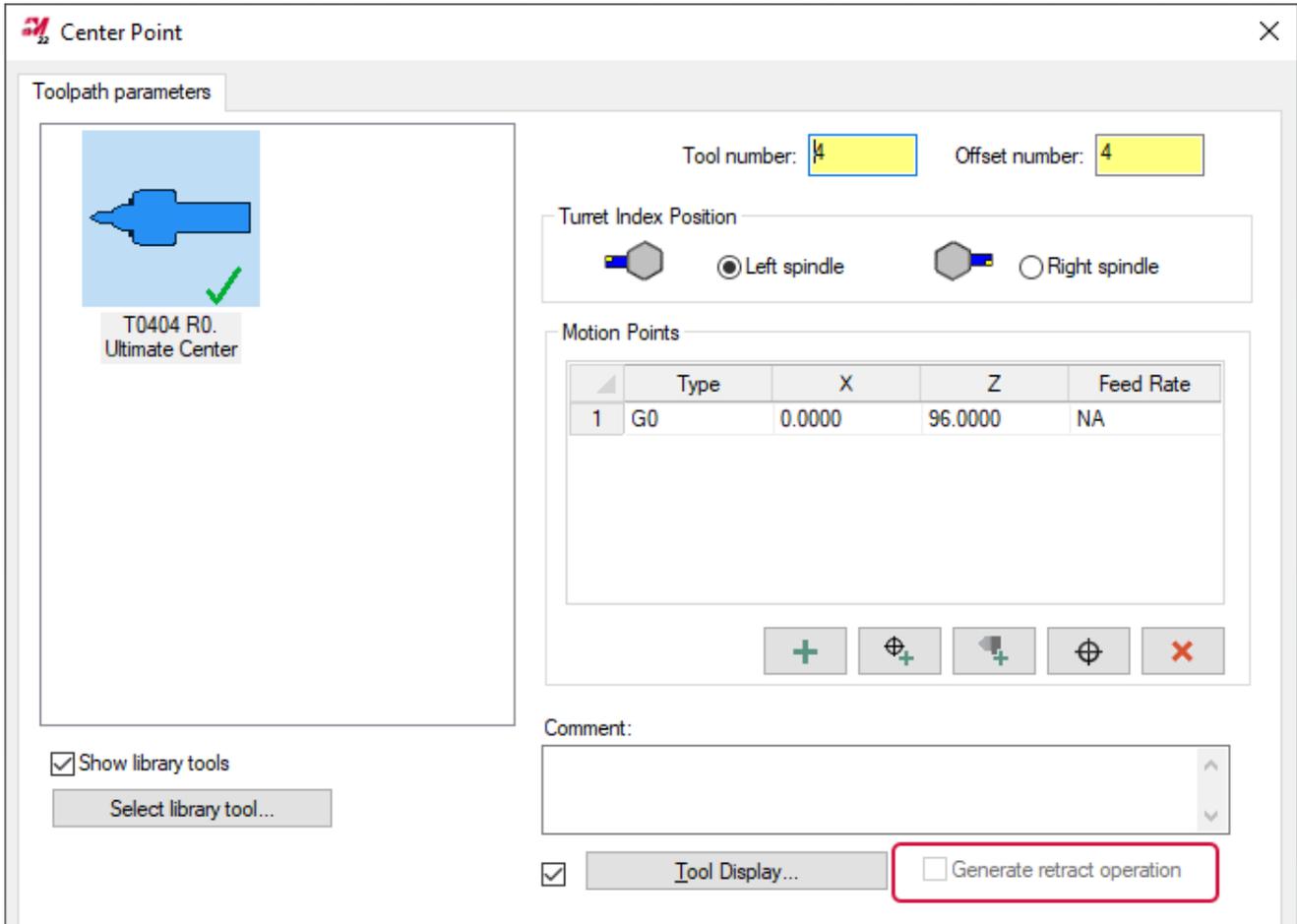
When creating a Center Point operation in Mill-Turn, you now have the option to have Mastercam automatically retract the center. Select the new **Generate retract operation** option to enable the new operation. This will create a second operation in the Toolpaths Manager, separate from the engage operation.



In the example above, the engage operation contains two moves: a rapid to Z96, then the move to engage. The new operation will contain a single retract move to the position immediately before the center engages the part—in this example, Z96. After selecting **OK**, Mastercam creates two operations in the Toolpaths Manager.

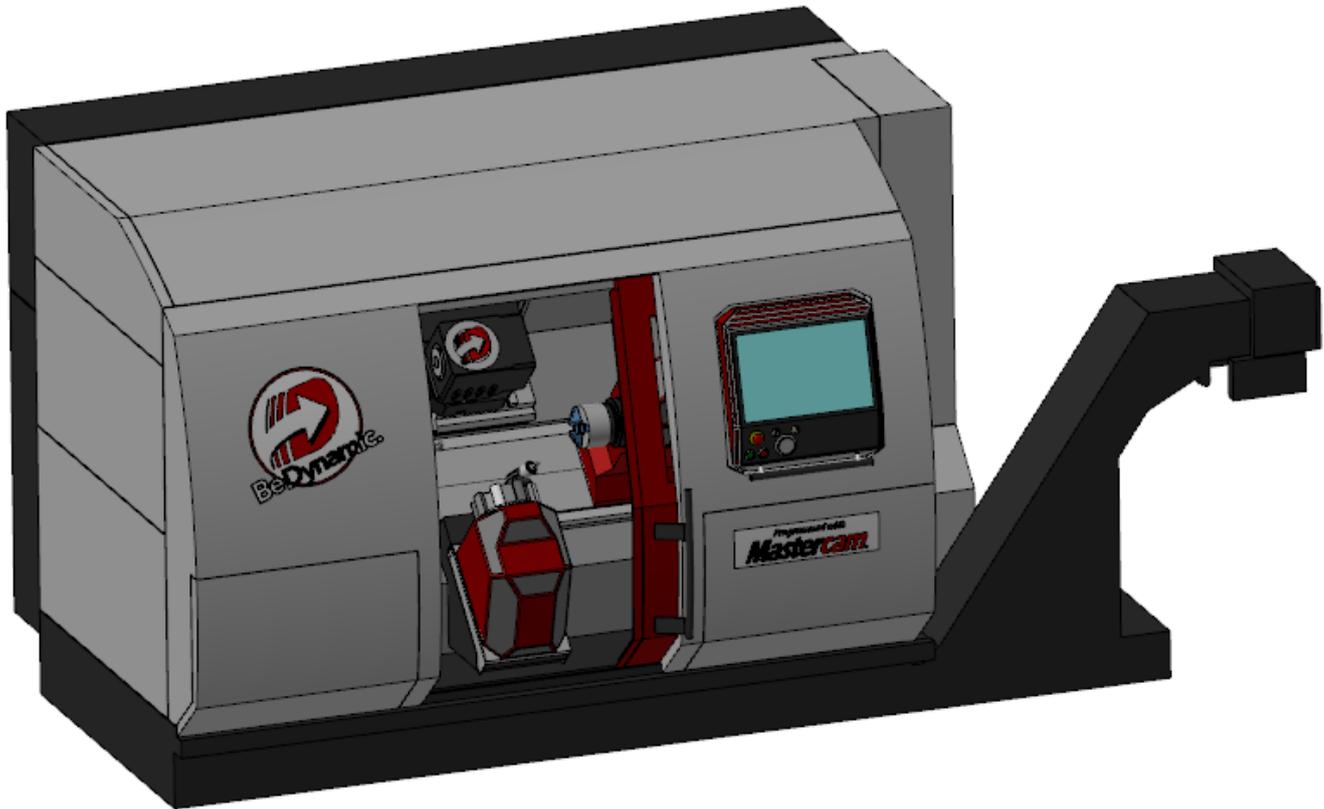


The second operation contains a single move back to Z96. Also, in this case the **Generate retract operation** option is disabled. Mastercam disables the option whenever the center is already engaged in the part.



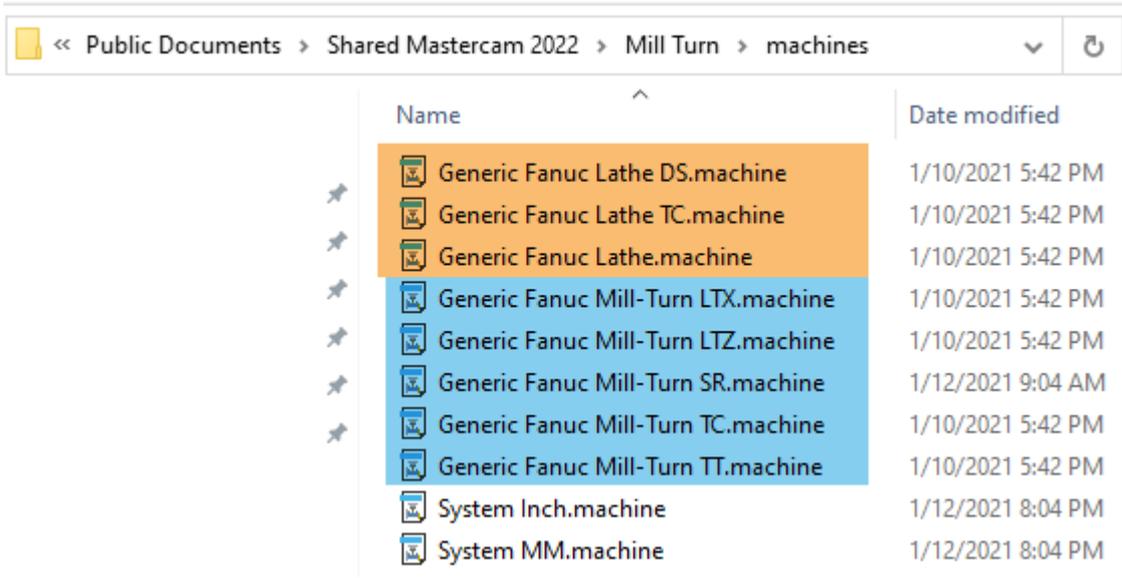
Generic .machine Files Installed with Mastercam

Mastercam 2022 now includes a complete set of generic .machine files that are installed with Mastercam. Several machines include steady rest components so you can try out the new steady rest capabilities in Mill-Turn. These include a new Generic Fanuc Mill-Turn SR machine equipped with a B-axis head and standalone steady rest. In addition, the Generic Fanuc Mill-Turn LTZ (shown below) and Generic Fanuc Mill-Turn TT include turret-mounted steady rests in their component libraries.



The generic Fanuc .machine files that are installed with Mastercam can be used with Demo/HLE without an additional license. The generic Siemens .machine files can also be used with Demo/HLE, but these need to be downloaded from the [Tech Exchange](#).

The following machines are installed:



- Machines highlighted in orange are single-stream machines that can be accessed by licensed Lathe in Mill-Turn users. You can access all Mill-Turn functionality appropriate to the capabilities of the machine and your Mill license, but they cannot run the multi-stream machines highlighted in blue.
- Machines highlighted in blue are multi-stream machines that require a Mill-Turn license.
- The System Inch and System MM machines are for internal use by Mastercam. Do not try to load or open these machines.

CAUTION

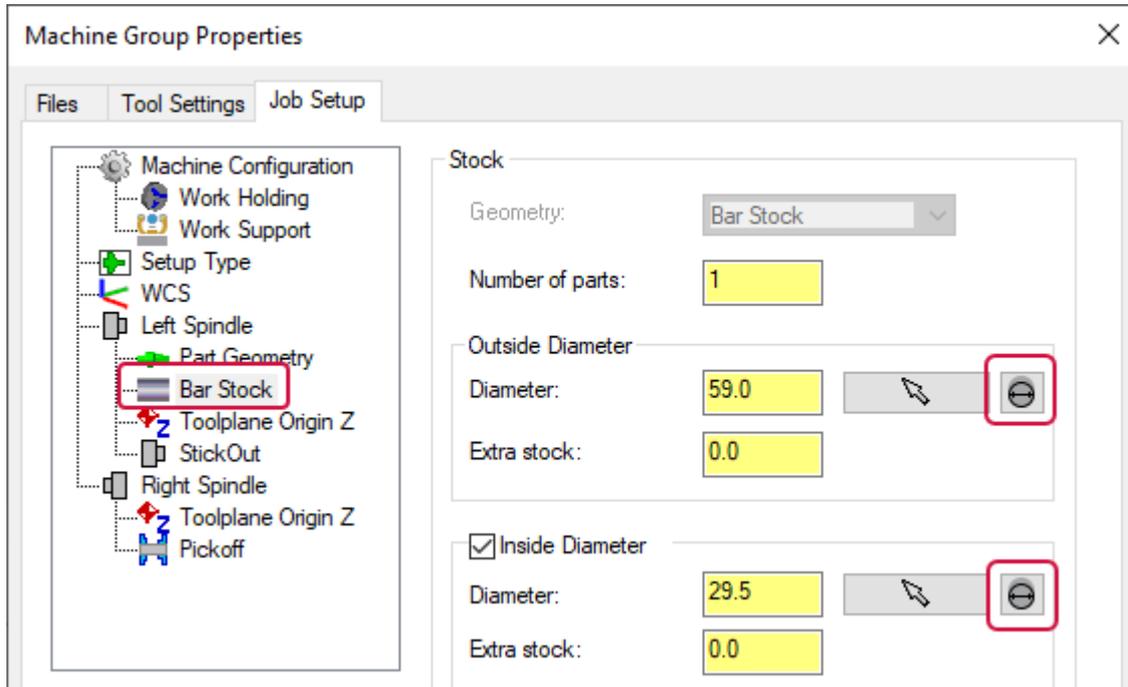
The posts in these machines are designed to output generic NC code for demonstration and training purposes. They are **not** intended to produce code for any actual machine tool. Do **not** use these to cut parts on an actual machine!

The following table describes each machine.

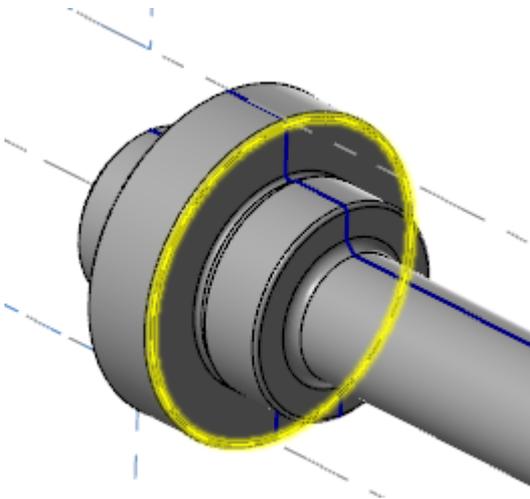
Machine	License	Application
Generic Fanuc Lathe	Lathe + Mill	Single turret, single spindle.
Generic Fanuc Lathe DS	Lathe + Mill	Single turret, dual spindle.
Generic Fanuc Lathe TC	Lathe + Mill	Single turret, single spindle with tailstock.
Generic Fanuc Mill-Turn LTZ	Mill-Turn	Dual spindle with B-axis head and lower turret that rotates about the Z-axis.
Generic Fanuc Mill-Turn LTX	Mill-Turn	Dual spindle with B-axis head and lower turret that rotates about the X-axis.
Generic Fanuc Mill-Turn SR	Mill-Turn	Dual spindle with B-axis head and standalone steady rest.
Generic Fanuc Mill-Turn TC	Mill-Turn	Single spindle with B-axis head and tailstock.
Generic Fanuc Mill-Turn TT	Mill-Turn	Dual spindle, dual turret.

Setting the Stock Diameter as an Arc or Radial Face

When defining stock dimensions in Job Setup, a new button lets you specify the outer or inner diameter by selecting an arc or radial face.

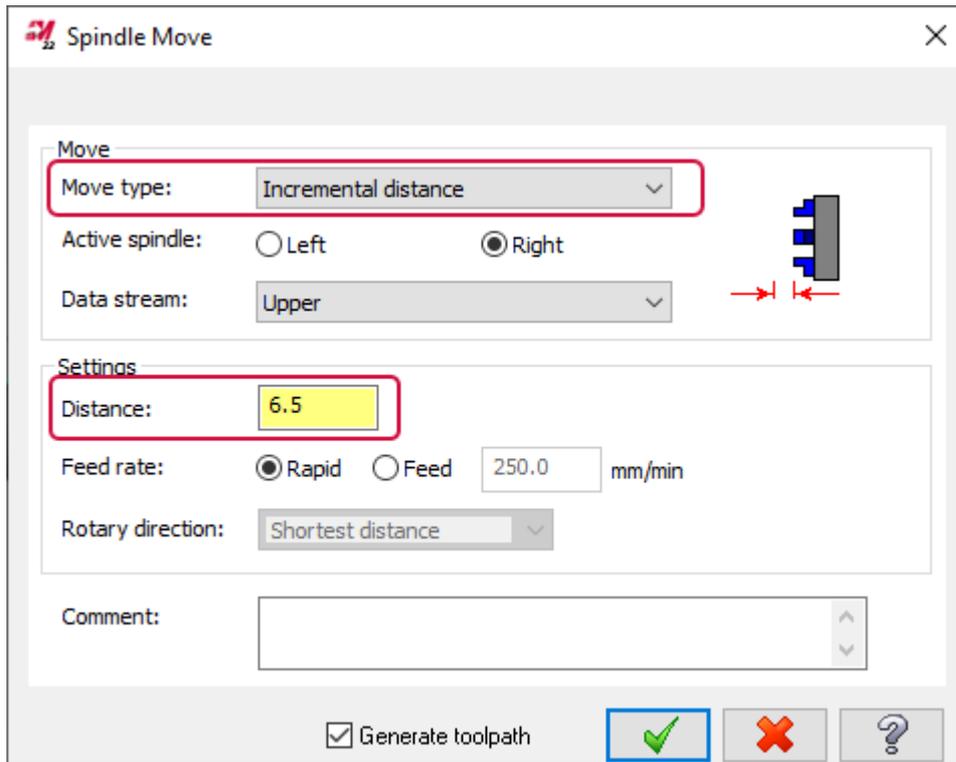


After selecting the button, you are returned to the graphics window where you can select an arc or radial face. You can also select an edge. Mastercam automatically reads the proper diameter value from the selected geometry.



Adding Incremental Spindle Moves

Spindle Move operations in Mill-Turn now let you specify the destination as an **Incremental distance**.



Enter a positive value in the **Distance** parameter to move the spindle towards its home position, or a negative value to move the spindle away from its home position.

DESIGN ENHANCEMENTS

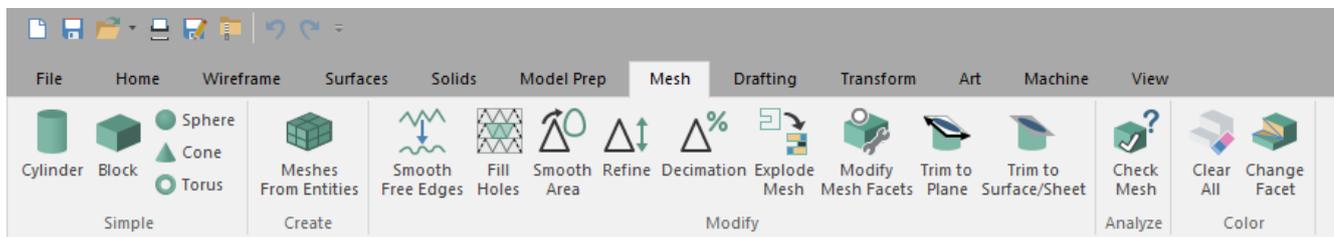
Listed below are major enhancements to the Design product.

New Mesh Functions and Support for Mesh Bodies

Mastercam 2022 expands support for the creation and editing of mesh entities. The new **Mesh** tab contains the new and updated functions. Listed below are the new and improved functions for creating and editing meshes.

Creating Mesh Bodies

In Mastercam 2022, you no longer need to save a solid or surface to a stereolithography (STL) file, then merge it back into your part file to create a mesh. All primitive functions can now create mesh bodies.



Select the **Mesh** entity type in the function panel to create a mesh. The **Meshes From Entities** function allows you to create a single mesh from multiple surfaces, solid bodies, mesh entities, or facets.



The functions listed below now include the ability to interact with mesh entities.

- **Analyze Dynamic** (on the **Home** tab): Analyzes a mesh dynamically across its facets.
- **Curve Slice by Plane**, **Curve Slice Along Curve**, and **Curve at Intersection** (on the **Wireframe** tab): Now support meshes along with surfaces, solids, and wireframe to create wireframe on meshes to use for other processes. For example, to turn scanned data for an intake port into cross-sections that you can use to create lofted surfaces or solids.
- **Point Dynamic** (on the **Wireframe** tab): Creates points dynamically on a mesh.

- **Line Normal to Point, Line Normal to Grid, and Line Normal Along Chain** (on the **Wireframe** tab): Create normal lines on a mesh to control the tool axis in a multiaxis toolpath.
- **Disassemble** (on the **Model Prep** tab): Organizes multiple meshes into a specific plane.
- **Change Normal** (on the **Surfaces** tab): Reverses the normal direction of a mesh.
- **Project** (on the **Transform** tab): Now supports meshes along with surfaces and solids.
- **Geometry Nesting/Toolpath Nesting** (on the **Transform** tab and **Toolpaths** contextual tabs): Nests meshes to fit parts onto machining surfaces or for additive manufacturing.
- **Modify at Intersection** (on the **Wireframe** tab): Breaks, trims, or creates a point where wireframe entities intersect with a mesh.

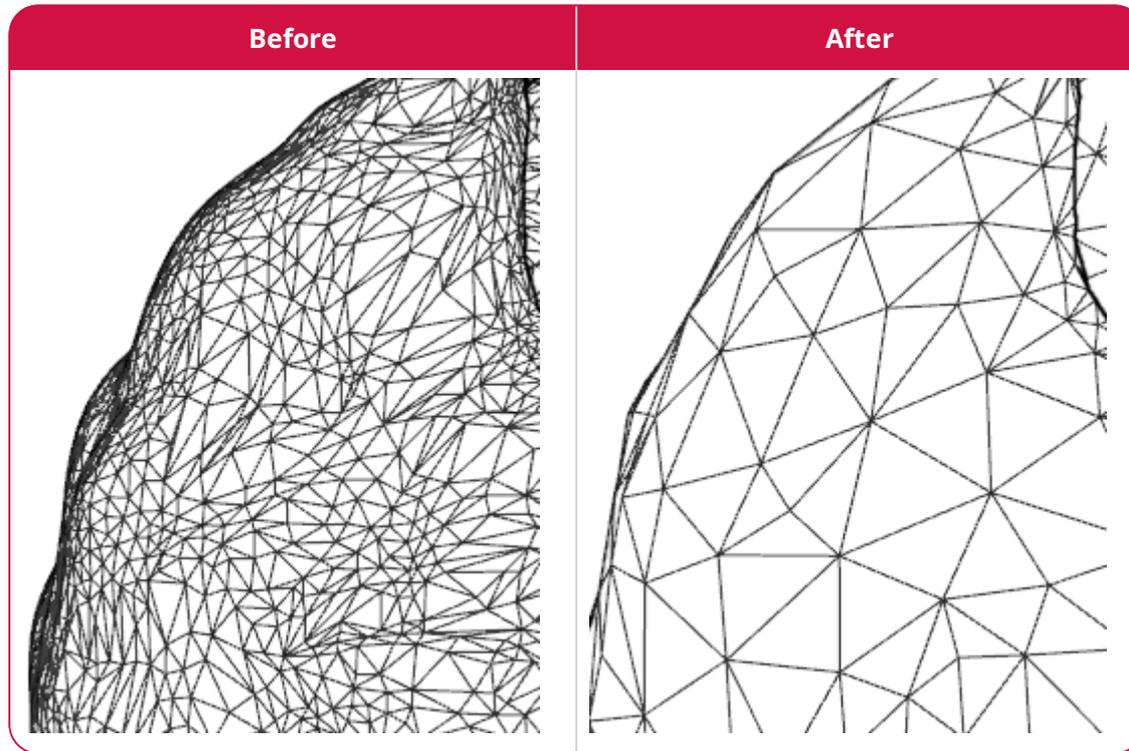
Modifying Mesh Bodies

In addition to creating meshes, you can now modify imported or scanned mesh bodies using the following new functions, found on the **Mesh** tab:

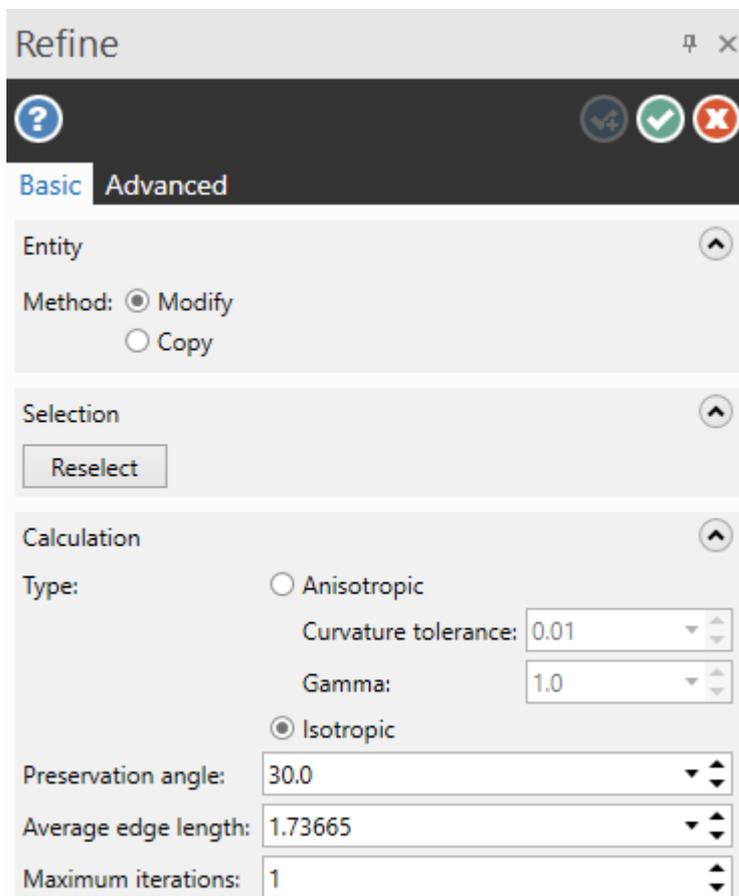
- Refine
- Decimation
- Trim to Plane and Trim to Surface/Sheet
- Fill Holes
- Smooth Free Edges
- Modify Mesh Facets
- Change Facet
- Smooth Area
- Explode Mesh

Refining Mesh Bodies

Use the new **Refine** function to improve your mesh model for 3D printing by adding more data points (facets) to detailed areas and reducing them in less detailed areas.

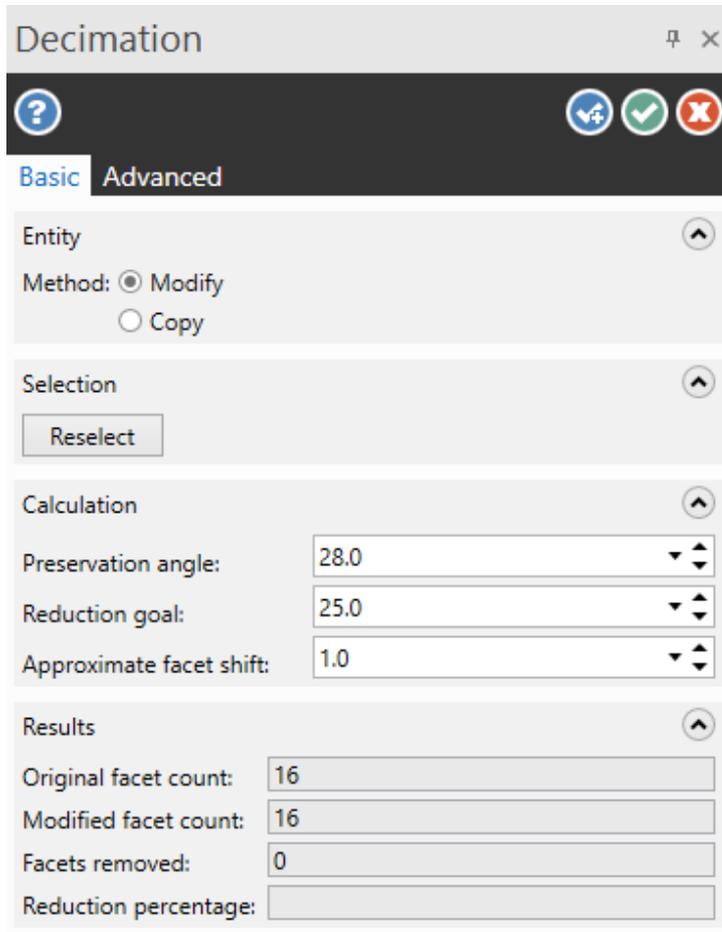


The **Refine** function panel lets you control the size of the facets and their edges to get a result that fits your needs.



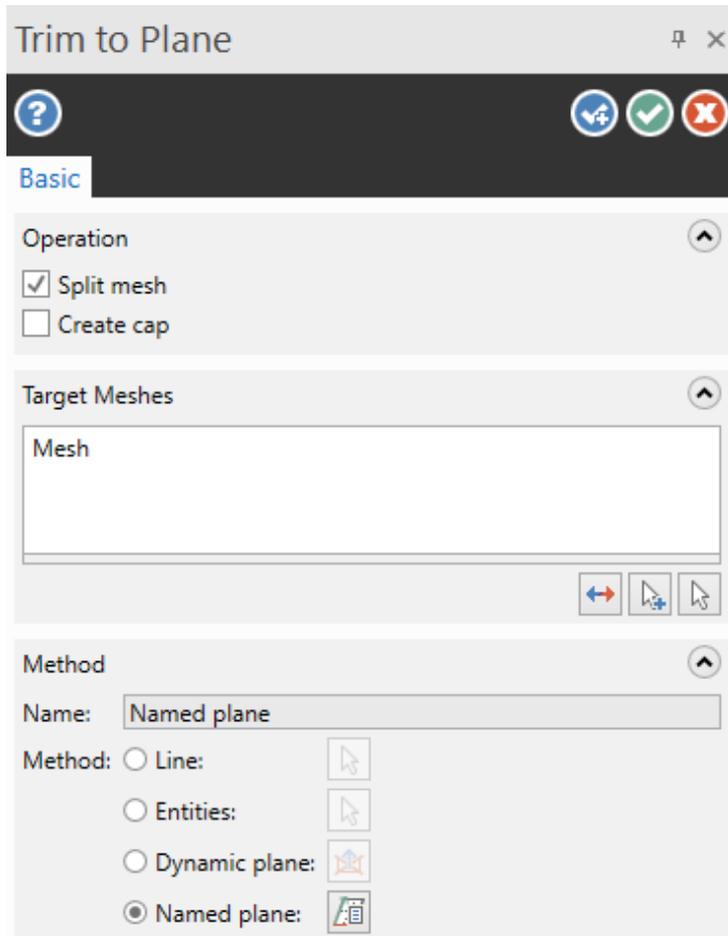
Simplifying Mesh Bodies

Often, imported or scanned mesh bodies have more information than necessary for use in Mastercam. This results in extreme file sizes and slower processing. The **Decimation** function reduces the number of facets in a mesh model while giving you control over how closely the modified mesh matches the original. This reduces the file size of the mesh entity.



Trimming Mesh Bodies

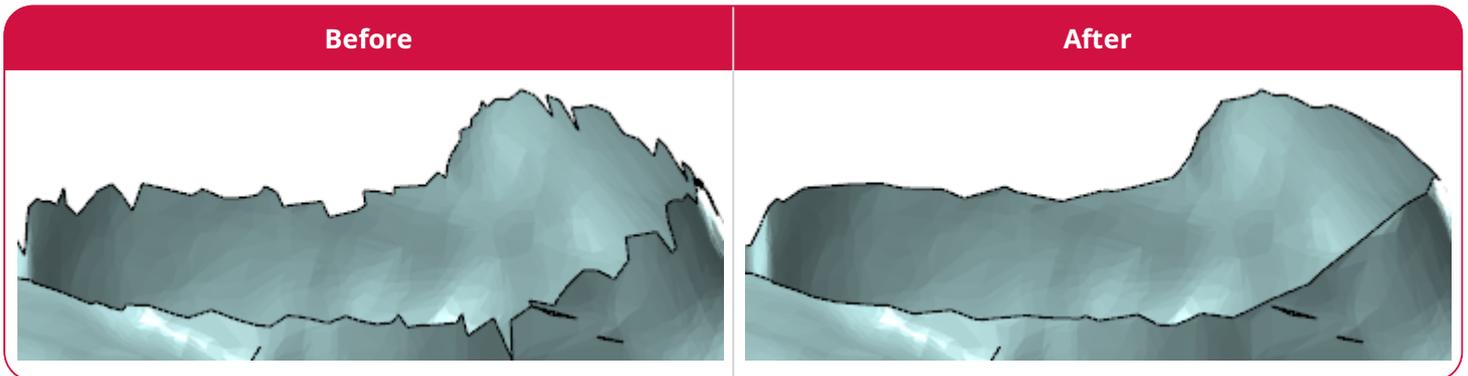
The **Trim to Plane** and **Trim to Surface/Sheet** functions remove unwanted areas in a model or split a mesh body. Choose a line, flat entity, plane, or surface to trim your selected mesh bodies. Unlike the solid functions, you can create a cap on each side of the split mesh or on the trimmed mesh entity that is left behind.



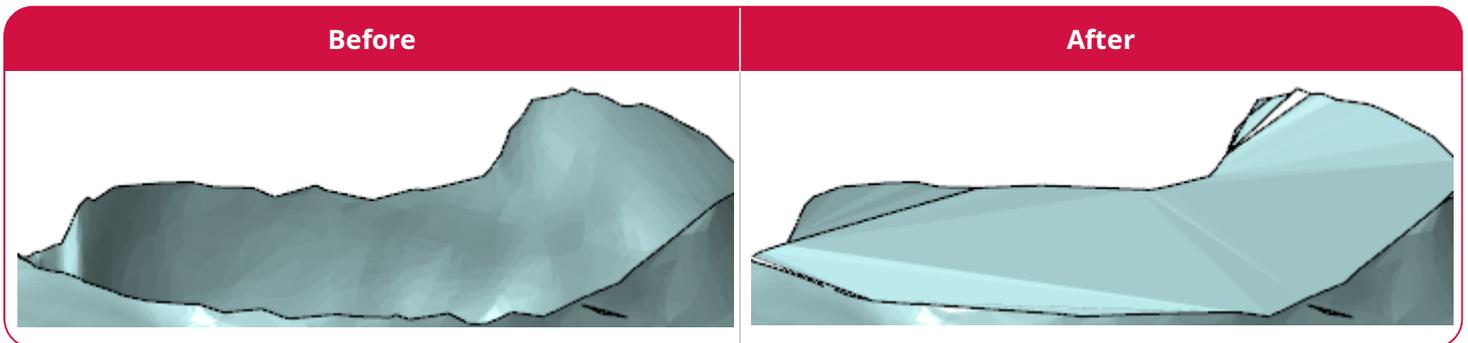
Filling Holes and Smoothing Mesh Edges

Often, you may create toolpaths on parts that use scanned or imported mesh data. The quality of the mesh data varies and may need improvement before working in Mastercam. Mesh entities can have holes in areas missed by the scanner or probe, or have ragged edges. Use the **Fill Holes** and **Smooth Free Edges** functions to improve the mesh data quality.

Use **Smooth Free Edges** to prepare a hole before using the **Fill Holes** function or an edge before sending the part to a 3D printer. It generates facets to fill-in any sharp edges that lie within a tolerance that you set. These facets smooth the edges of your mesh and improve your part for future applications.



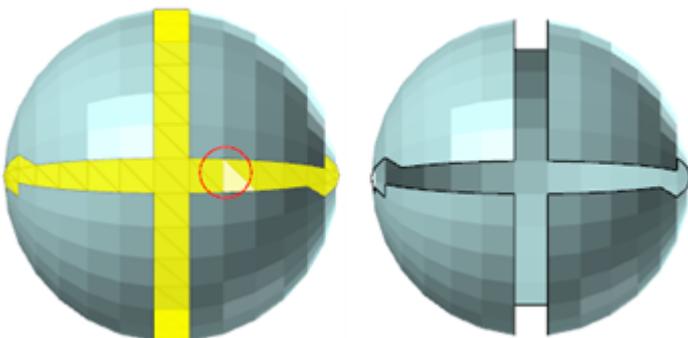
Fill Holes seals gaps and prevents tools on future toolpaths from plunging into holes in the entity. There are options to control the shape and construction of the mesh that is added to the selected entity.



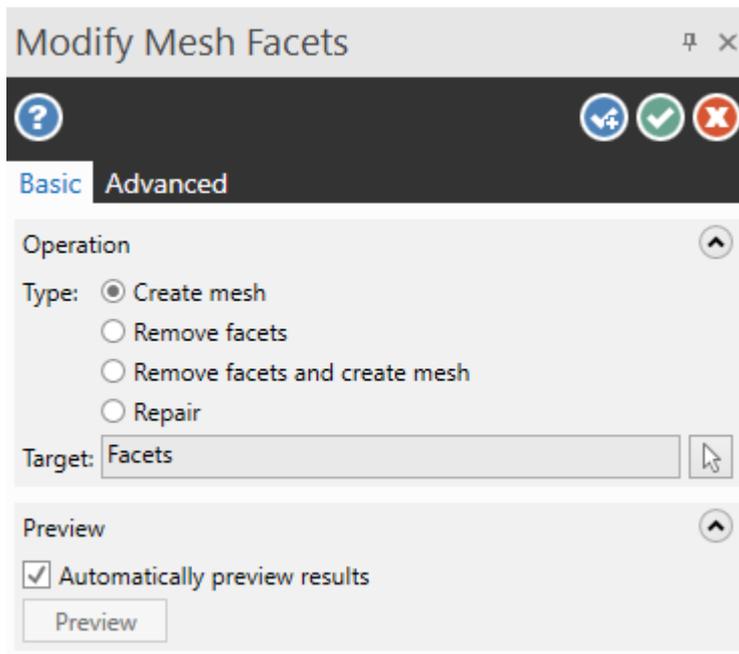
Both **Fill Holes** and **Smooth Free Edges** include automatic preview, as well as the ability to change the color of your results and/or place them on a selected level.

Modifying Mesh Facets

The **Modify Mesh Facets** function allows you to create or remove individual mesh facets.



This new function is particularly useful in helping correct errors found by the **Check Mesh** function. Use **Facet Selection** to choose the area of the mesh to work on. After you select the area, you can apply one of four modification methods.

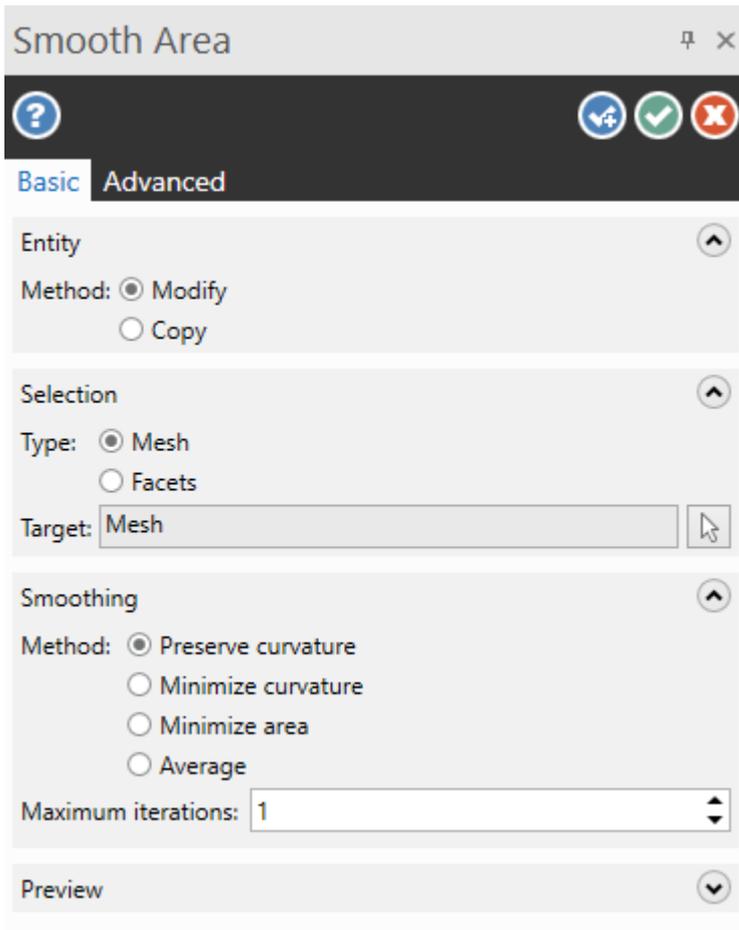


Smoothing Mesh Areas

The **Smooth Area** function prepares your mesh model for machining or 3D printing by smoothing out an entire mesh entity or a specific portion of the entity. **Smooth Area** can apply up to ten iterations of the following smoothing methods:

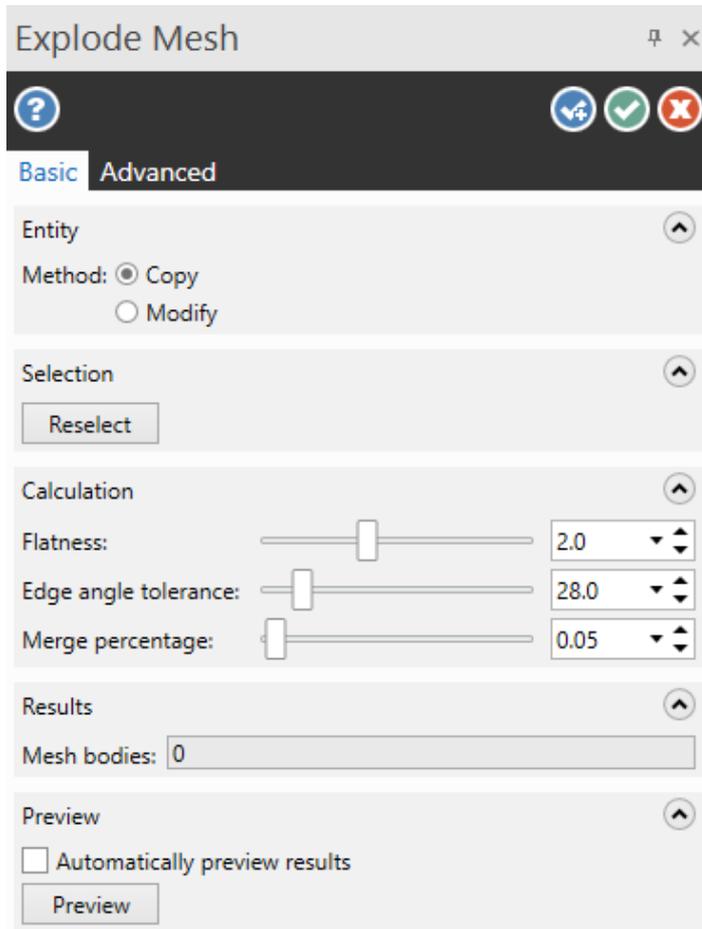
- **Preserve curvature:** Smooths with minimal changes to the curvature. This method can take the longest to process.
- **Minimize curvature:** Smooths by flattening the neighboring facets.
- **Minimize area:** Smooths by reducing the overall area of the mesh. This method is the most drastic and can reduce the volume of the mesh significantly.
- **Average:** Smooths by applying the neighbor's average vertex position to each vertex. This method is the quickest to process.

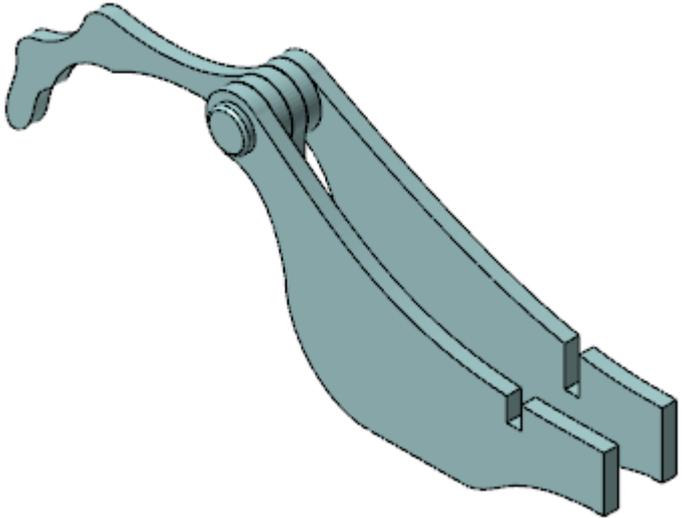
Smooth Area does not change the number of facets in a mesh, but modifies the positions of the facet vertices to reduce irregularities between facet vertices.

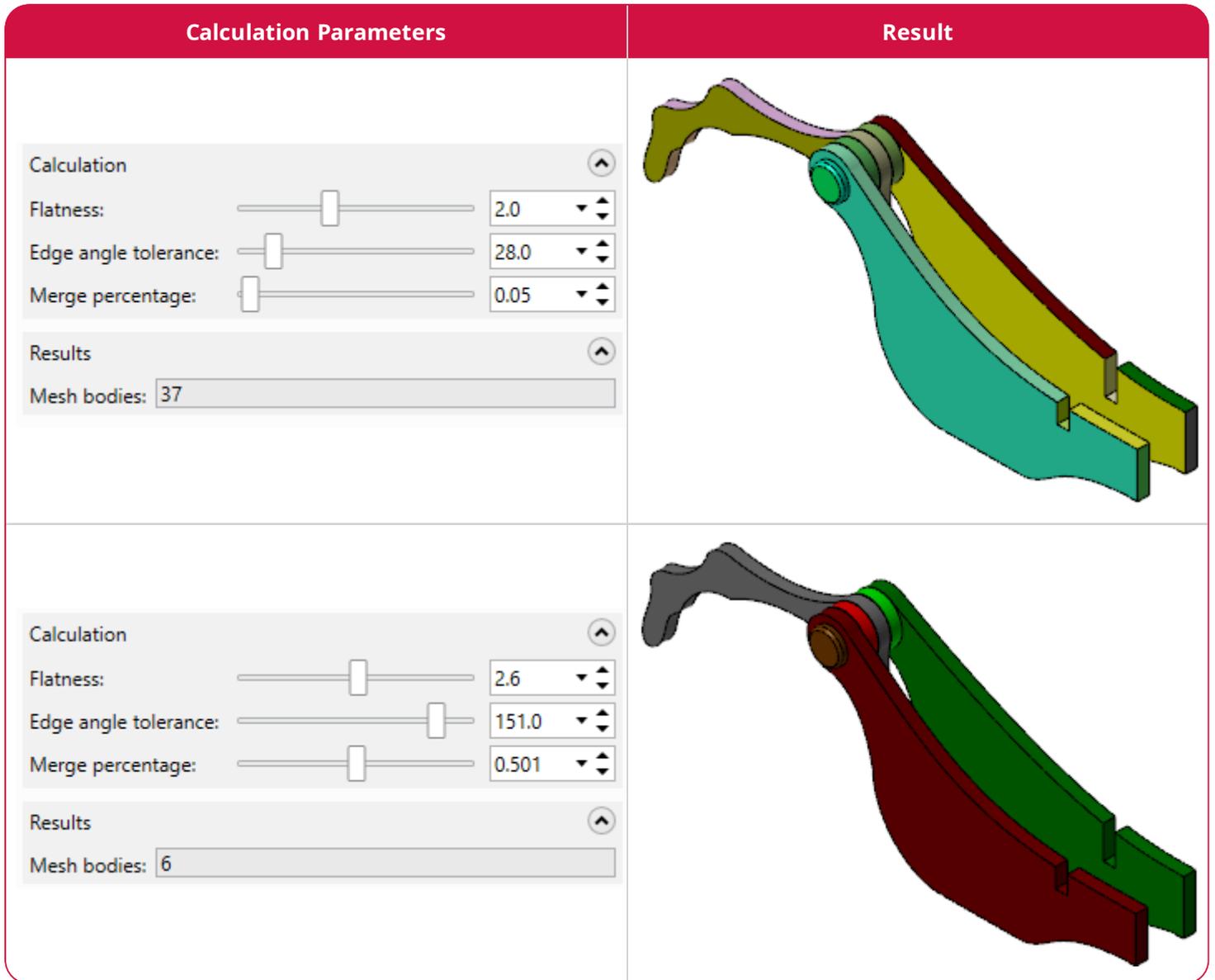


Exploding Meshes

When working with complex mesh models, it may be difficult to work on a specific part or area. The **Explode Mesh** function attempts to separate multi-part mesh models into separate mesh bodies. The number of resulting bodies is determined by the parameters set in the function panel.

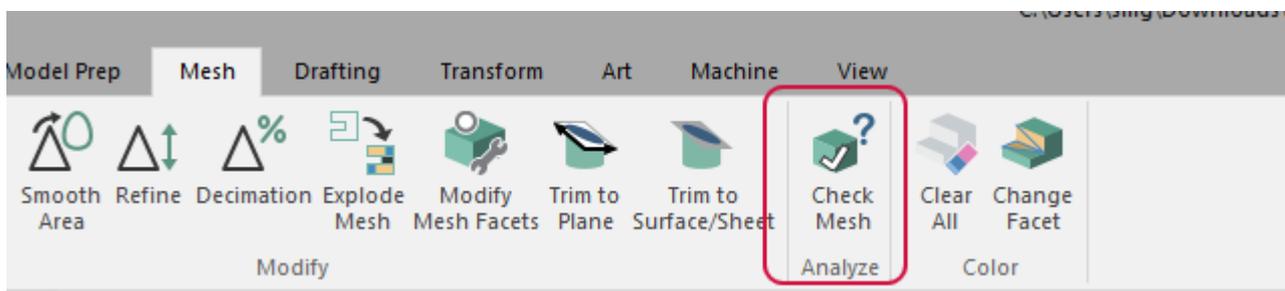


Calculation Parameters	Result
None.	

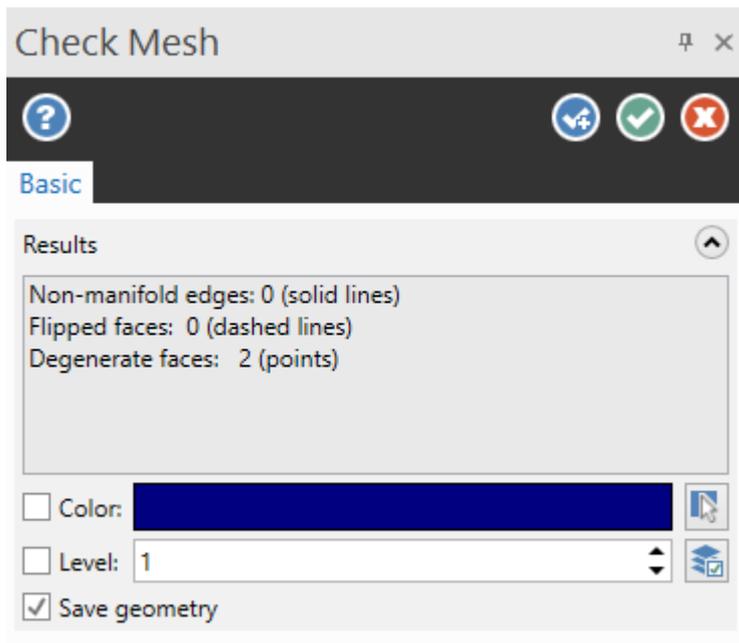


Identifying Errors in a Mesh Model

The new **Check Mesh** function allows you to identify three types of error conditions in mesh models. This new function is found on the **Mesh** tab or the **Home** tab in the **Check Solid** drop-down.

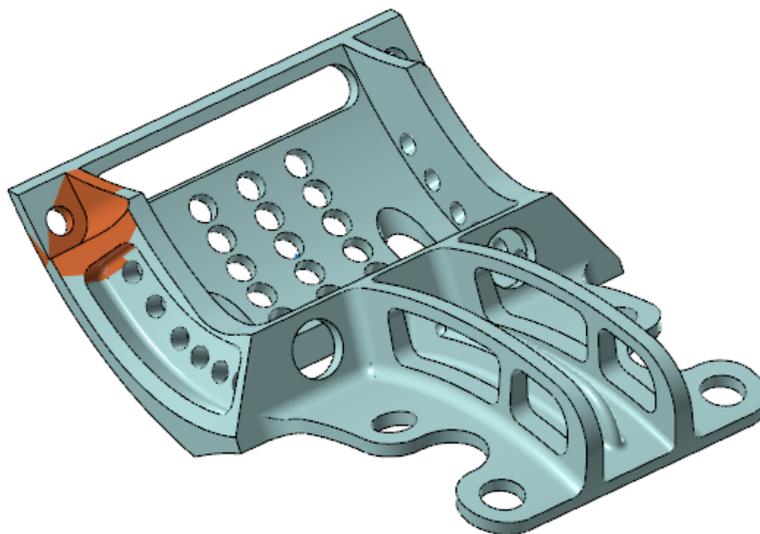
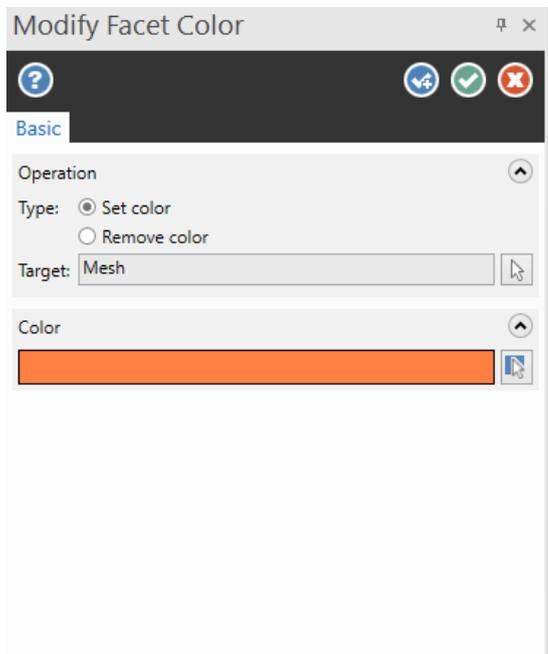


After starting the function and selecting your mesh in the graphics window, Mastercam lists the following issues in the **Results** list box and displays the entity types in parenthesis to mark each error.

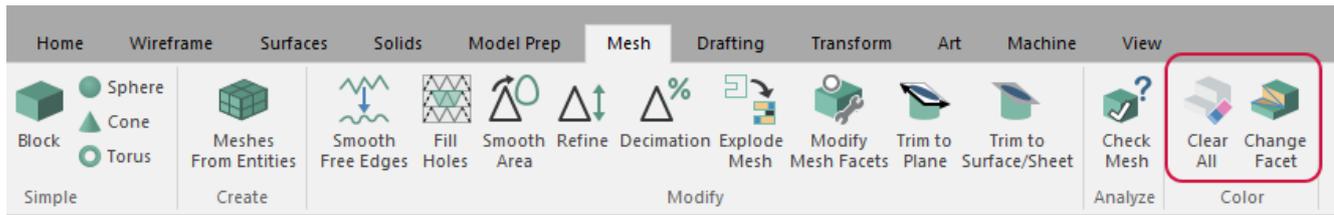


Adding or Removing Colors on Mesh Facets

In addition to the functions that support, modify, and improve mesh entities, Mastercam 2022 provides tools in the **Color** group to support polychromatic meshes. The **Change Facet** function allows you to color individual facets of a mesh using a different color than the mesh entity.



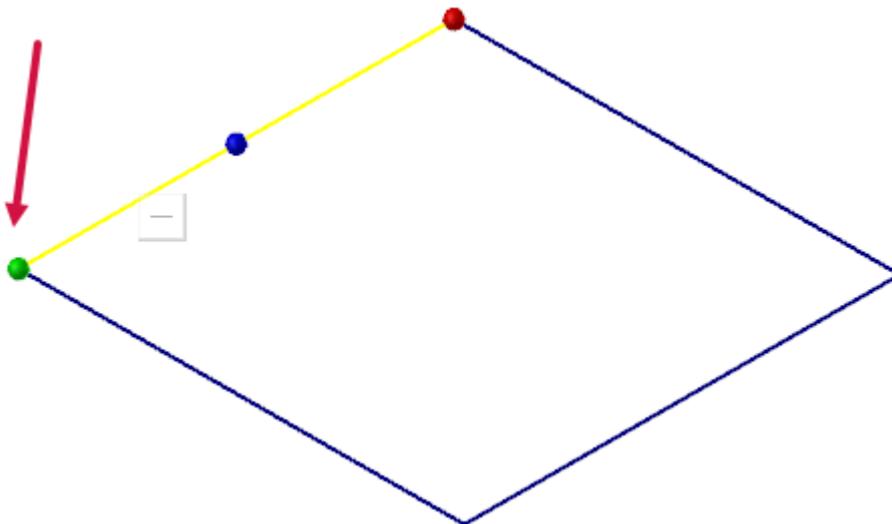
The **Clear All** function works on mesh facets exactly as it has worked on solids in previous versions of Mastercam. **Clear All** resets all facets to their original colors.



Performing Top-level Editing for Wireframe Entities

In previous versions of Mastercam, you needed to use the **Analyze** function to edit the values and properties of a wireframe entity by interacting with the dialog box and the screen. In Mastercam 2022, you can directly edit wireframe geometry without interacting with the dialog box. If no other function is active, double-click the lines, arcs, points, and splines to directly modify the entities in the graphics window with the following controls.

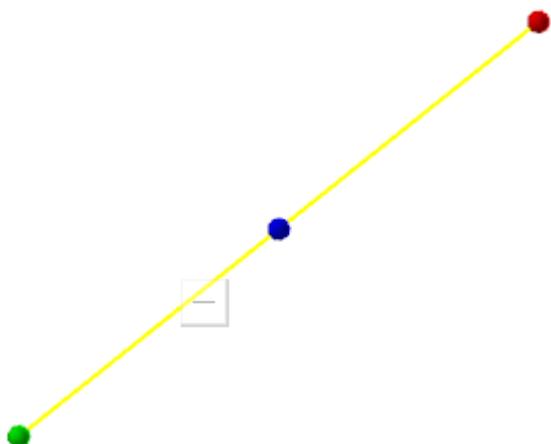
Editing with Point Controls



- Click and drag the green origin ball to move the point.
- Use the on-screen button toggles between move and copy modes.

Editing with Line Controls

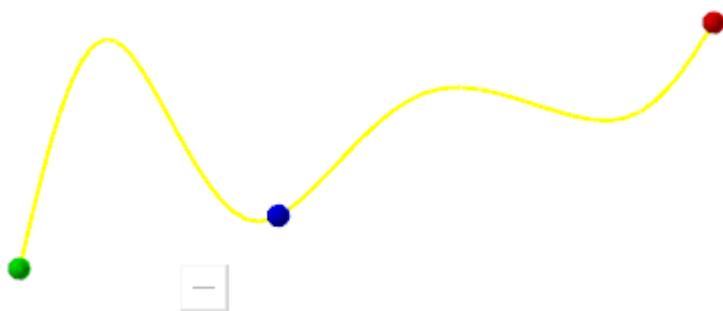
Three position spheres display when you select a line.



- Use the center sphere to reposition the line, maintaining its current length and angle.
- Hover over the midpoint and select the arrow to create an offset parallel line.
- Grab an endpoint to change the angle of a line.
- Hover over an endpoint and drag the arrow to extend or shorten the line.
- Toggle between move and copy using the **Mode** button. To make more than one copy, select the original again.

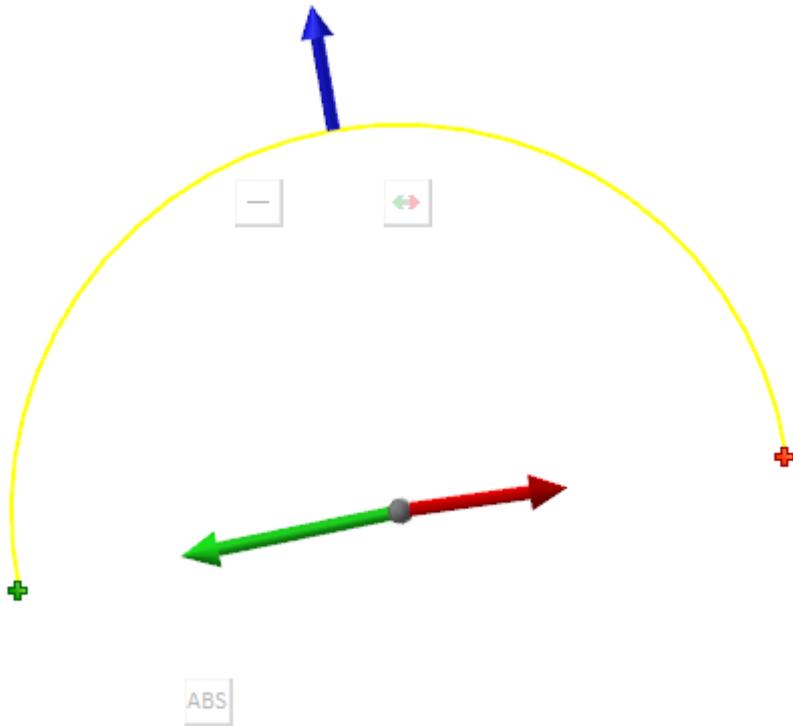
Editing with Spline Controls

Spline controls are the same as the line controls except that you cannot use top-level editing to change the curvature of a spline. You cannot shorten splines in this mode, but you can increase their length using a linear tangent extension. Use the **Edit Spline** function to do this.



- Use the spheres to reposition the spline, maintaining its current length and angle.
- Hover over the endpoints and select the arrow to extend the length of the spline. It cannot be shortened.

Editing with Arc Controls



- Use the origin ball to edit the center point position.
- Use the blue arrow to edit the radius. The value appears when you hover over the arrow.
- Use the green arrow to edit the starting sweep angle of the arc.
- Use the red arrow to edit the ending sweep angle of the arc.
- Arcs have two mode buttons in addition to the **Move/Copy** button.
 - Use the **Flip** button to invert the arc to display its complement.
 - Use the **ABS/REL** button to toggle the display of the sweep angle value between its absolute and relative values.

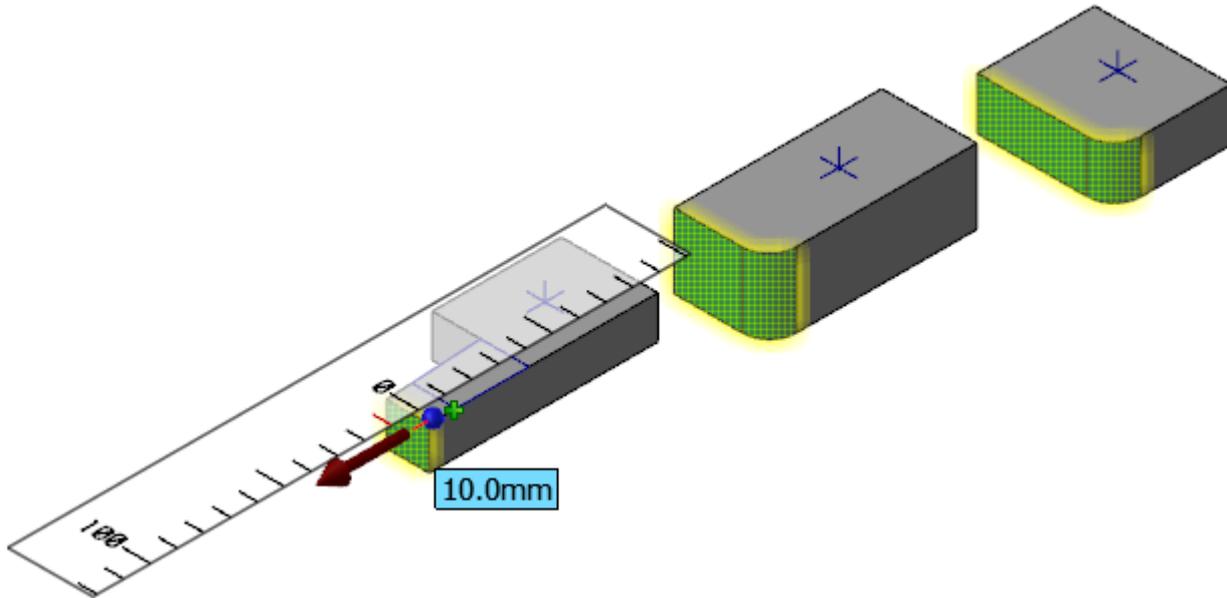
You cannot edit multiple entities with top-level editing. To do that, use the **Analyze** function. To display the **Analyze** dialog box from the top-level, press the **[End]** key. You can continue to use the new graphic controls as well as change entity attributes such as level, color, and line style. Press the **[End]** key again when you are finished.

Model Prep Enhancements

Listed below are enhancements to the functions on the **Model Prep** tab.

Applying Push-Pull Modifications to Multiple Bodies

Mastercam's Push-Pull function allows you to dynamically modify solid faces and edges. In Mastercam 2022, you can now apply your push-pull modifications to multiple, independent bodies in a single operation. Making your edits in one operation saves time and eliminates inconsistencies between modifications.

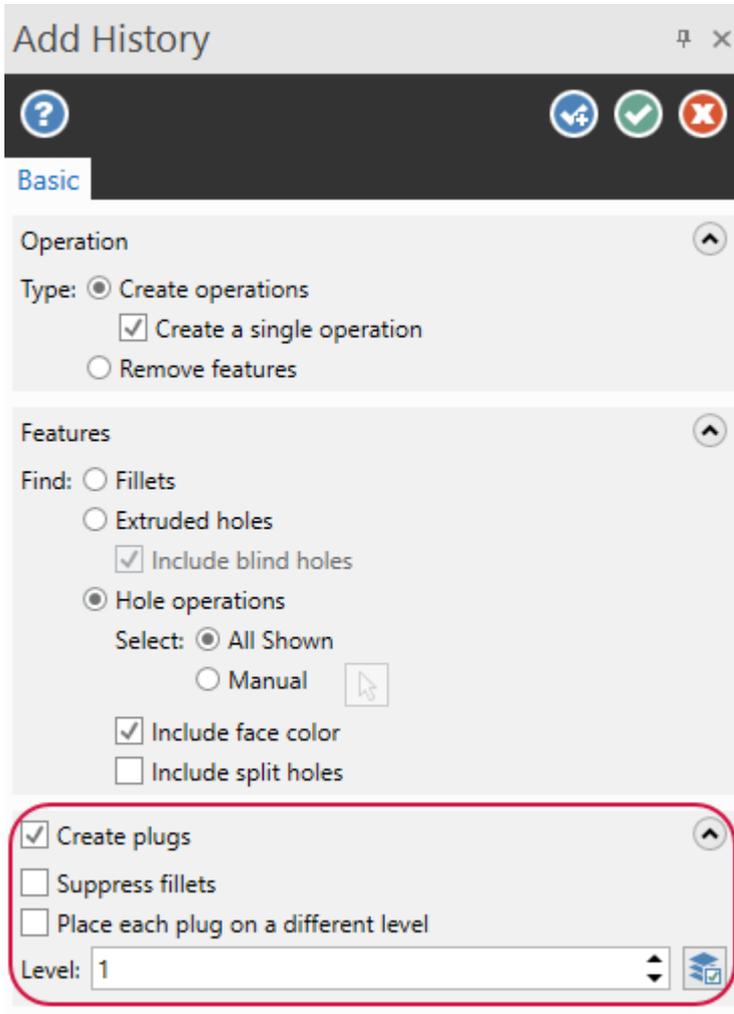


To take advantage of this new functionality, simply select the faces, edges, surfaces, or chains you would like to modify and enter a delta value, or drag the entity to your desired result.

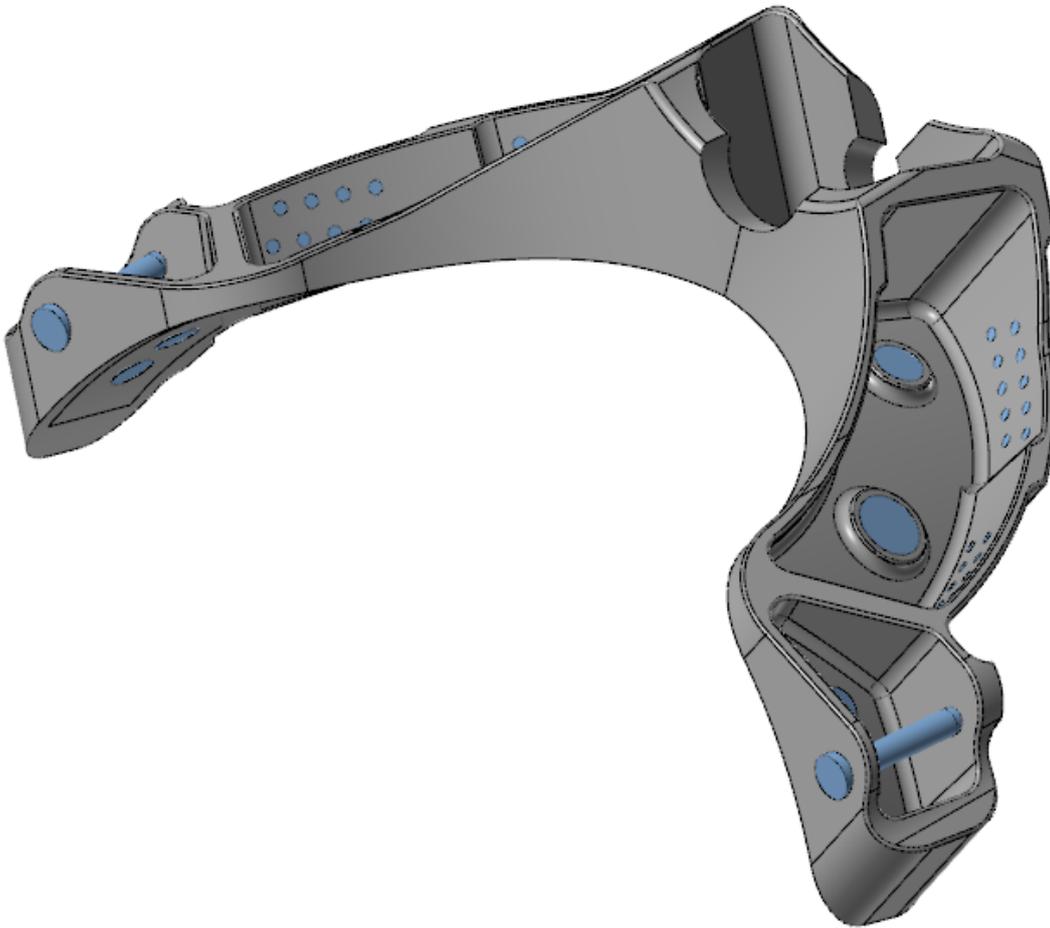
After you make your edits, Mastercam marks any associated toolpaths as dirty in the Toolpaths Manager and updates any dimensions associative to the solid.

Recognizing Interrupted and Intersecting Holes

In Mastercam 2022, Add History includes improvements to recognizing interrupted and intersecting holes. These include intersections, interruptions, and dimensional configurations that make it difficult for Mastercam to generate Hole operations when they are detected. Add History now includes a **Create plugs** option that creates independent solid bodies (plugs) where the problematic holes are located.



You can choose to create the new bodies on a different level than your part. You can then view them separately from the part to determine how the holes intersect. In the example below, the created plugs are colored light blue.



Improved Selection for Hole Axis

After listening to customer feedback, we have made Hole Axis selection more powerful and discriminating. Previously, when you used the [Ctrl+click] method to select faces on tapered holes, Mastercam would select the faces that matched the taper angle. However, this could result in unintended selections. In this release, using the same selection method, Mastercam matches the top and bottom diameters of the taper in addition to the taper angle so you only get what you intended to select.

Two additional new selection methods have been added:

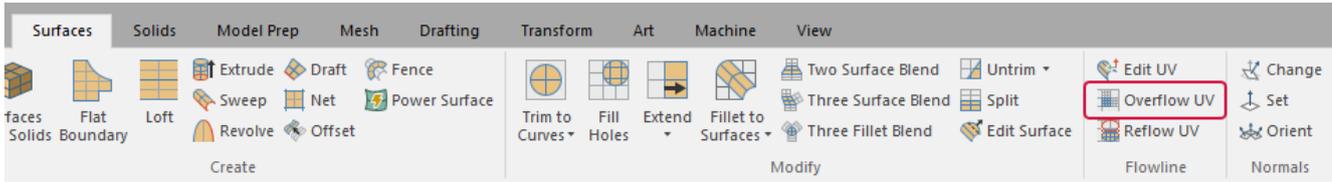
- Double-click a face to select an entire hole.
- Double-click a face while pressing the [Ctrl] key to select all the holes in a Solid Hole operation.

Surface Enhancements

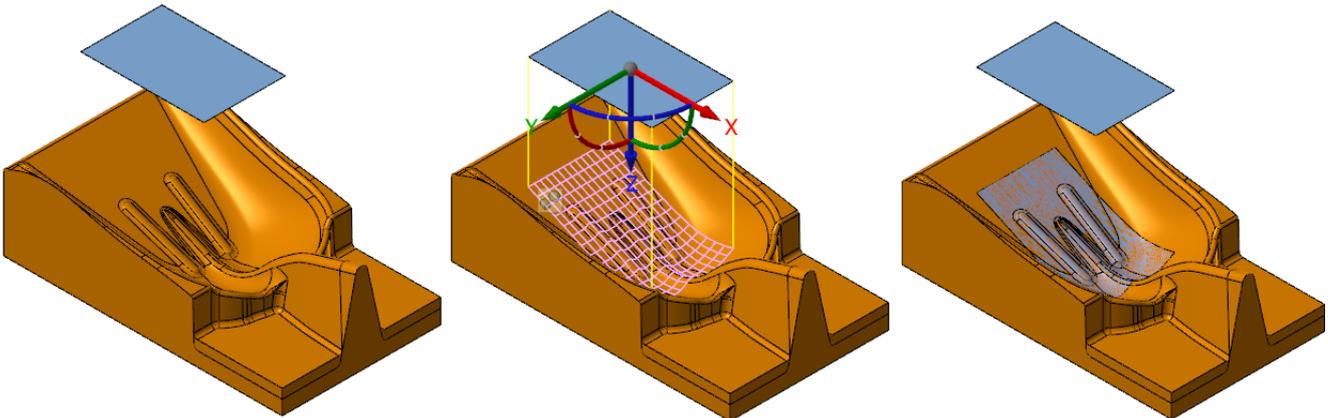
Listed below are enhancements to the functions on the **Surfaces** tab.

Creating and Controlling Flowlines with Overflow UV

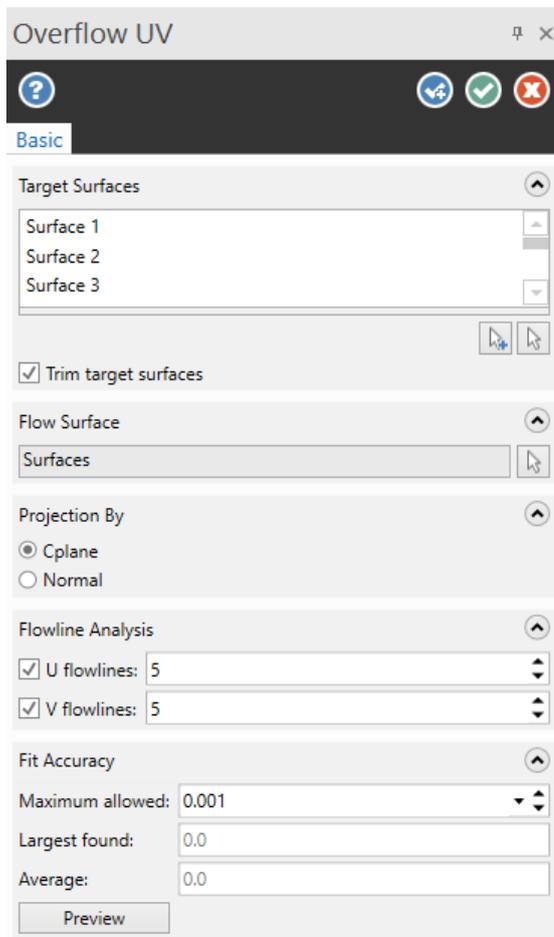
Mastercam 2022 introduces the new **Overflow UV** function. Located on the **Surfaces** tab in the **Flowline** group, **Overflow UV** generates a single unified surface with a desired UV flow on a connected set of surfaces. It allow you to replace multiple surfaces with inconsistent flows with a single surface for use with flowline toolpaths. You can also simply surfaces for general modeling.



Select **Overflow UV** from the **Surfaces** tab and select your flow surface. After confirming your selection, **Overflow UV** generates a single NURBS surface from your targets, covering small gaps and holes. By default, the function trims the target surfaces to the boundaries of the overflow surface. If you wish to keep the set of surfaces, use the **Trim target surfaces** option to project a new surface upon the set.



The function panel allows you to change the visualization of the UV flowlines and how the flow surface is projected on to the target surfaces. By default, the function uses the current construction plane (Cplane) to project the flow surface onto the targets.



Creating Surfaces in Notes

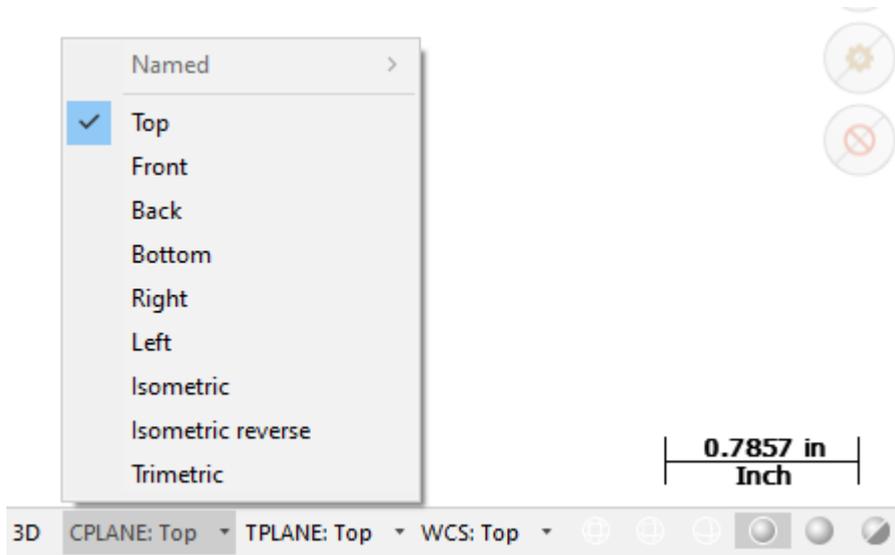
You can now create a surface inside a note without breaking it into geometry. Keeping the note intact allows you to edit it later.

Mastercam

After creating a note using a closed TrueType font, such as Arial, use the **Flat Boundary** function on the **Surfaces** tab or the **Cross Hatch** function on the **Drafting** tab to create surfaces or a hatch pattern. The note and the created surfaces are not associative; changes to one do not result in changes to the other.

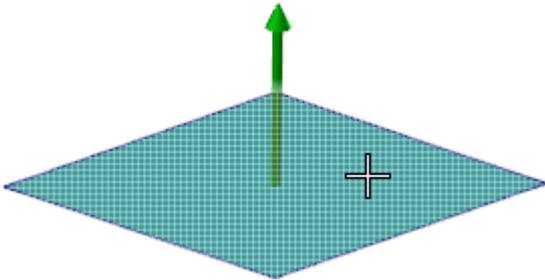
Creating Draft Surfaces in Multiple Planes

When creating draft surfaces, you can now change the active construction plane without exiting the function. In previous versions, the Cplane must be set in advanced. Now while in a dimension function, change the Cplane using the **Status Bar** at the bottom of the graphics window. The Cplane you select remains active until your change it.



Flipping Surface Normals Using On-screen Controls

When creating a surface, its normal will display in the graphics window when you hover your cursor over it. Click the surface to reverse the normal direction. In previous versions, normals were determined by the chaining direction. With Mastercam 2022, you do not have to re-chain or use the normal tools to correct your work.



Wireframe Enhancements

Listed below are enhancements to functions on the **Wireframe** tab.

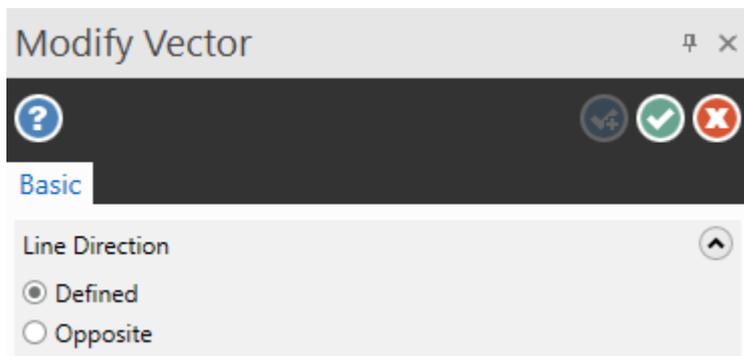
Enhancements to Support Multiaxis Machining

Mastercam 2022 includes enhancements to Wireframe functions that will simplify creating and editing multiaxis toolpath geometry. **Line Normal** is now **Line Normal to Point** and three new functions are available:

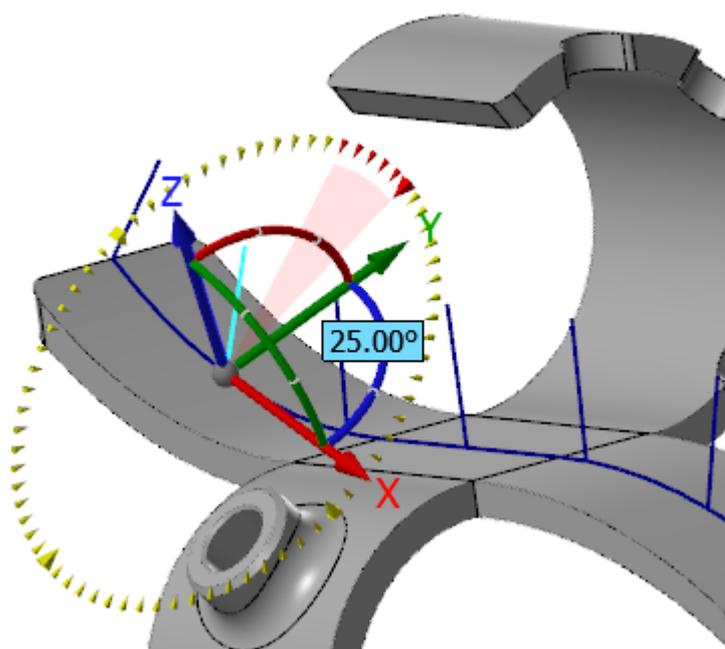
- Modify Vector
- Line Normal to Grid
- Line Normal Along Chain

Editing Tool Axis Control Lines

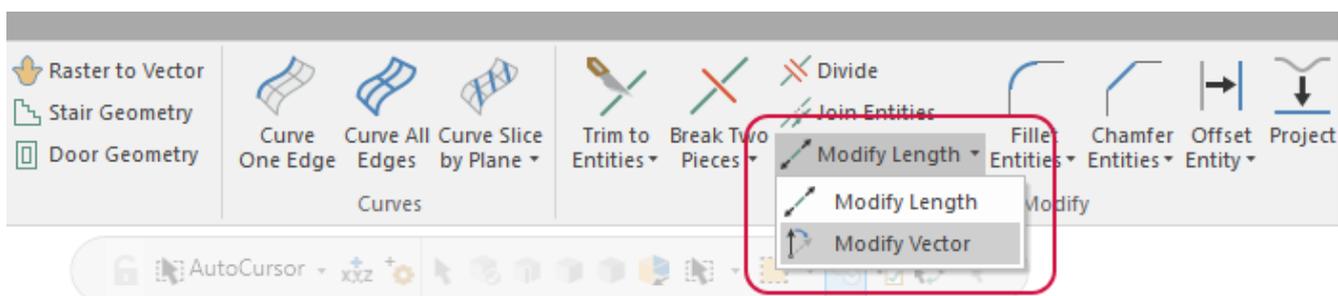
The new **Modify Vector** function makes editing tool axis control lines easier, saving you from the multi-stepped, manual editing that previous versions of Mastercam required.



Simply start the function and select a line. Then, use the Dynamic Gnomon to alter the tilt of the vector along the XY, XZ, or YZ axes. When possible, the gnomon's X axis aligns with U of the surface or the construction plane. The function is especially useful to adjust tool axis control lines so that the tool can avoid collisions with clamps and fixtures on current or future multiaxis operations.

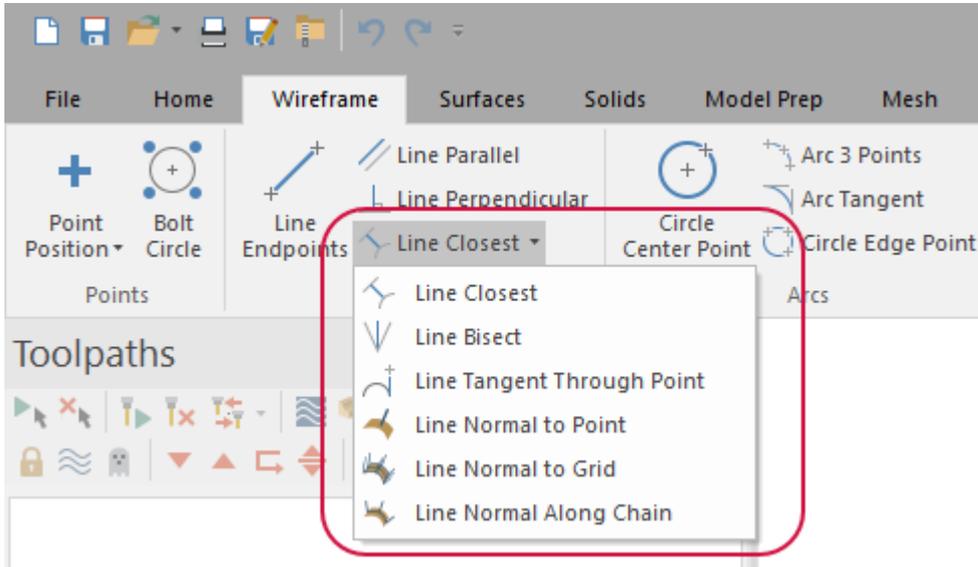


This new function is on the **Wireframe** tab, under the **Modify Length** drop-down.

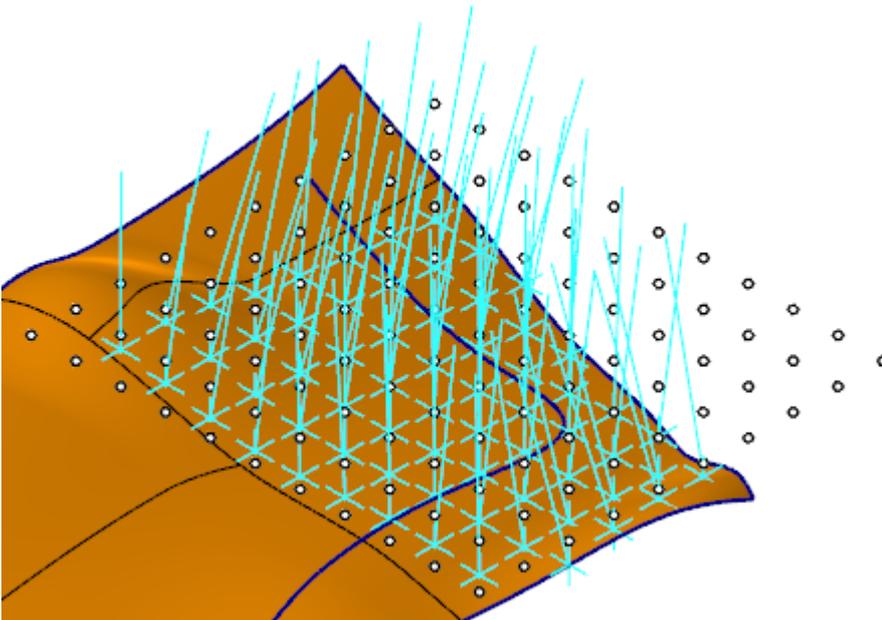


Expanded Line Normal Functions

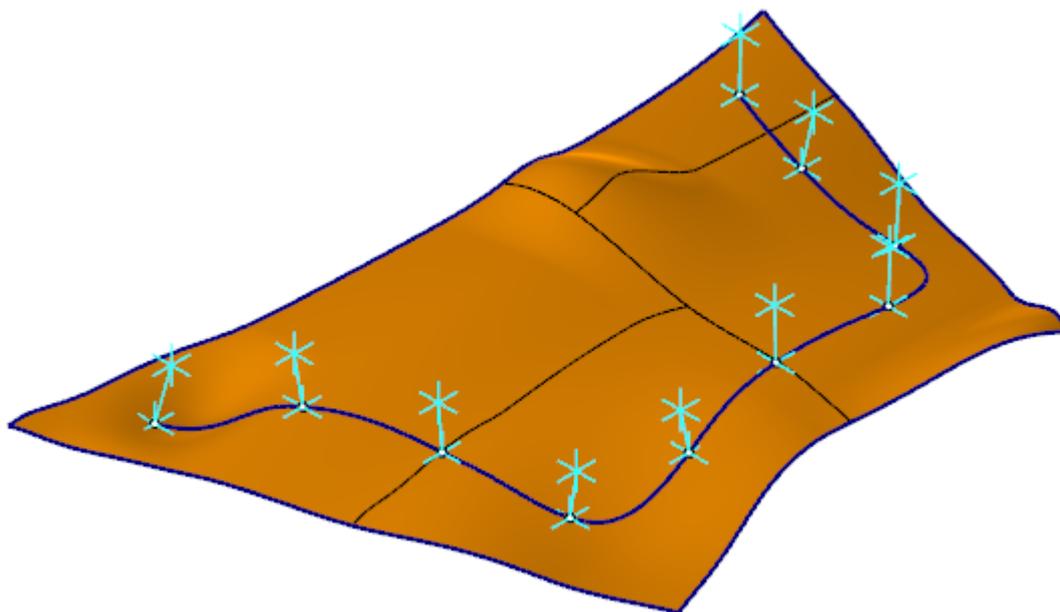
The **Line Normal** function has been expanded to give you multiple ways to add normals to your geometry. These functions simplify the creation of tool axis control lines that are required for multi-axis machining. You can find the following functions under the **Line Closest** drop-down in the **Lines** group.



Line Normal to Grid creates a grid of lines that are normal to your selected surfaces, solid faces, meshes and open sheet bodies. Choose a start point (the grid origin) and use the parameters in the function panel to control length, point creation, and spacing of the lines generated from the grid. **Line Normal to Grid** replaces the need for the GridPock add-in.

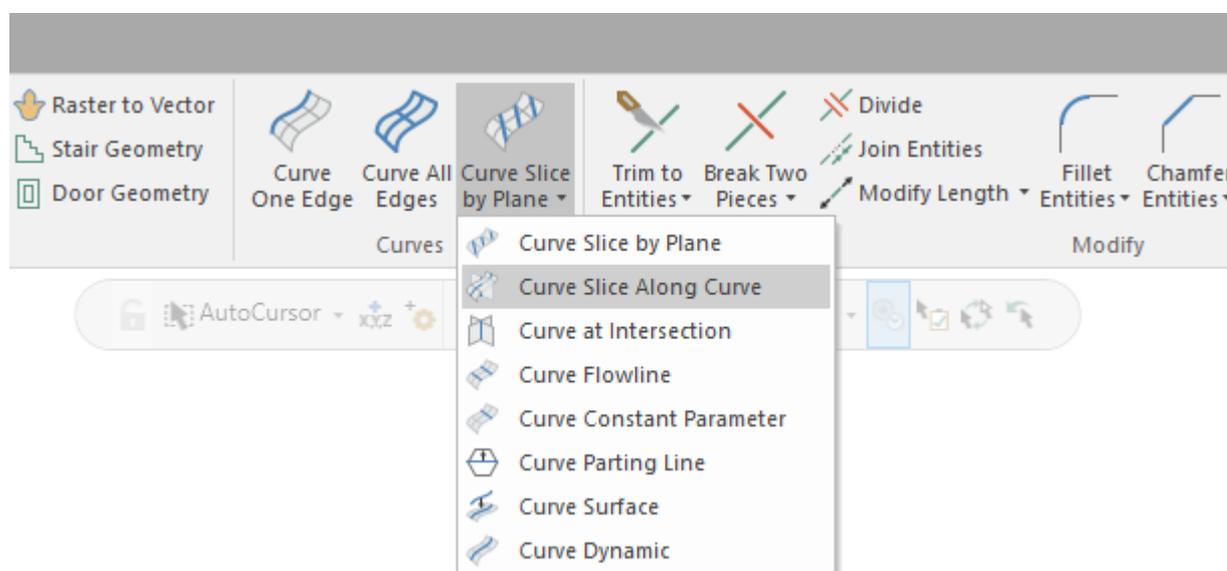


With **Line Normal Along Chain**, you use Mastercam's Chain Manager to select and generate lines on surfaces, solid faces, meshes and open sheet bodies. As with **Line Normal to Grid**, you can use the parameters in the function panel to control length, point creation, and spacing of the lines generated along the chain.



Creating Cross-Sections Along a Curve

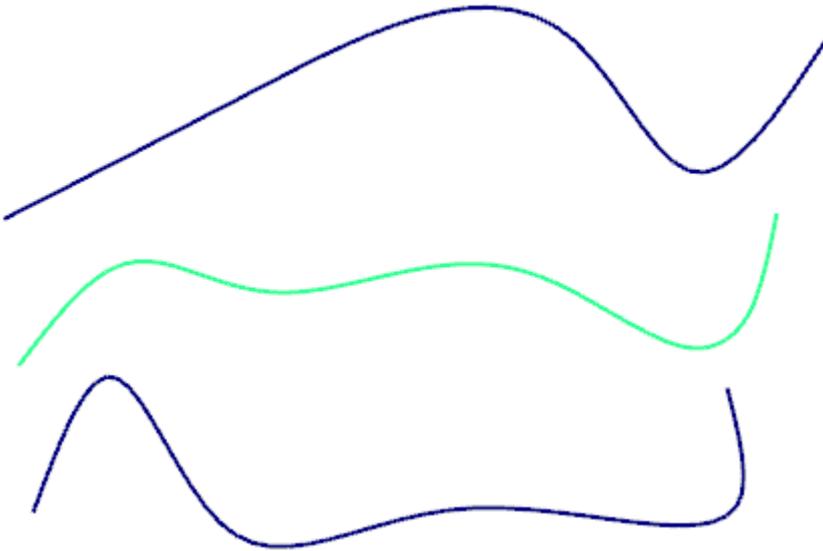
The new **Curve Slice Along Curve** function creates wireframe slices through surfaces, solids, and meshes along a selected chain. This new function is located on the **Wireframe** tab in the **Curves** group under the **Curve Slice by Plane** drop-down.



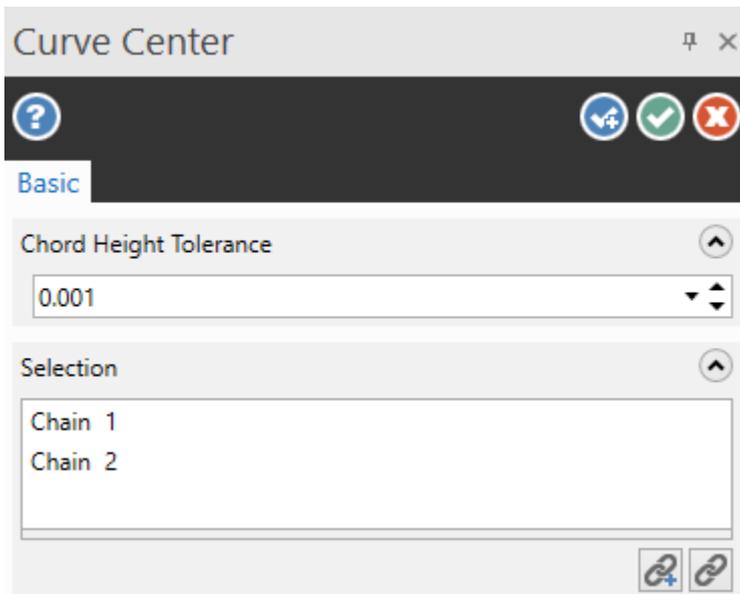
Use **Curve Slice Along Curve** on low-quality, imported or scanned mesh data to help build surfaces and solids. In contrast to the **Curve Slice by Plane** function which creates wireframe that is parallel to an axis, when the **Along curve** is at the midline of the object, the cross sections **Curve Slice Along Curve** produces remain perpendicular to the chain. This generates wireframe that more accurately represents the curvature of the selected body.

Creating a Midline Between Wireframe

The new **Curve Center** function builds a curve at the midline between two or more selected chains. This function is located on the **Wireframe** tab, in the **Curve Slice by Plane** drop-down.

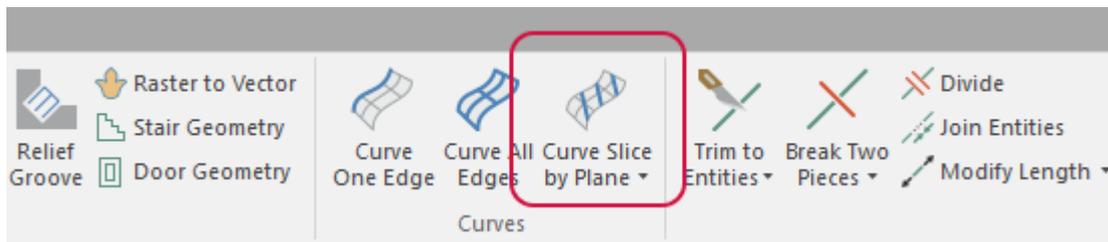


In the **Curve Center** function panel, right-click in the **Selection** list to reverse, remove, add, or reselect your chains. Reversing the direction of a chain will change your results. Additional chains and chains at multiple Z depths will also affect the shape and location of the result.

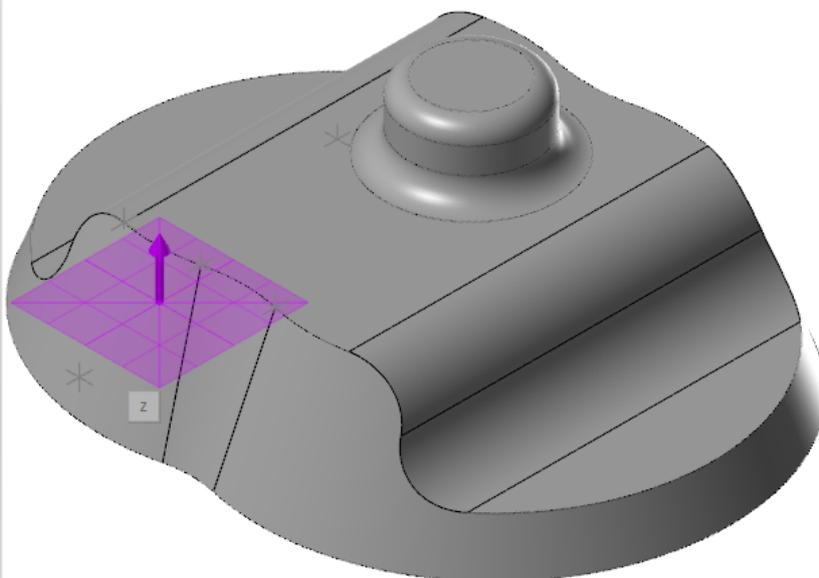
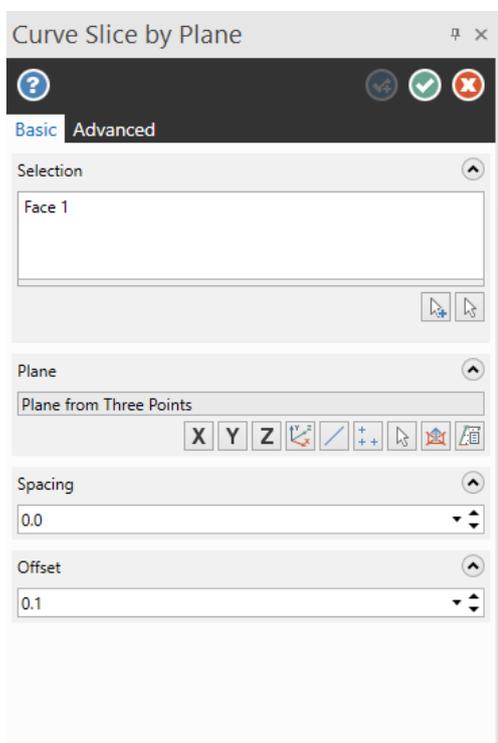


New Name and Interface for Curve Slice

To distinguish it from the new **Curve Slice Along Curve** function, the **Curve Slice** function has been renamed to **Curve Slice by Plane**. This function retains all of its past functionality, but in Mastercam 2022, it comes with a new, more intuitive interface.



The panel now includes a selection list box to make adding and removing your selections easier and integrated plane selection controls. Additionally, you can drag the on-screen arrow to change the position of the defined plane and a new mode button moves the plane to the current Z value as defined in the **Status Bar**.



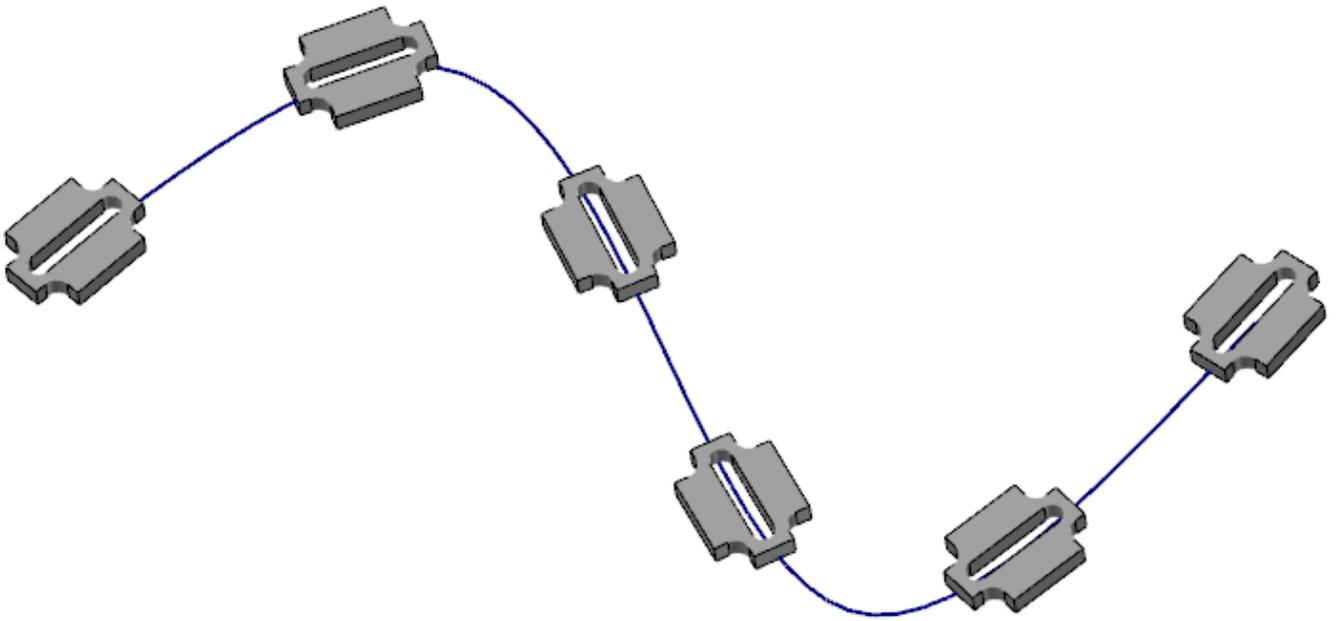
Transform Enhancements

Listed below are enhancements to the functions on the **Transform** tab.

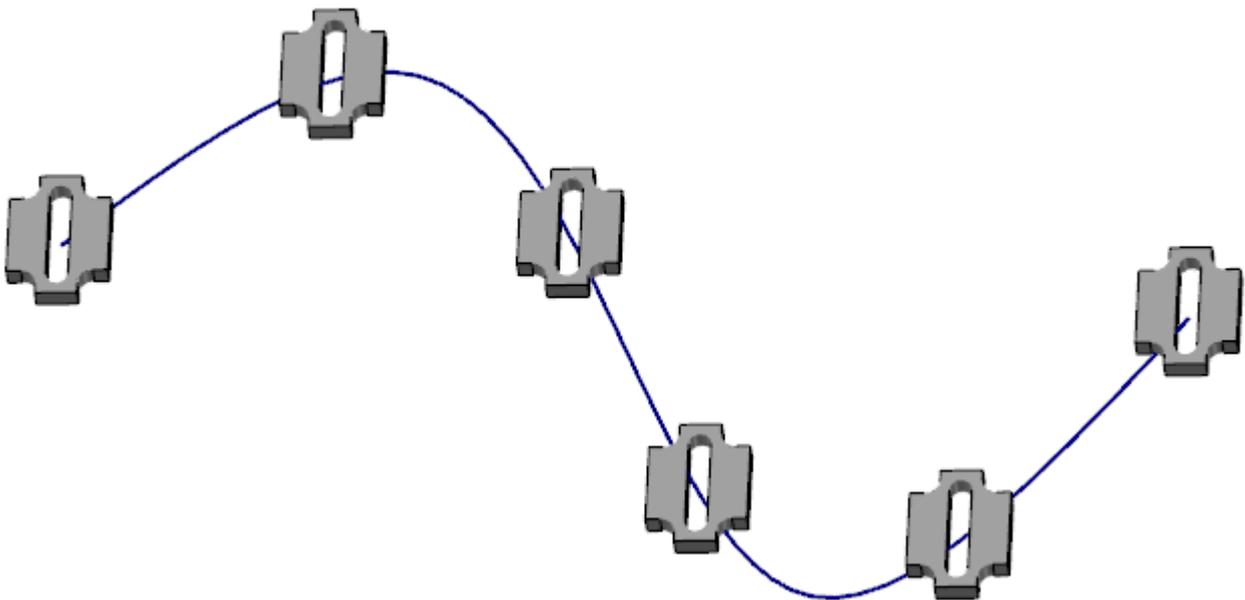
Distributing Entities Along a Chain

In previous versions, Mastercam's **Distribute** function was limited to creating multiple copies of geometry between two defined points along a vector. In Mastercam 2022, you can now distribute these copies along a chain or multiple chains. Use the parameters in the function panel to arrange the copies. Use the two new **Chaining type** options in this panel to distribute your geometry along a chain.

- **Rotate:** Copies of the geometry change orientation as they are distributed along the chain.

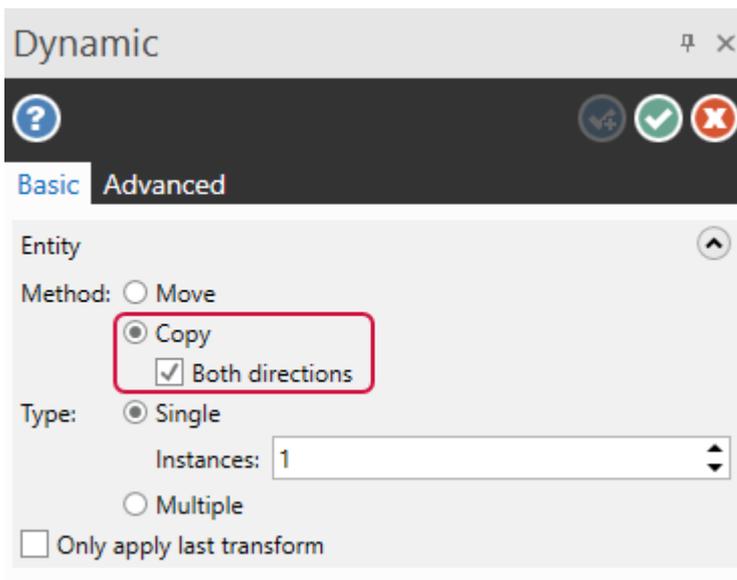
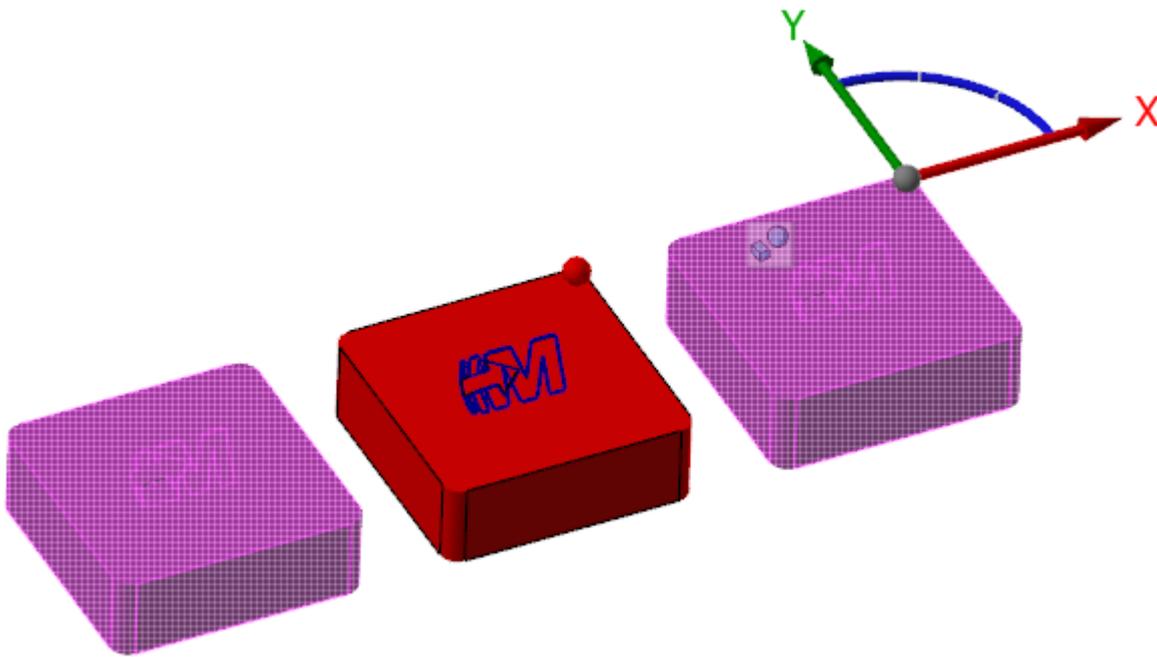


- **Translate:** Copies of the geometry keep the same orientation as the original.



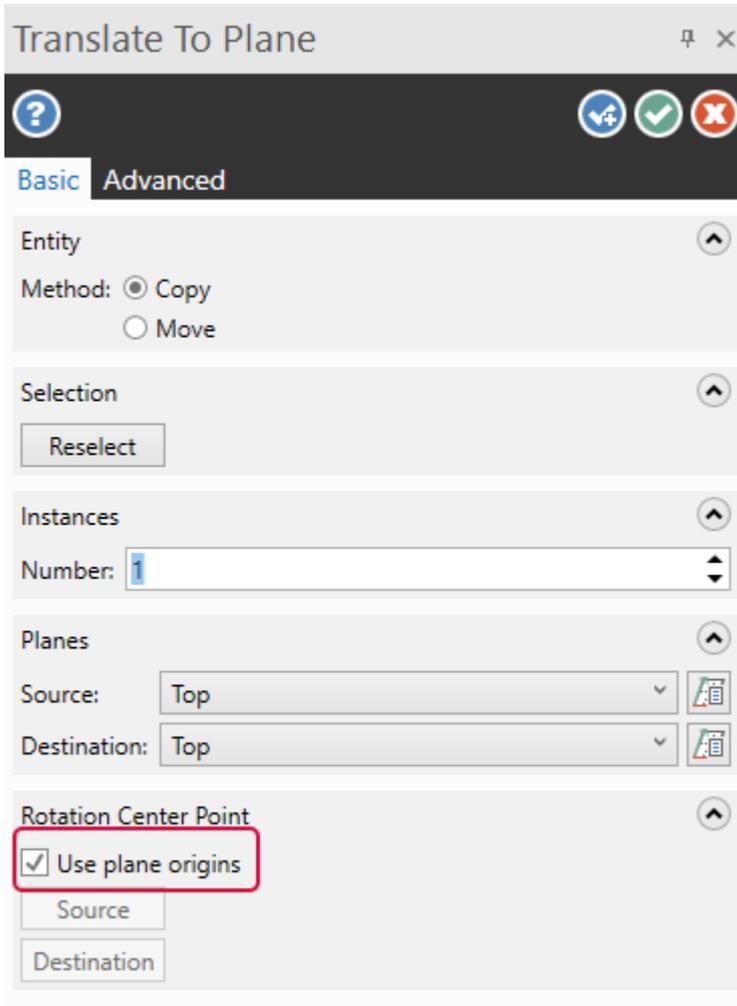
Copying Entities in Two Directions

When using **Dynamic Transform**, you can now copy entities in two directions along the axis or from the origin you specify using the new **Both directions** checkbox in the **Dynamic Transform** function panel.



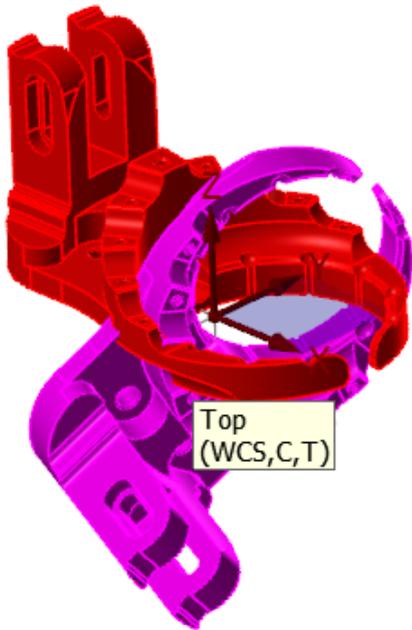
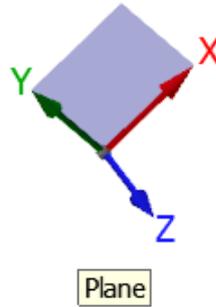
Rotating Geometry Around Plane Origins

Translate to Plane moves or creates copies of selected entities from one plane to another. In previous versions of Mastercam, unless you overrode it with the **Source** and **Destination** buttons on the function panel, the selected geometry would rotate around the World origin. In Mastercam 2022, **Translate to Plane** includes the new **Use plane origins** option that automatically rotates and places your geometry using the origins of the source and destination planes as reference points.

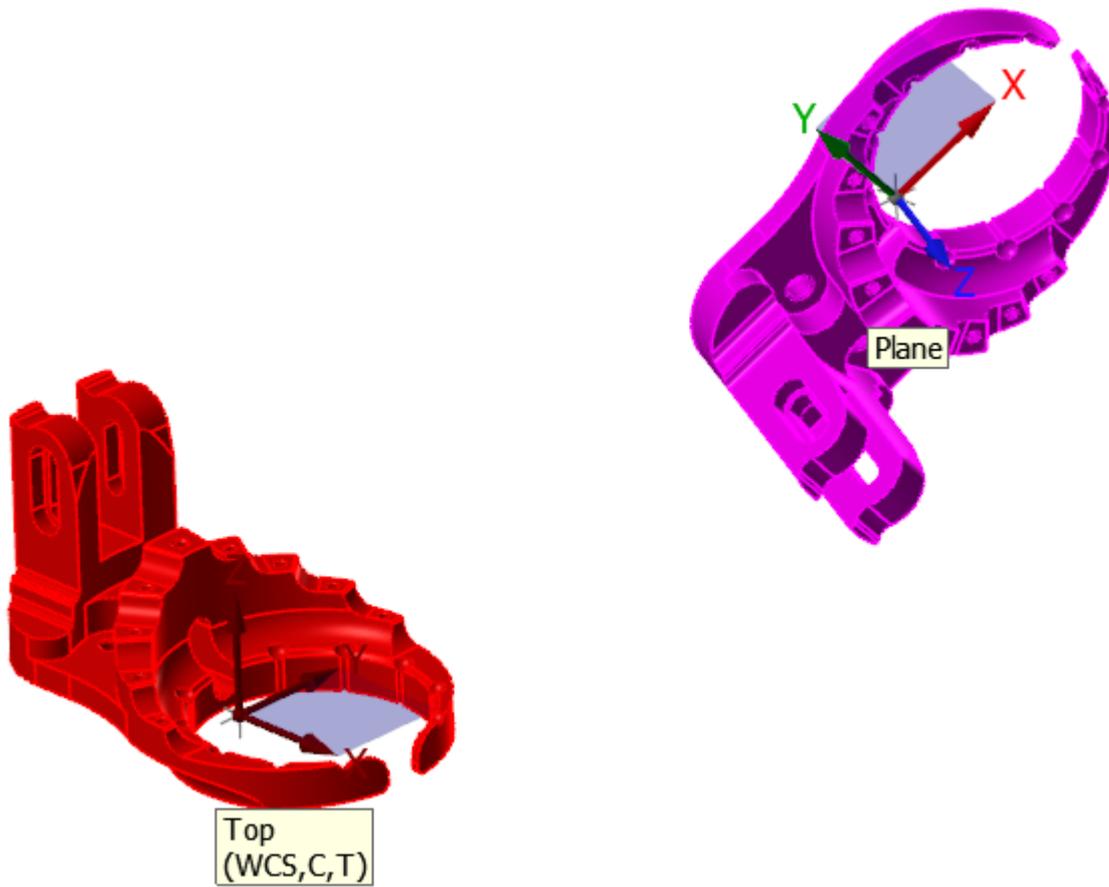


Select the **Use planes origins** option to keep the translated geometry at the same distance and in the same orientation from the destination plane as it is at the source plane. You can continue to use the **Source** and **Destination** buttons if you would like to select specific rotation points on your part.

The example below shows a part translated using the Top plane as the **Source** and **Plane** as the **Destination**. **Use plane origins** is not selected.



The example below shows the same part, but **Use plane origins** is selected.



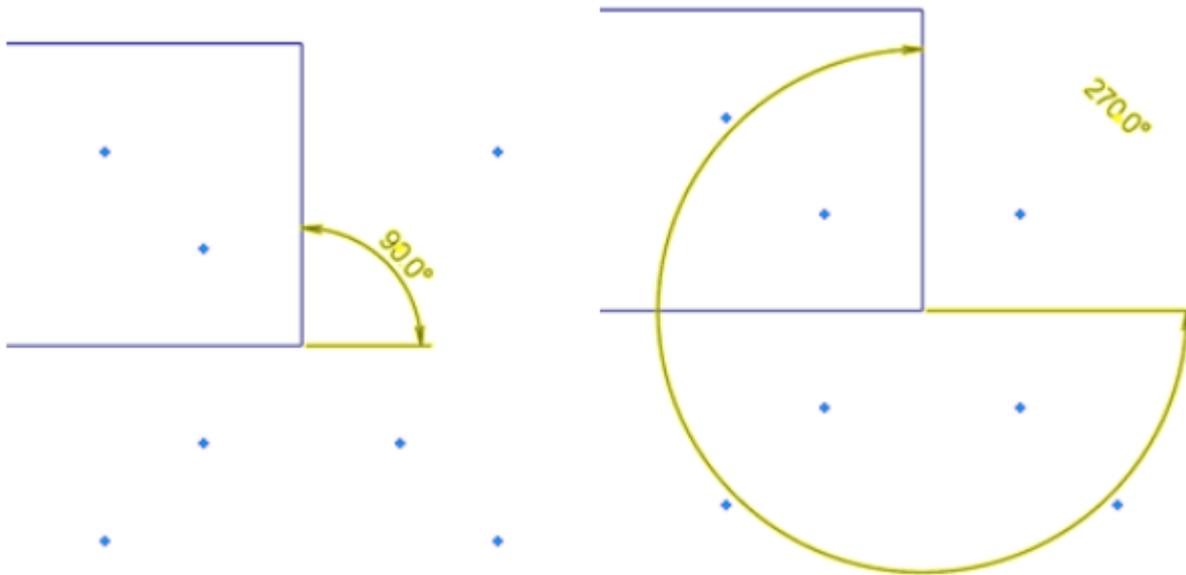
Dimension Enhancements

Listed below are enhancements to Dimension functions, located on the **Drafting** tab.

Create Conjugate Angle Dimensions with On-Screen Trigger Points

The conjugate angle is a pair of angles that sum to 360 degrees. In previous Mastercam versions, you needed to follow a multi-step process to dimension the conjugate angle. First, you had to create an angular dimension. Then, you used the **Angular sweep** option on the **Advanced** tab to find the conjugate.

In Mastercam 2022, the trigger point used for dimensioning arcs and circles can now be used to find the conjugate angle. When entering the **Drafting** function, eight triggers display at each quadrant of the potential dimension. Hover over the trigger points to select the dimension to you want to create. The inner trigger then snaps your dimension to the lesser angle, while the outer trigger snaps it to the conjugate.

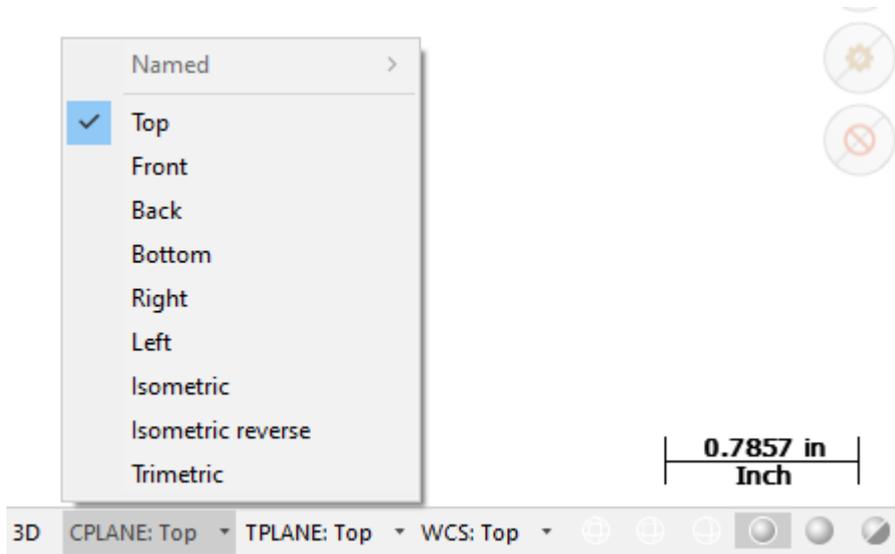


Associativity to Solid Geometry

In Mastercam 2022, drafting dimensions are now associative to solid geometry, whether or not the solid has a history.

Dimensioning Your Part in Multiple Planes

You can now create dimensions in multiple planes without exiting the function. To do this, select a Cplane from the **Status Bar** at the bottom of the graphics window while dimensioning. However, changing the construction plane only affects newly created dimensions. If you create a dimension in the Top plane and then select a new plane, the dimension is not updated.



The Cplane you select remains active until you change it.

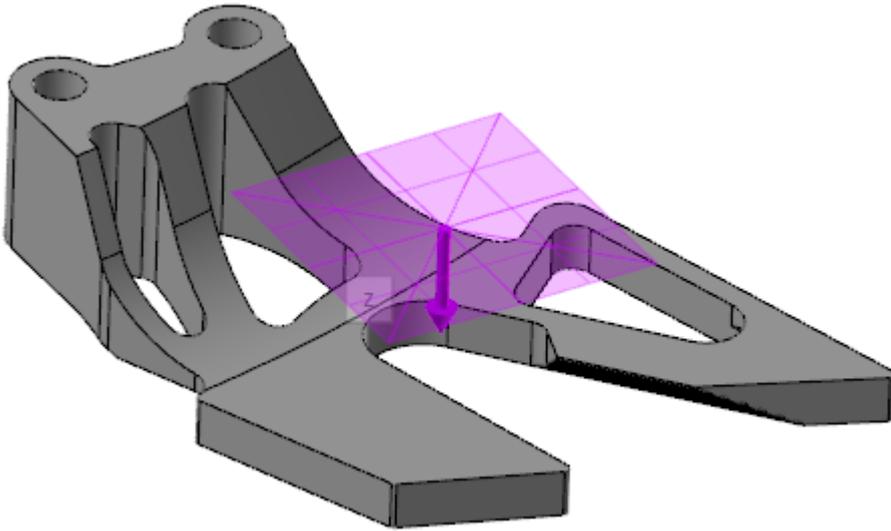
Better Cursor Control When Dimensioning Multiple Z Depths in 3D Mode

In previous Mastercam versions, the cursor stayed at the current Z depth while drafting, even if the entity being dimensioned was not at the same Z depth. This made it difficult to control the cursor.

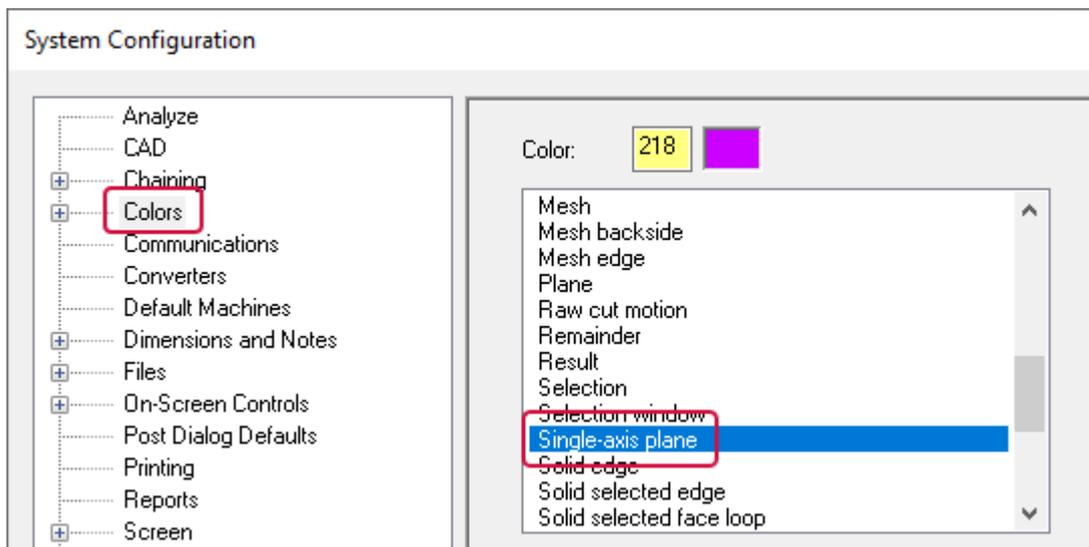
Mastercam 2022 temporarily sets Z to the depth of the entity you are dimensioning. This change allows the cursor and the dimension to track consistently. When you exit the function, the cursor resets to the current Z depth.

Using the Interactive Plane Depth Arrow

Mastercam 2022 allows you to edit plane depth when using the single-axis plane control in the Curve Slice to Plane and Trim to Plane. The plane depth arrow will snap to geometry as you pull it. Like other interactive graphics arrows in Mastercam, move your mouse over the ruler to snap to a value on the ruler's increments.

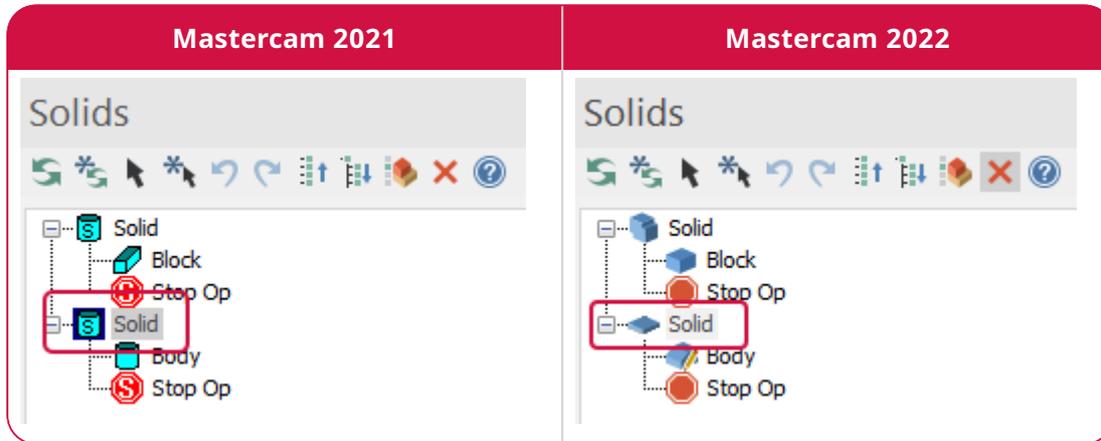


You can change the plane grid color by setting the **Single axis plane** option in the **Colors** page of the **System Configuration** dialog box.



New Solids Manager Icons

New Solids Manager icons contribute to a clean, modern look for Mastercam 2022. Sheet bodies also have a new icon, easily identifying them in the Solids Manager. In previous releases, you needed to analyze the sheet body to distinguish it from a solid body.

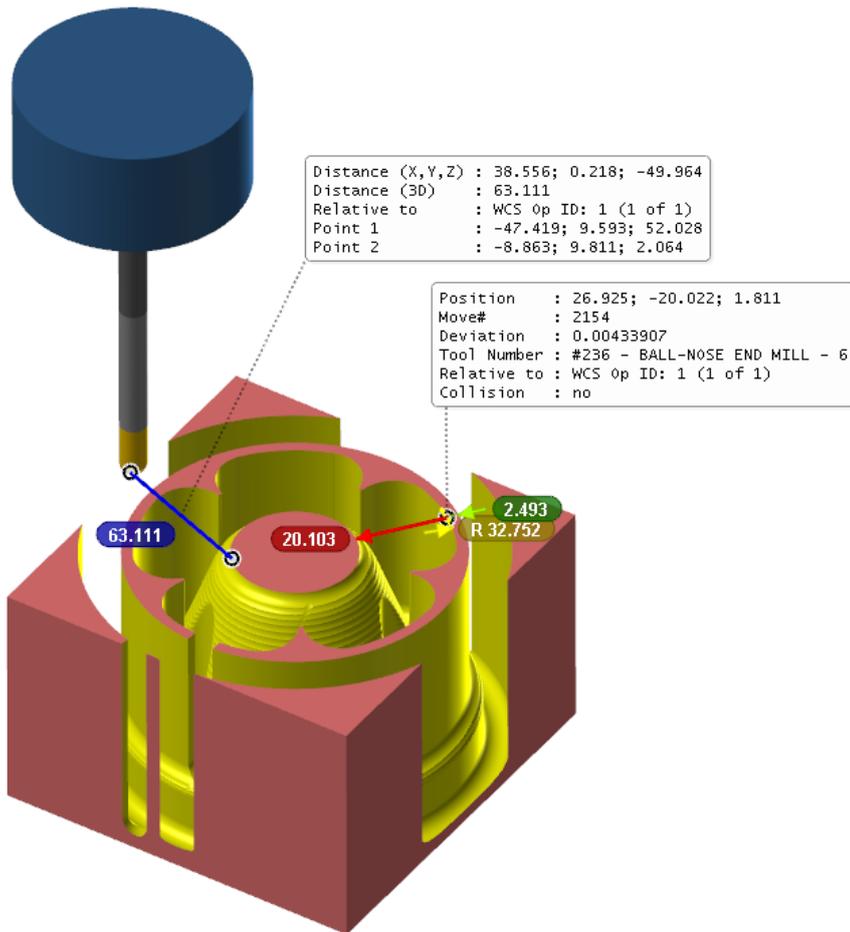


SIMULATION ENHANCEMENTS

Listed below are major enhancements to Mastercam Simulator.

Analyzing Distances Using Smart Measuring

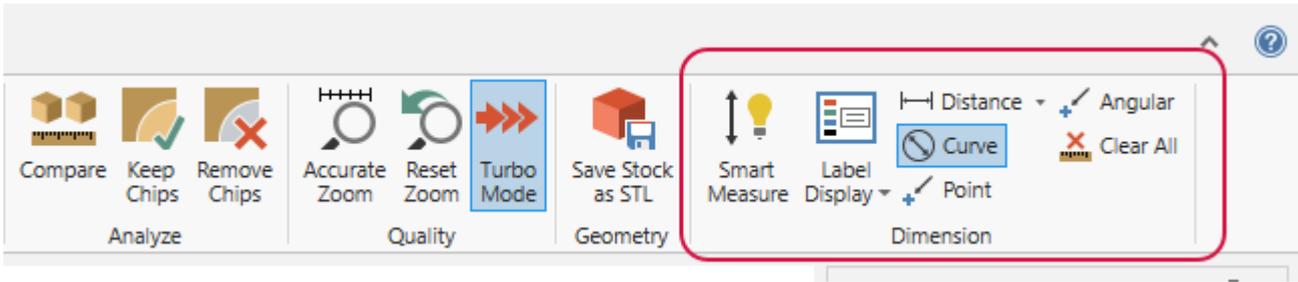
Mastercam Simulator includes the new Smart Measure feature, allowing you to quickly and accurately measure distances in the graphics window during simulation.



With this new feature, you can measure the following:

- Lengths and distances by snapping to an edge or corner.
- Multiple lengths and distances by snapping to an edge or corner.
- Depths and clearances from a surface.
- A radius with radius snapping.
- The radius and depth of a hole using hole circle edge snapping.
- Center-to-center distances between holes using hole circle center snapping.
- Edge angles using edge snapping.

Smart Measure is on the **Simulation** contextual tab in Mastercam Simulator.

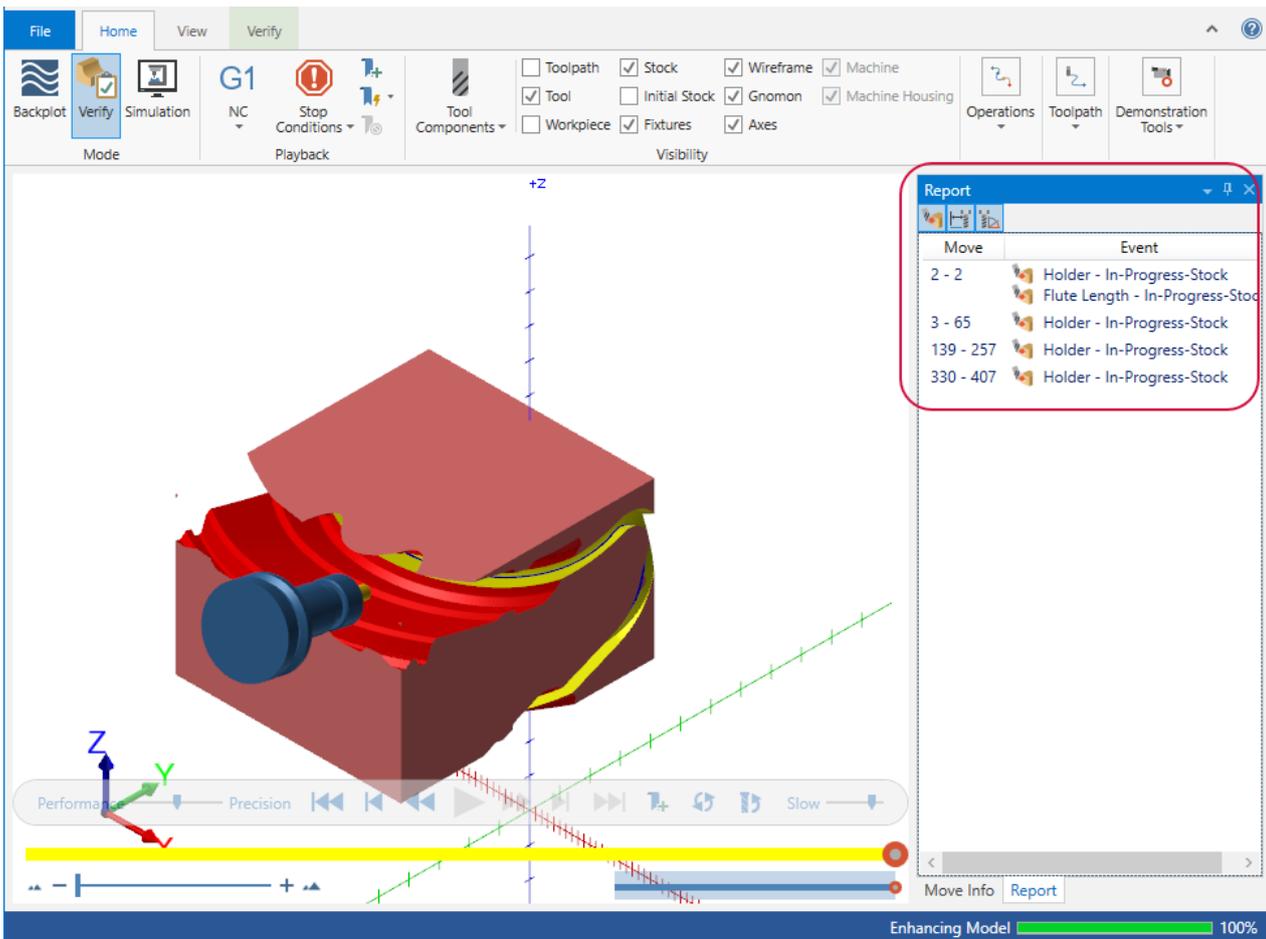


Improved Toolpath Processing Time for Mastercam Simulator

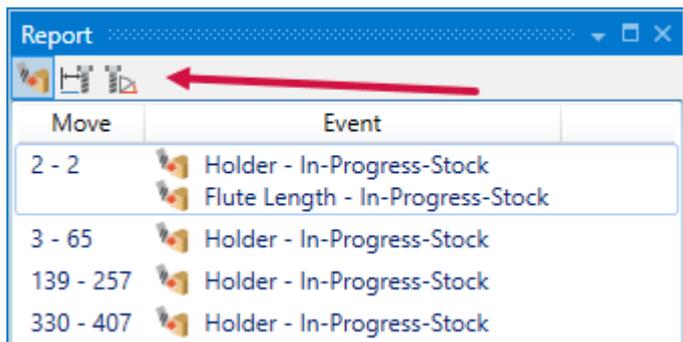
Mastercam Simulator now processes standard 3-axis and 5-axis operations faster when backplotting, verifying, or simulating. This does not include C-axis toolpaths or axis substitution.

Viewing Collisions and Alerts Using the New Report Pane

Mastercam Simulator now lists all collisions, proximity alerts, and axis overruns in the new **Report** pane, replacing the **Collision Report** pane in previous Mastercam versions. This new pane includes some of the functionality of the **Report** pane found in Machine Simulation and provides consistency between the two functions.



Filter the generated report by using the icons at the top of the pane so you can see only collisions, proximity alerts, axis overruns, or a combination.

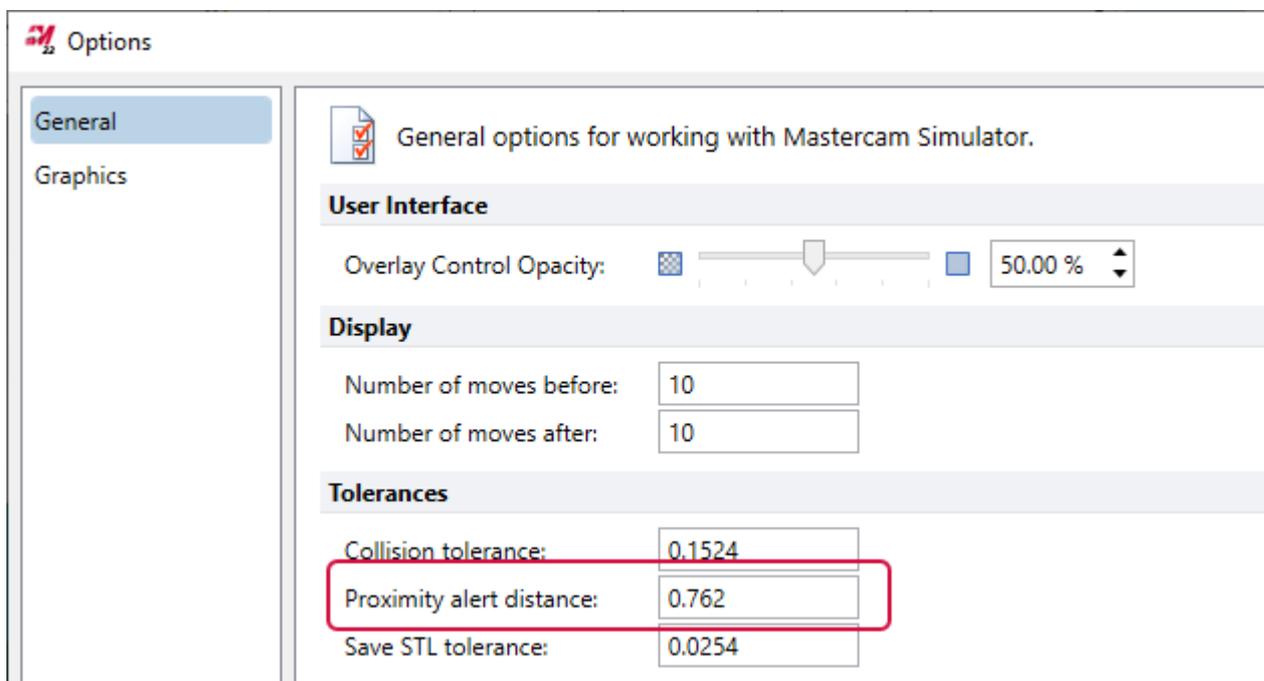


Toggle the **Report** pane on the **View** tab.



Setting Proximity Alerts for Verifying and Simulating

A new option, **Proximity alert distance**, is now available in Mastercam Simulator. This new option is adapted from Machine Simulation and is available in **Simulation** mode. This option is located on the **General** page of the Mastercam Simulator's **Options** dialog box.

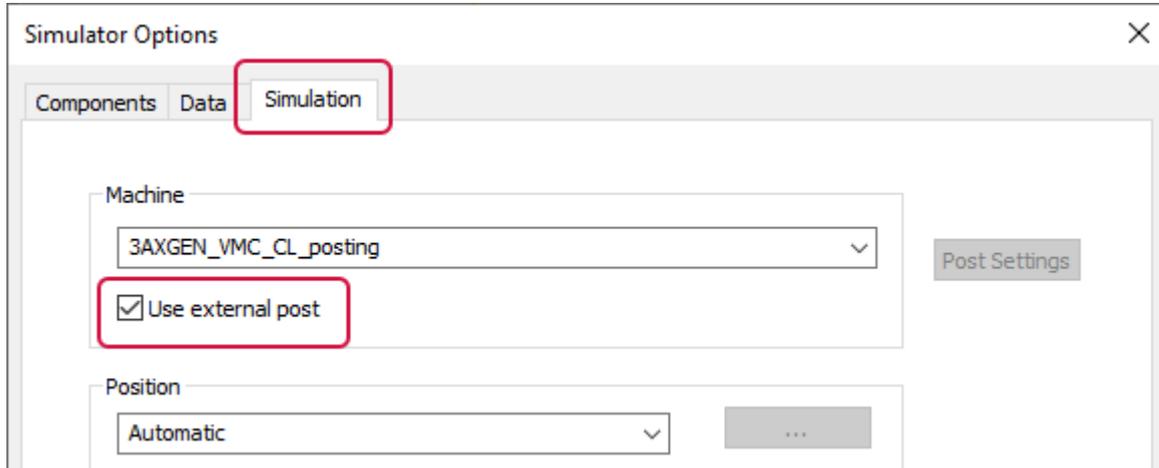


When collision checking is activated, you will receive proximity alerts in the **Report** pane when one of the chosen items comes within the **Proximity alert distance**. Double-clicking on the collision in the **Report** pane will position the tool at that move so you can see what causes the warning.

Simulating with a Specific External Post

When simulating toolpaths in Mastercam Simulator, you now can use an external post instead of a generic post. By using your posts, you can view a more realistic simulation.

Select the **Use external post** option on the **Simulation** tab of the **Simulator Options** dialog box, found on the **Machine** tab or the Toolpaths Manager to enable this new functionality.

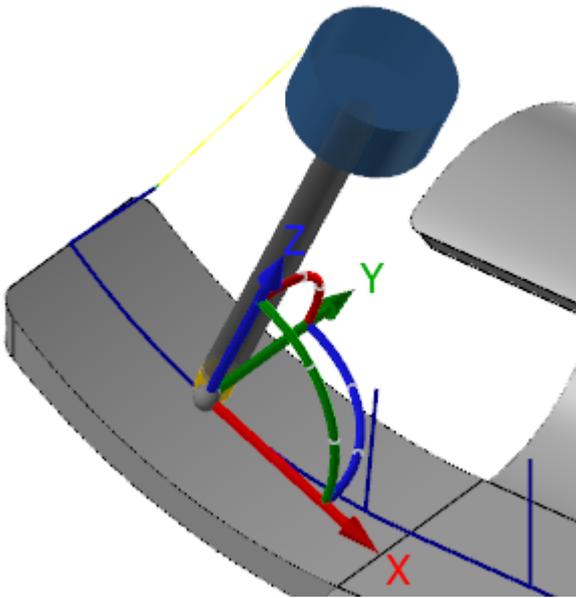


TOOLPATH UTILITY ENHANCEMENTS

Listed below are major enhancements to toolpath utilities, including the Toolpaths Manager and stock model.

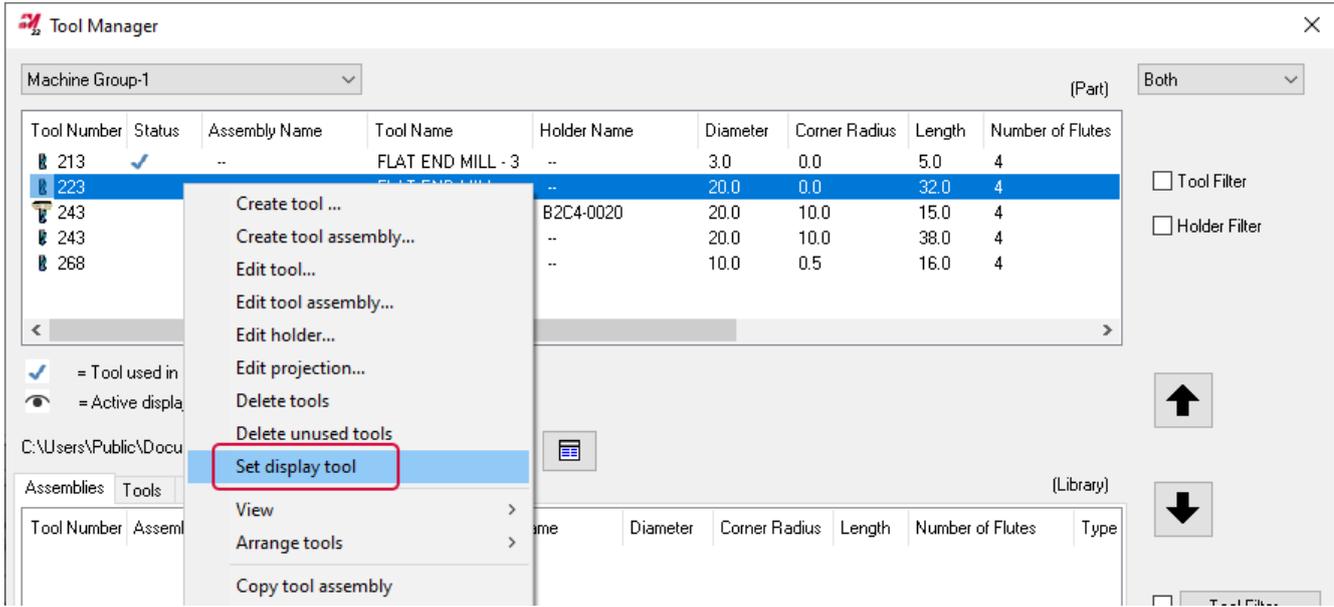
Viewing Your Tool Assembly During Part Programming

Mastercam 2022 features the **Show Tool** function, which displays a mill, router, or lathe tool assembly with the tool tip attached to your mouse cursor. By moving your cursor around, you can quickly see if your tool or holder might cause a problem while simultaneously modeling or programming your part. You have control over displaying a tool, holder, or both, as well as controlling the shading and opacity.

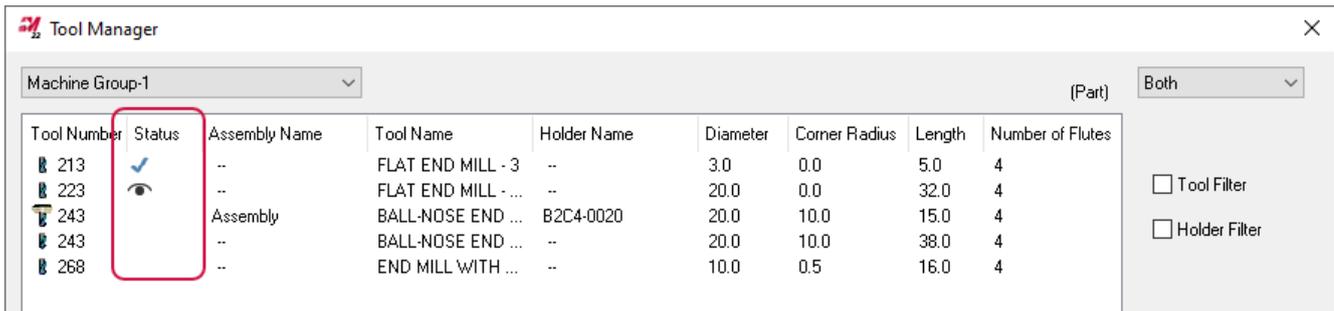


Multiple functions have been enhanced to show the display tool, including Mastercam 2022's new **Modify Vector** (refer to "[Enhancements to Support Multiaxis Machining](#)" on page 125 for more information), allowing you to identify potential issues upfront while you edit. For functions that support a display gnomon or green arrow, such as **Create/Edit Plane**, the tool detaches from your cursor and aligns with the blue Z axis of the gnomon or the green normal arrow. Streamline your workflow by identifying potential holder collisions while creating a new plane for tilt machining.

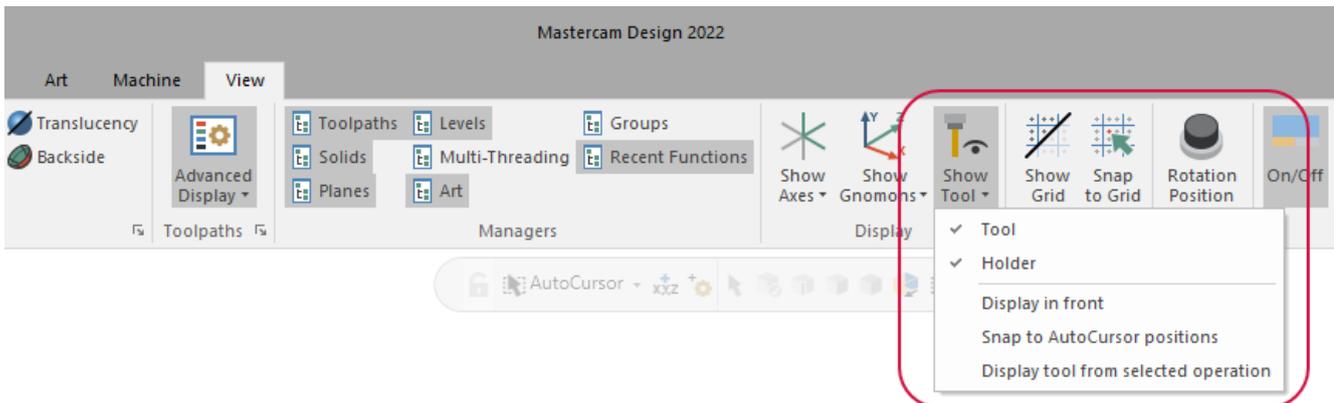
To use **Show Tool**, right-click on the desired display tool in the Mill Tool Manager or the Lathe Tool Manager part tool list and select **Set display tool**. Your tool does not have to be associated with a particular toolpath. Mastercam remembers your selection even after you close the part file.



When a tool is set as the display tool, it is indicated by an icon in the **Status** column.



Toggle **Show Tool** on or off in the **Display** group of the **View** tab or use the **[F8]** shortcut key. Your selection persists throughout Mastercam sessions. Choose to display the tool, holder, or both with respect to the active tool plane.

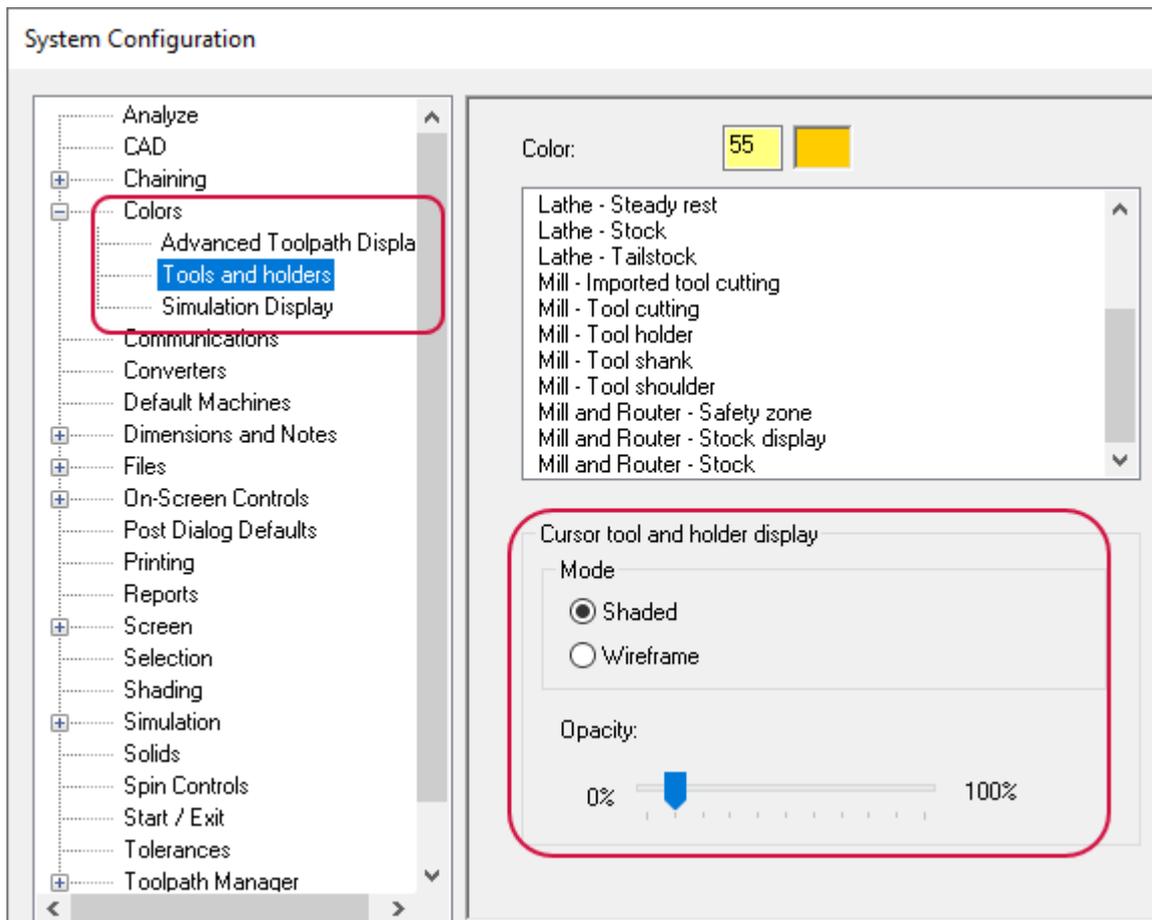


Show Tool includes the following options:

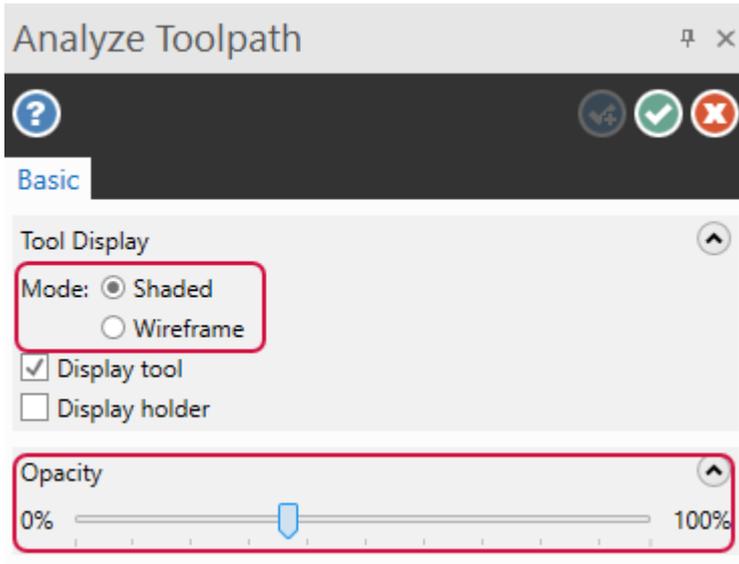
- **Display in front:** Displays the designated tool in front of all other entities.
- **Snap to AutoCursor positions:** Snaps the display tool to any active AutoCursor positions, as defined in the AutoCursor settings, while you are not currently in a function.
- **Display tool from selected operation:** Switches between the default display tool and the tool of any selected Mill, Lathe, or Router operation.

Additionally, when using Dynamic Rotation, the tool is placed on the orientation origin point. If you select multiple operations, no tool is displayed.

To customize your tool assembly display, use the **Cursor tool and holder display** group, found in the **Tools and Holders** page, under **Colors** in the **System Configuration** dialog box.

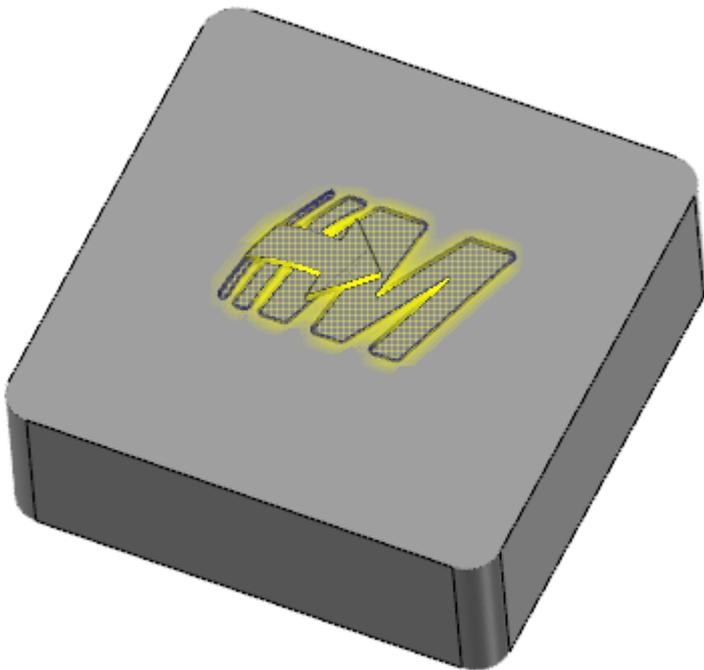
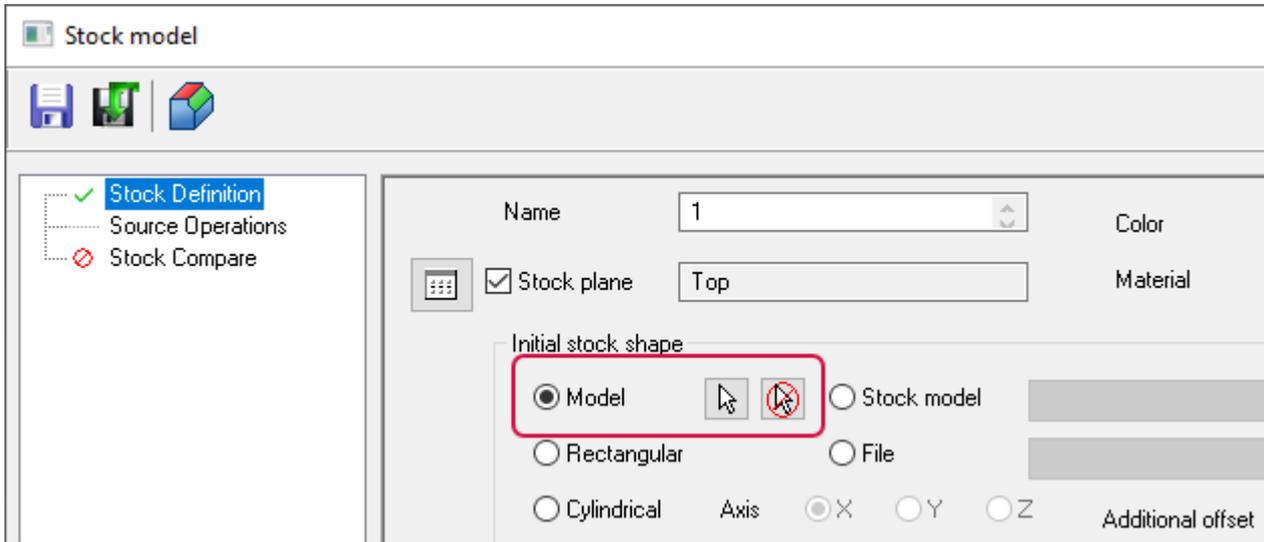


These settings are also applied to **Analyze Toolpath** function panel by default.



Selecting Individual Solid Faces for Stock Model

When creating a stock model operation, you can now select individual solid faces when you select the **Model** option for the initial stock shape on the **Stock Definition** page. In previous releases, you were limited to only solid bodies.



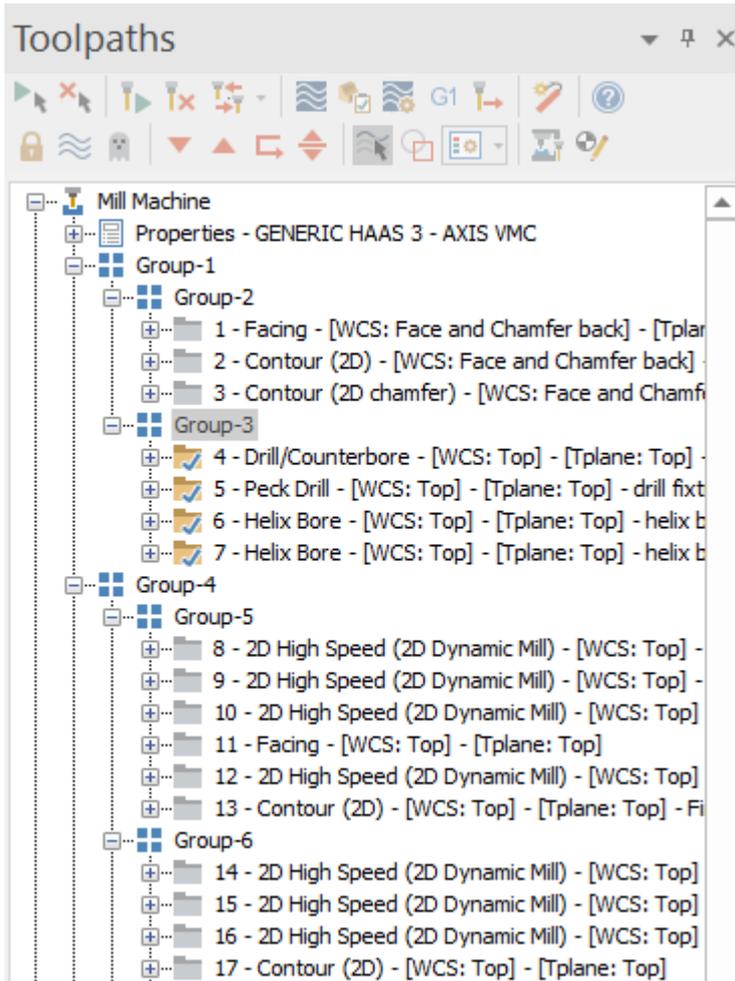
Toolpaths Manager Enhancements

Listed below are enhancements to the Toolpaths Manager.

New Toolpaths Manager Icons

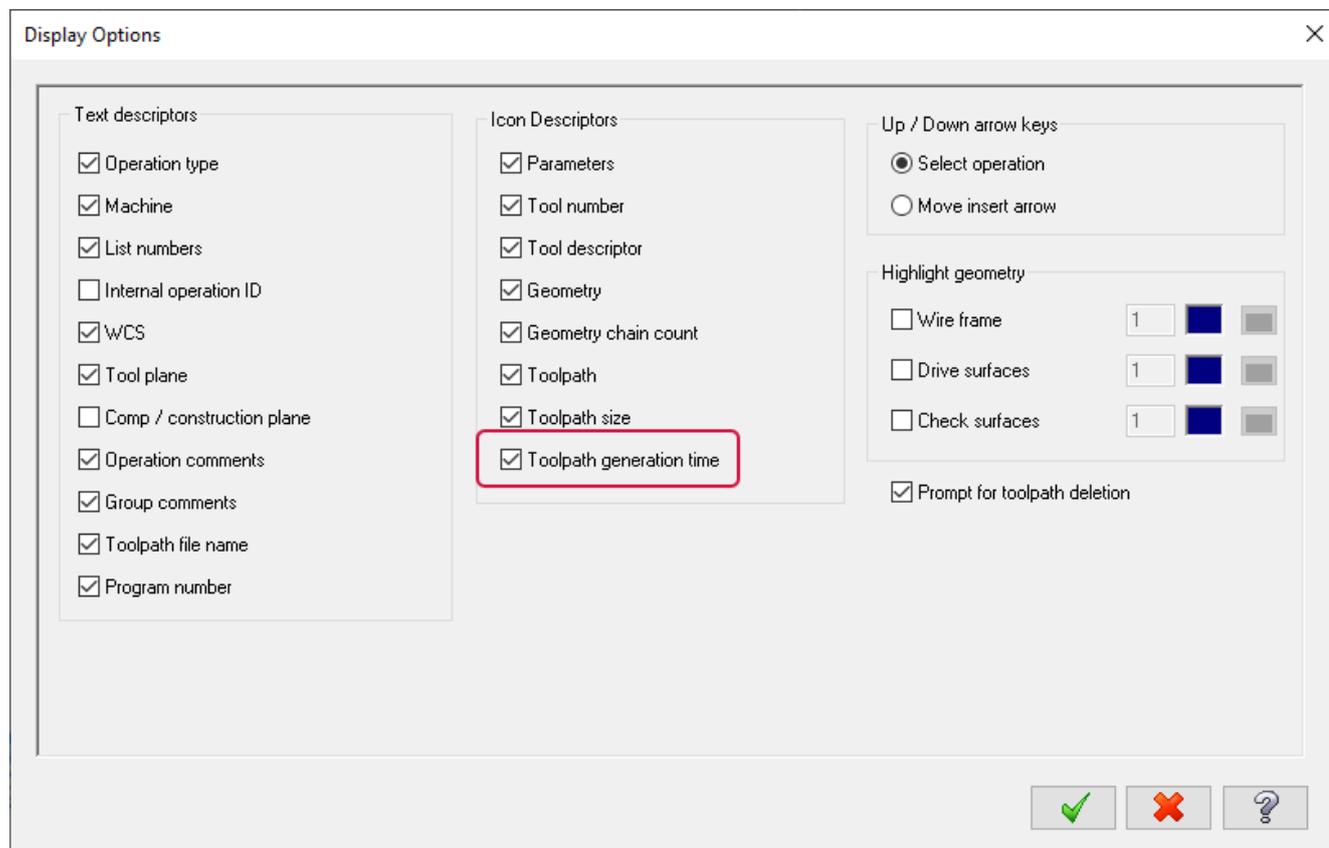
New Toolpaths Manager icons contribute to a clear, modern look for Mastercam 2022. The new icons also scale with font size. Other changes also include:

- Machine specific stock setup icons in the Toolpaths Manager.
- A dimmed appearance makes the disabled status is easier to see on operations with their geometry toggled off.

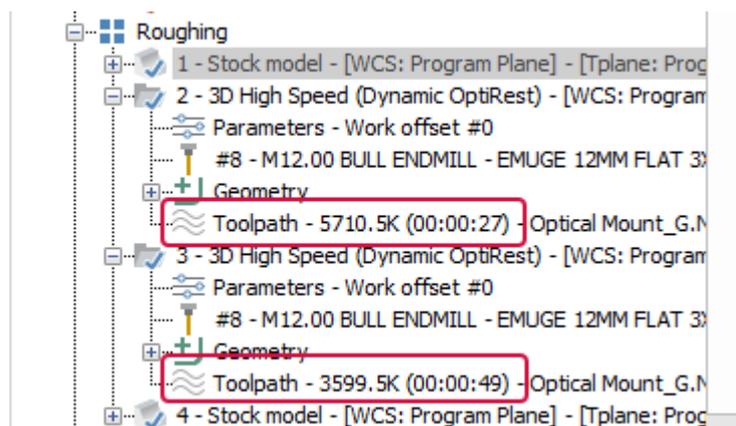


Exploring Toolpaths Manager Display Options

The Toolpaths Manager display options have been reorganized in Mastercam 2022 and are now accessible from two locations: the new **Display Options** page of the **System Configuration** dialog box, and the original location in the right-click menu of the Toolpaths Manager.



A new option, **Toolpath generation time**, is now available and is deselected by default. Select this option to display the time spent generating the respective toolpaths.



SYSTEM ENHANCEMENTS

Listed below are enhancements made to Mastercam's core features, including graphics, planes, and levels.

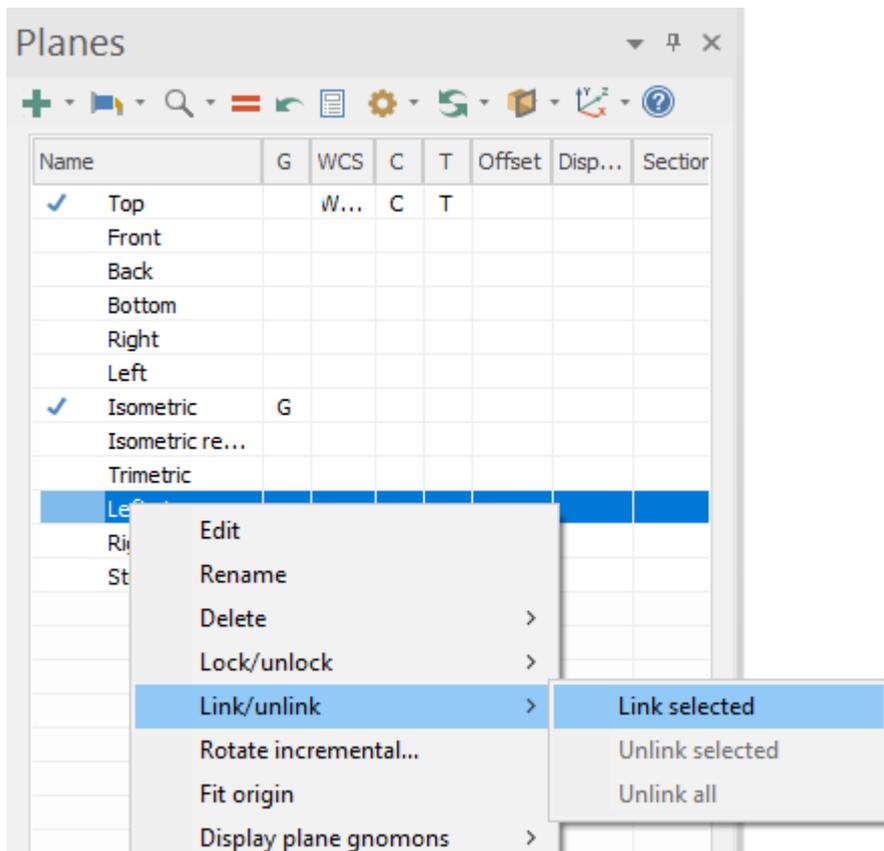
Planes and Planes Manager Enhancements

Listed below are enhancements to planes and the Planes Manager.

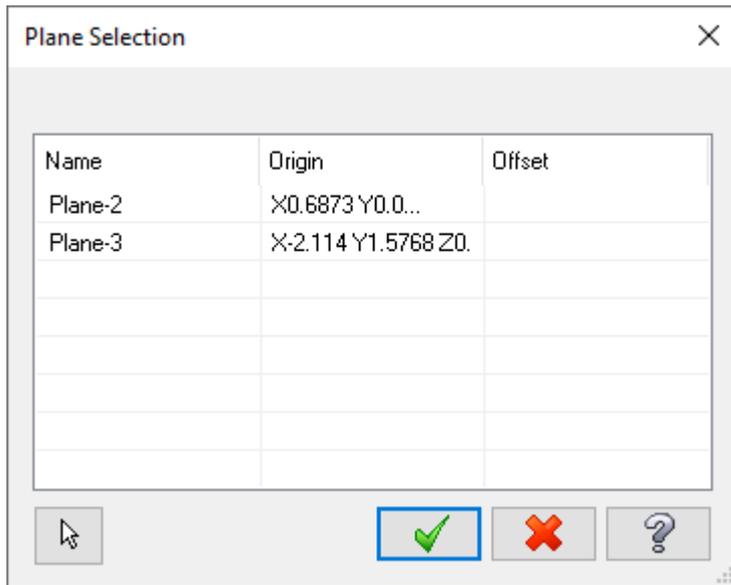
Linking and Unlinking Planes

Mastercam 2022 includes the ability to link and unlink custom planes in the Planes Manager. Link planes when you wish to maintain a spatial relationship between them. For example, if you link **Plane A** to **Plane B**, moving **Plane A** will move **Plane B** by the same amount. A parent-child relationship exists between linked planes, with child planes dependent on the parent plane. Parent planes cannot be linked to each other.

Access the **Link/Unlink** option via the Planes Manager right-click menu, the Dynamic Gnomon right-click menu, or the **Link** column in the Planes Manager. You can select this option when creating a plane.

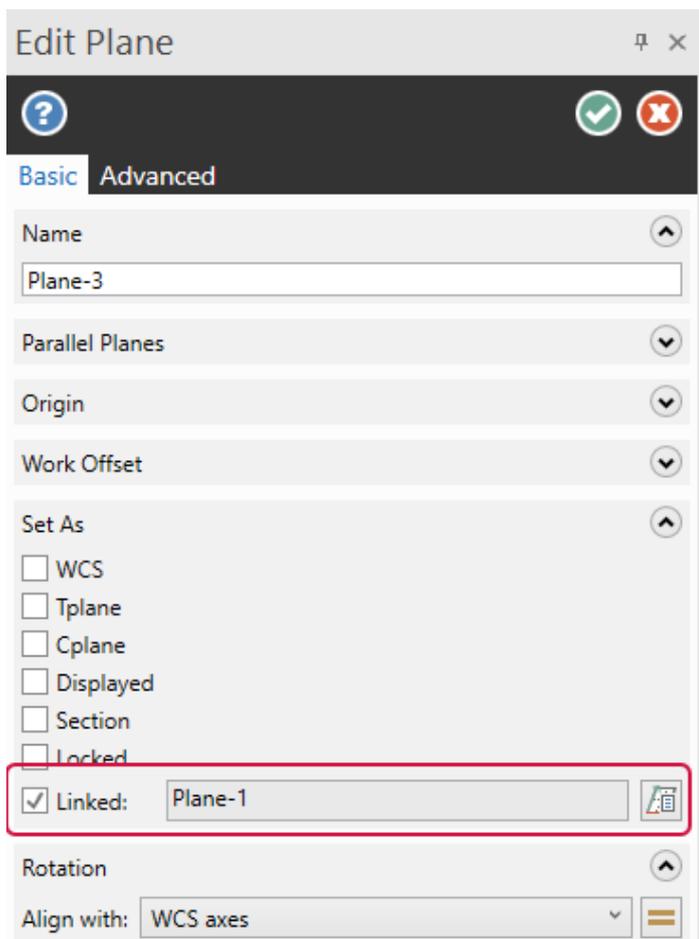


Set a selected plane as the child plane by using any of the methods described previously. Then select the parent plane from the **Plane Selection** dialog box.



To unlink planes, follow the same steps and select **Unlink selected** or **Unlink all**.

While creating or editing a plane, you have the option to link it. Select **Linked** in the **New Plane** or **Edit Plane** function panel. Then, select the **Named Plane** button to select a parent plane.



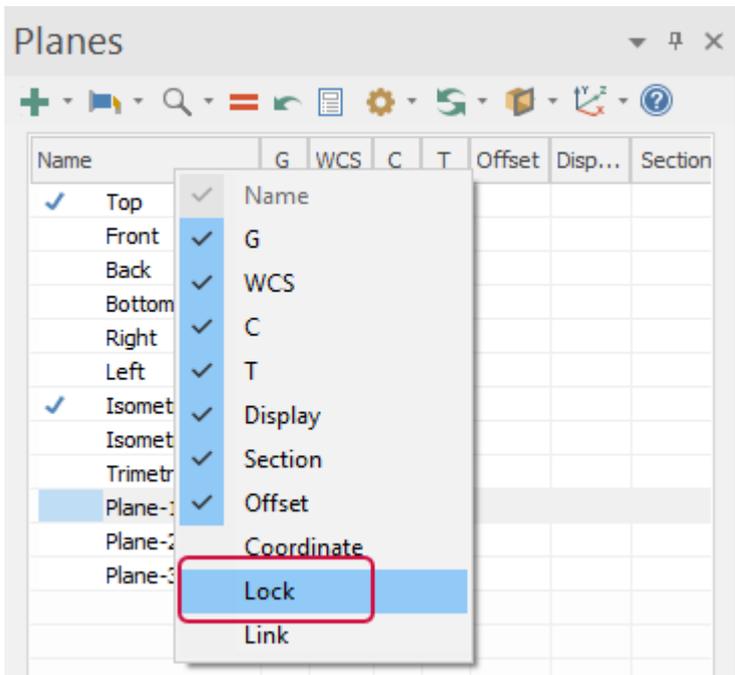
You can view the parent-child relationships by selecting and hovering over a plane in the Planes Manager. New icons in the **Name** column also help you see which planes are linked.

Name	G	WCS	C	T	Offset	Disp...	Section
✓ Top		W...	C	T			
Front							
Back							
Bottom							
Right							
Left							
✓ Isometric	G						
Isometric re...							
Trimetric							
Plane-1							
Plane-2							
Plane-3							

Directly Locking and Unlocking Planes

With this release of Mastercam, you can use the new **Lock** column in the Planes Manager to quickly lock and unlock planes. Previous versions of Mastercam required several steps to lock and unlock a plane.

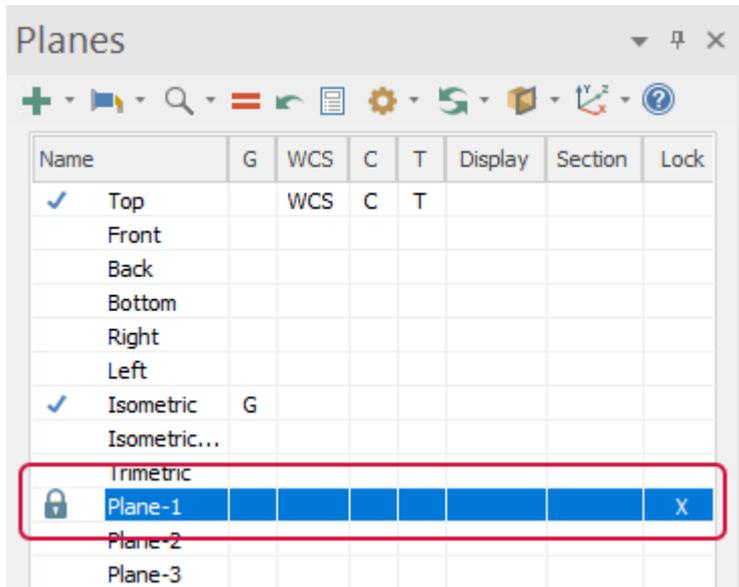
To add the **Lock** column to Planes Manager, right-click in the header and select **Lock**. You can also remove the **Lock** column from the grid: right-click in the header and deselect **Lock**.



Click in the **Lock** column for that plane to toggle the **Lock** option on or off.

toggling Display and Section for Locked Planes

Work more efficiently with the new ability to toggle **Display** and **Section** for locked planes. Click in the **Display** or **Section** column of the locked plane to toggle each setting on or off. Other settings for locked planes cannot be changed.

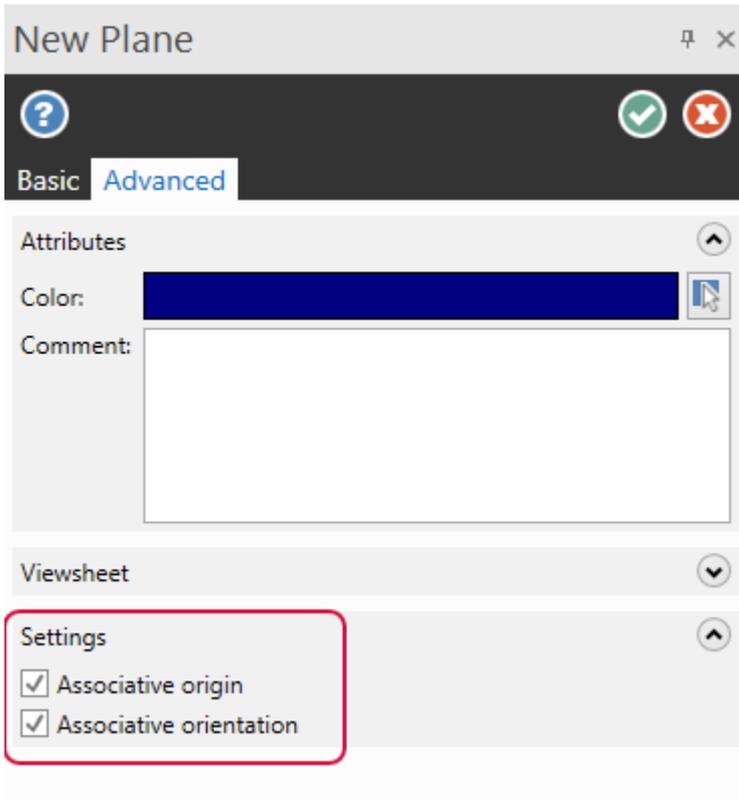


Controlling Plane Associativity

To give you greater control over plane associativity, Mastercam 2022 has split the **Associative** setting into two options:

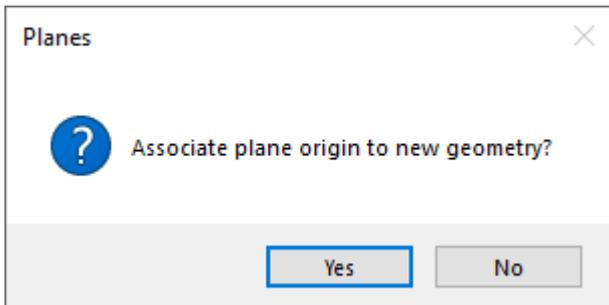
- **Associative origin** controls the origin associativity for the selected plane.
- **Associative orientation** controls the view associativity for the selected plane.

Switch between settings as needed to gain clarity while working on your part. For quick access, you can find these settings in both the Planes Manager and the **Advanced** tab of the **New/Edit Plane** function panel. As with the previous **Associative** option, both settings are selected by default.



Associating the Plane Origin to New Geometry

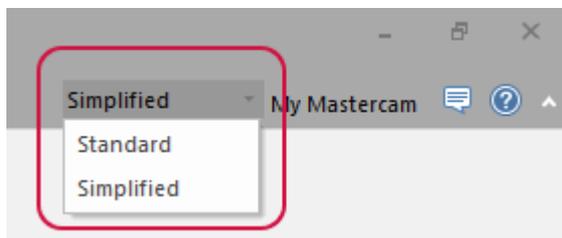
Mastercam 2022 gives you additional control when moving the origin of a named plane to a new location. Upon selecting new geometry you are given two options:



- Select **Yes** to associate the plane's origin to the newly selected geometry.
- Select **No** to keep the plane origin's current associations.

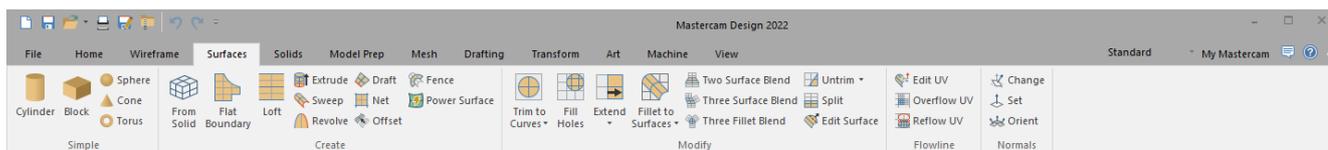
Simplifying the Mastercam Ribbon

A simplified view of the Mastercam ribbon has been introduced with the new **Ribbon Mode** function. This function replaces the Quick Part add-in, which was previously available as an Add-In only for Education users. To toggle between **Standard** and **Simplified** mode, select the **Ribbon Mode** drop-down in the upper right corner of the Mastercam window. Mastercam remembers your selection between sessions.

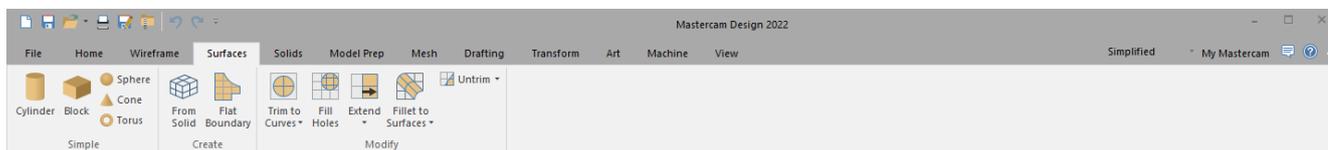


New users, students, and educators can use this mode to familiarize themselves with the most common Mastercam functions, while maintaining the ability to switch back to **Standard** mode. Functions that are hidden from the Mastercam interface when set to **Simplified** includes some of the more complex Analyze functions, Drafting functions, Curve and Line functions, Delete functions, and View options. Wire and Mill-Turn are not available in **Simplified** mode.

Surfaces tab, Standard mode:



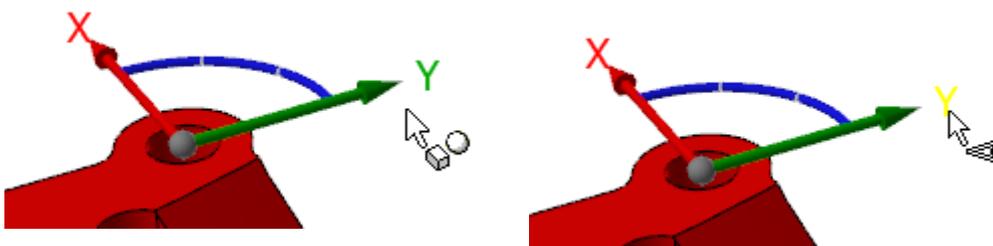
Surfaces tab, Simplified mode:



Setting the View with the Dynamic Gnomon

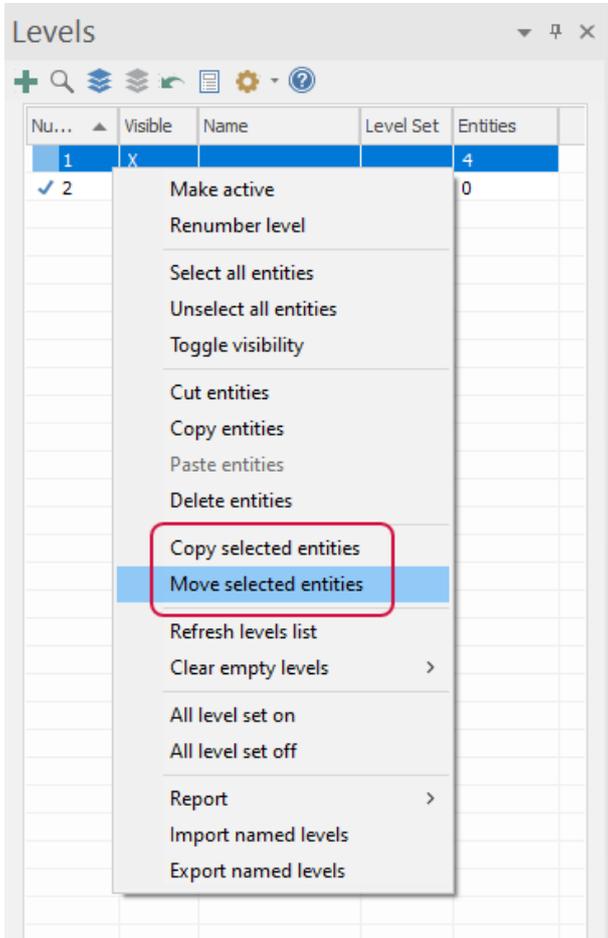
While using the Dynamic Gnomon in functions such as plane creation and Dynamic transform, you can set the view to be normal to any of the three axes (X, Y, Z). This adjustment is relative to the orientation of the Dynamic Gnomon itself.

To use this feature, move your mouse over the desired axis letter of the gnomon. Click the letter when it becomes highlighted yellow. The view adjusts so that it is normal to the axis you have selected.



Copying and Moving Selected Entities Between Levels

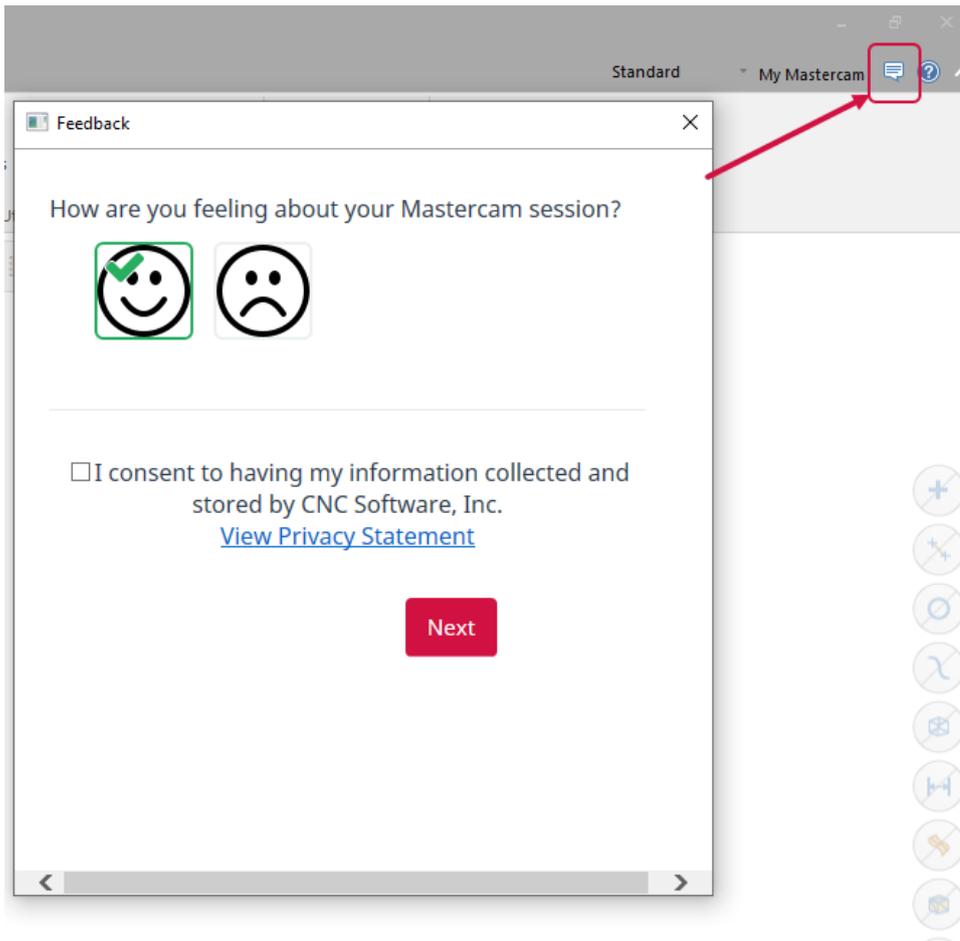
The Levels Manager now allows you to copy or move selected entities between levels. By copying or moving entities, you can avoid regenerating toolpaths that are associated with the geometry you wish to move. To copy or move the entities, first select the entities in the graphics window. In the Levels Manager, right-click on the level you wish to add to and select **Copy selected entities** or **Move selected entities**.



Providing Feedback During Your Mastercam Session

Mastercam 2022 features a new in-application customer feedback survey, designed to collect the most important information from you in the least amount of time. This survey is translated into 17 languages and automatically displays in the same language as the Mastercam application. You can submit feedback, including comments, scores, and general satisfaction, as many times as you like to help improve Mastercam.

To send your feedback, click the **Feedback** button in the upper right-hand corner of the Mastercam window and follow the prompts in the **Feedback** dialog box.

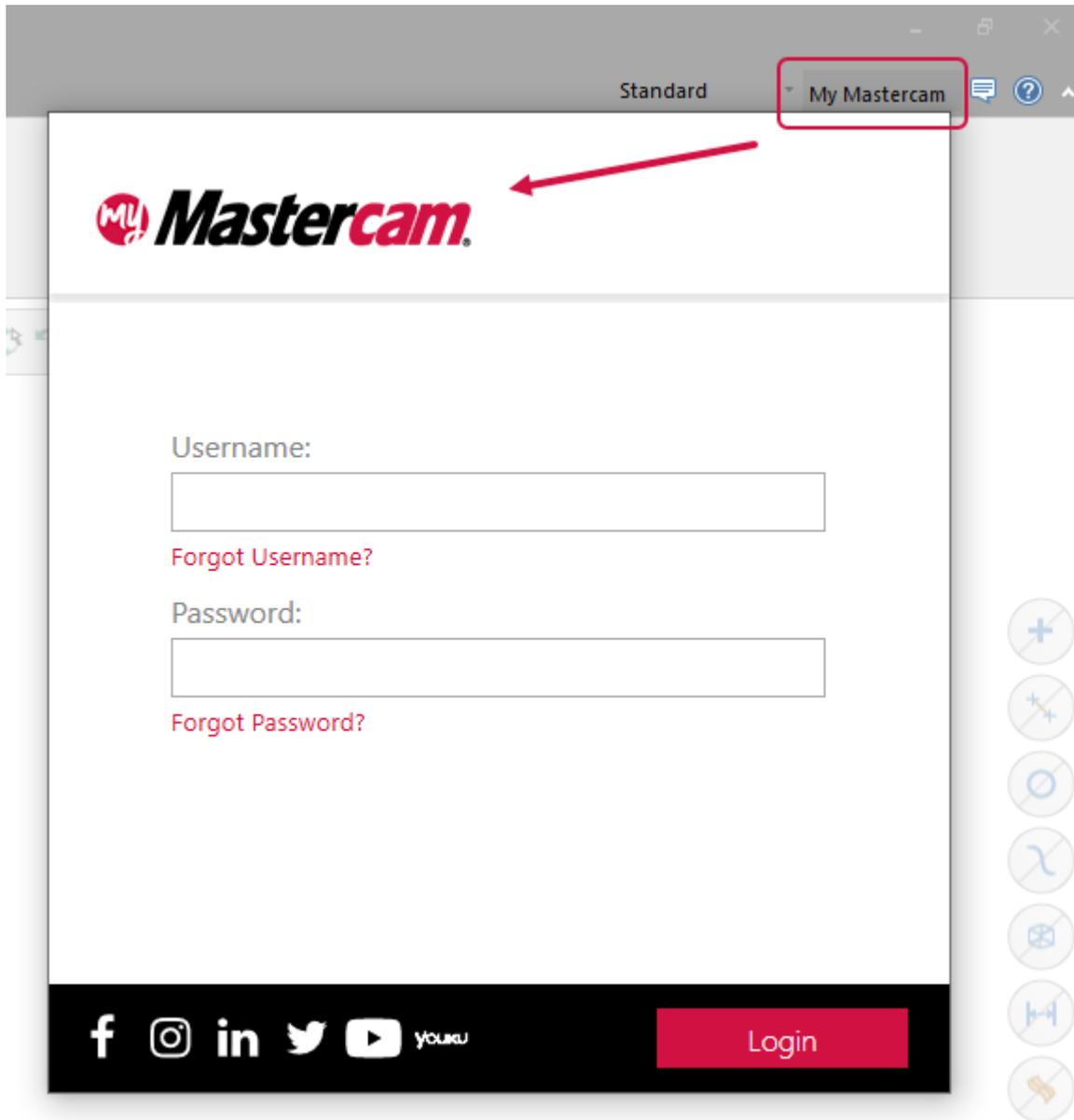


When you click **Next**, you are then prompted to provide details about your Mastercam experience, by entering comments and answering a few questions.

The screenshot shows a 'Feedback' dialog box with a close button (X) in the top right corner. The first question is 'How is your Mastercam experience today?' followed by a large empty text input box with a vertical scrollbar. Below this is a horizontal line. The second question is 'Mastercam's capabilities meet my requirements.' with a 7-point Likert scale. The scale is labeled '1' through '7' above a row of seven radio buttons. 'Strongly disagree' is on the left and 'Strongly agree' is on the right. A horizontal line follows. The third question is 'Mastercam is easy to use.' with another 7-point Likert scale, also labeled '1' through '7' above radio buttons, with 'Strongly disagree' on the left and 'Strongly agree' on the right. A vertical scrollbar is on the right side of the dialog box.

My Mastercam Login Enhancements

The **My Mastercam** dialog box has been revamped, featuring a clean, modern look.



Industrial and educational users can now use this dialog box as a simple and straightforward way to link their My Mastercam user account and Mastercam license. Account linking allows you to gain access to features on [Mastercam.com](https://www.mastercam.com), as well as view information about resources relevant to your account. These features include access to the following:

- User forums
- Product downloads

If you have active Mastercam maintenance, you gain access to even more features, including:

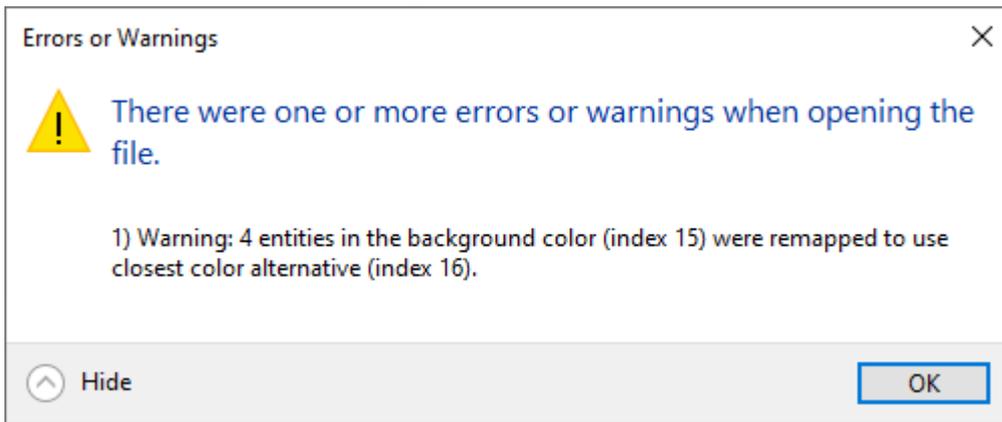
- Knowledge Base
- Tutorials

- Webinars
- Documentation

After entering your user name and password in the **My Mastercam** dialog box, click **Link account**. You can unlink accounts at any time by clicking **Unlink account**. This feature is available to users with industrial and educational licenses.

Automatically Change Geometry Color When Opening a File

In Mastercam 2021, if you had a part file that was the same color as the background color, and did not change it when opening the file, it would not be visible after opening. Now upon opening a file, Mastercam alters background-matching geometry to the nearest color match so that it is visible.

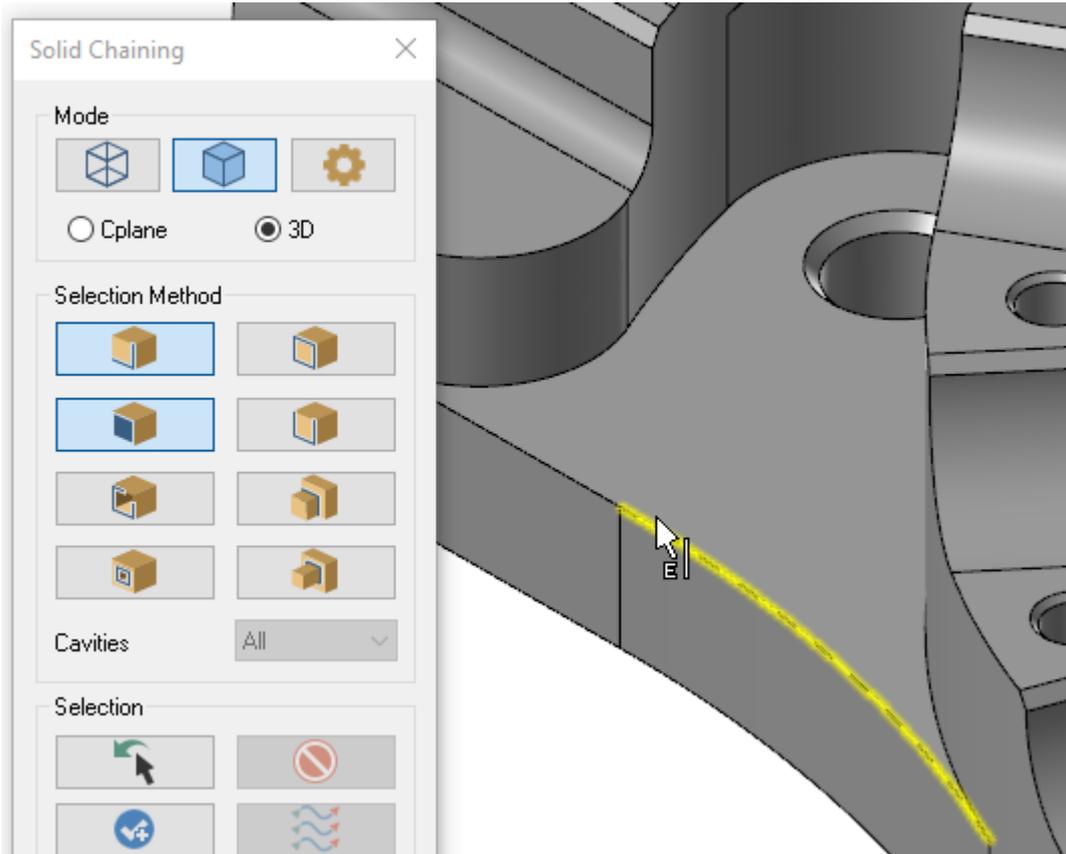


SELECTION AND CHAINING ENHANCEMENTS

Listed below are enhancements made to selection and chaining in Mastercam.

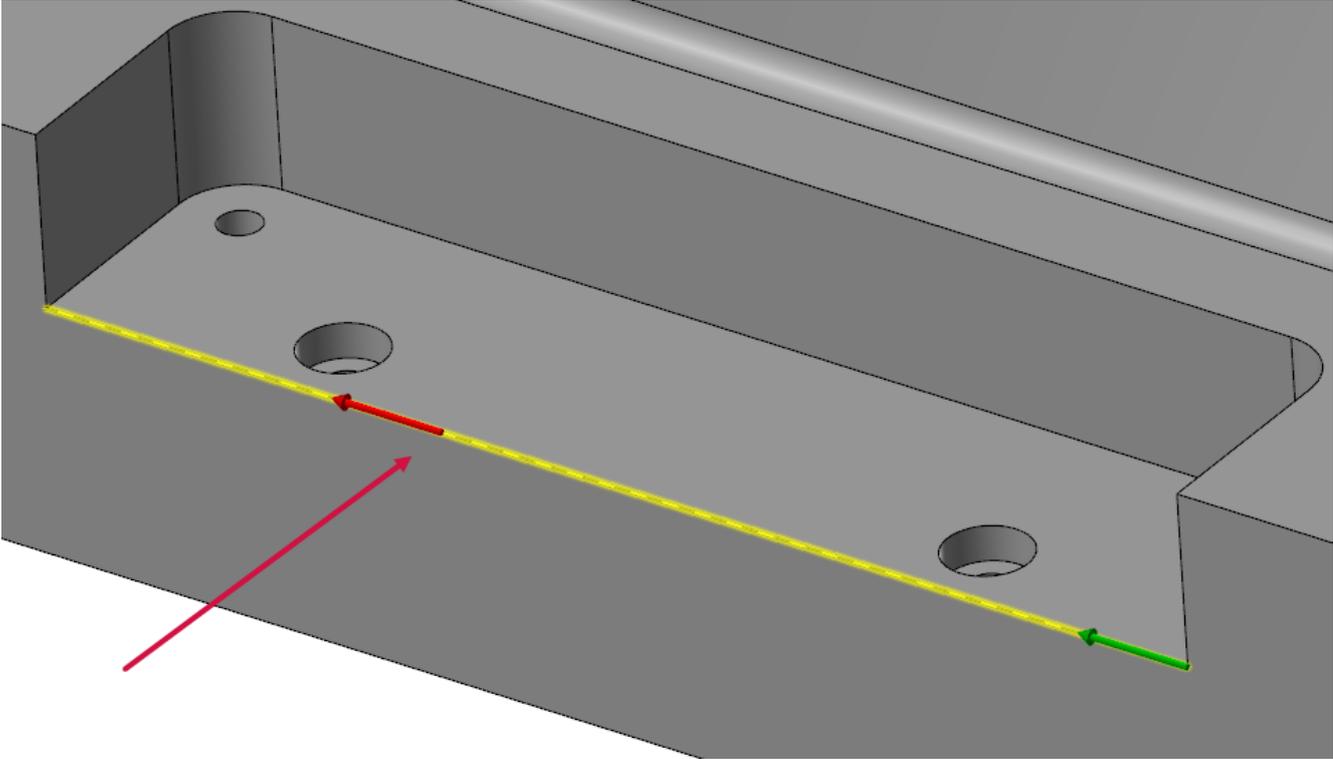
Rapidly Select Solid Edges

Solid chaining features a speed improvement in Mastercam 2022. Enhanced edge propagation displays your selected chains more rapidly and is particularly noticeable when chaining features such as tangent edges. This enhancement applies automatically when you select solid edges, saving you time while enhancing your productivity.



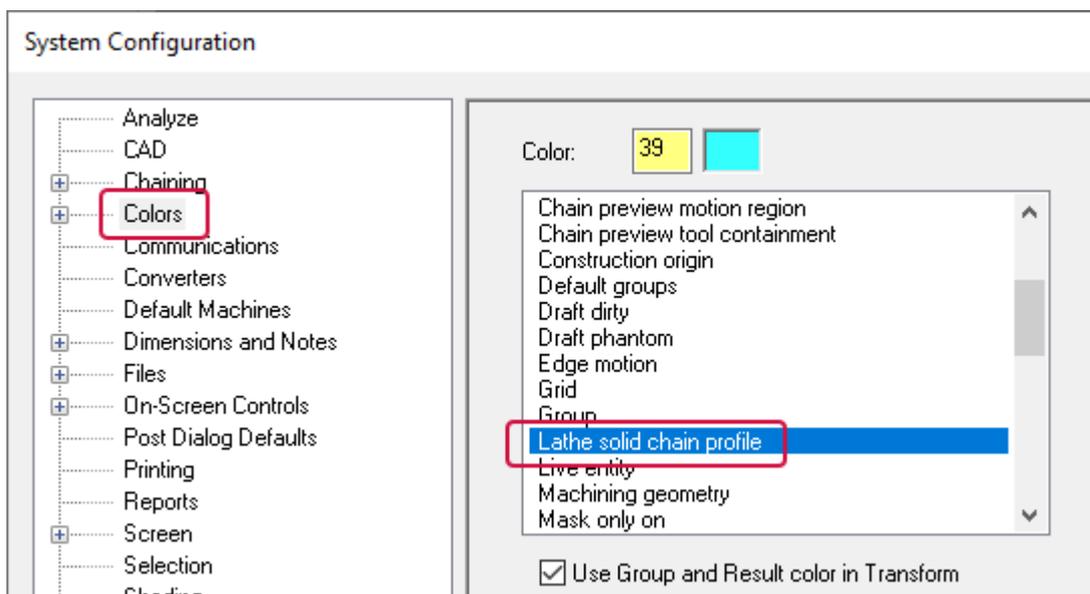
Dynamically Adjust the Start and End Point of Chains

Dynamic chaining now allows you to set the start or endpoint of your chain by clicking snap points that are outside of your chain. Select **Dynamic** in the **Chaining** dialog box and then select a chaining arrow to move. The arrow remains perpendicular to your cursor even when you hover over unchained geometry. Cursor snap points make it easy to align start and endpoints with existing geometry. **Dynamic** also remains active until you press [Esc] or [Enter]. You save time by rapidly adjusting start and endpoints with fewer mouse clicks.



Setting the Color for Lathe Solid Chain Profile Preview

You can customize the color for the lathe solid chain profile preview by selecting the new **Lathe solid chain profile** option in the **Colors** page of the **System Configuration** dialog box and setting the color.



Re-selecting and Ending Chains for Lathe Operations

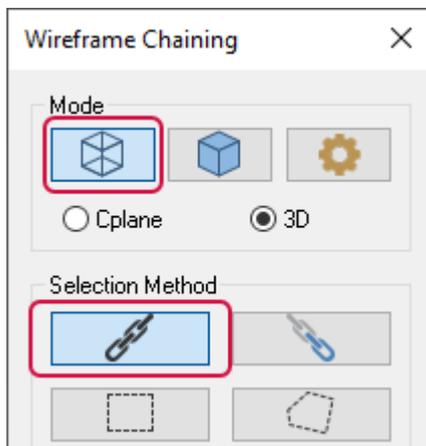
Lathe solid chaining now features the **Last** and **End** chain options found in other Mastercam chaining functions.

- **Last** re-selects entities chained during the previous chaining operation. This is useful when correcting chains for a surface that did not generate properly. Mastercam re-chains the entities so that you can edit the chains and correct chaining direction or order.
- **End chain** ends the current chain, allowing you to start a new one.

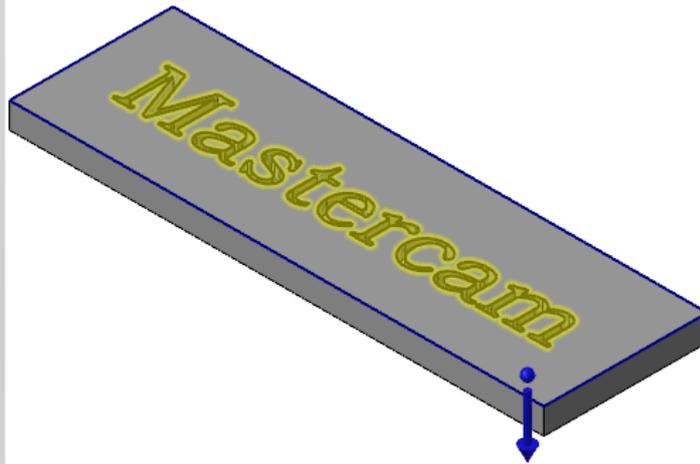
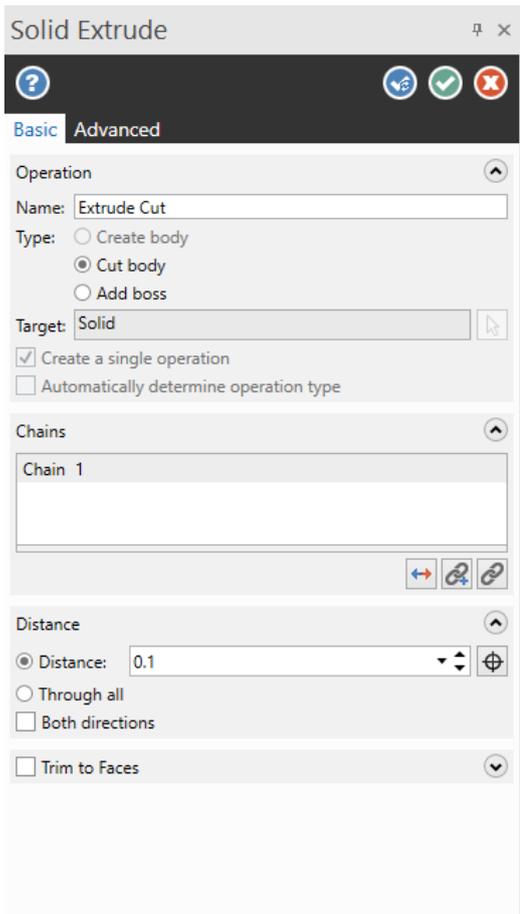
Chaining and Extruding Notes

Mastercam 2022's new note chaining functionality allows you to chain notes for toolpaths. Additionally, chained notes are associative. You can change the font or text as many times as needed, then regenerate the toolpath to reflect changes made to the note. Notes can also be associated to geometry. If the geometry is moved, the note moves with it.

You can only chain notes when **Wireframe** mode is selected in the **Wireframe Chaining** dialog box.



Along with the ability to chain notes, Mastercam supports using the **Solid Extrude** function for notes. You can modify solids using the **Add boss** or **Cut body** options in the **Extrude** function, located on the **Solids** tab.

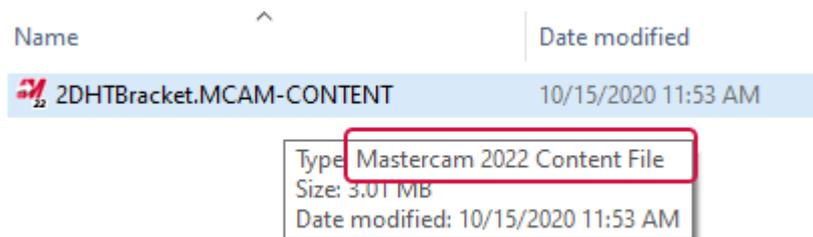


FILE MANAGEMENT ENHANCEMENTS

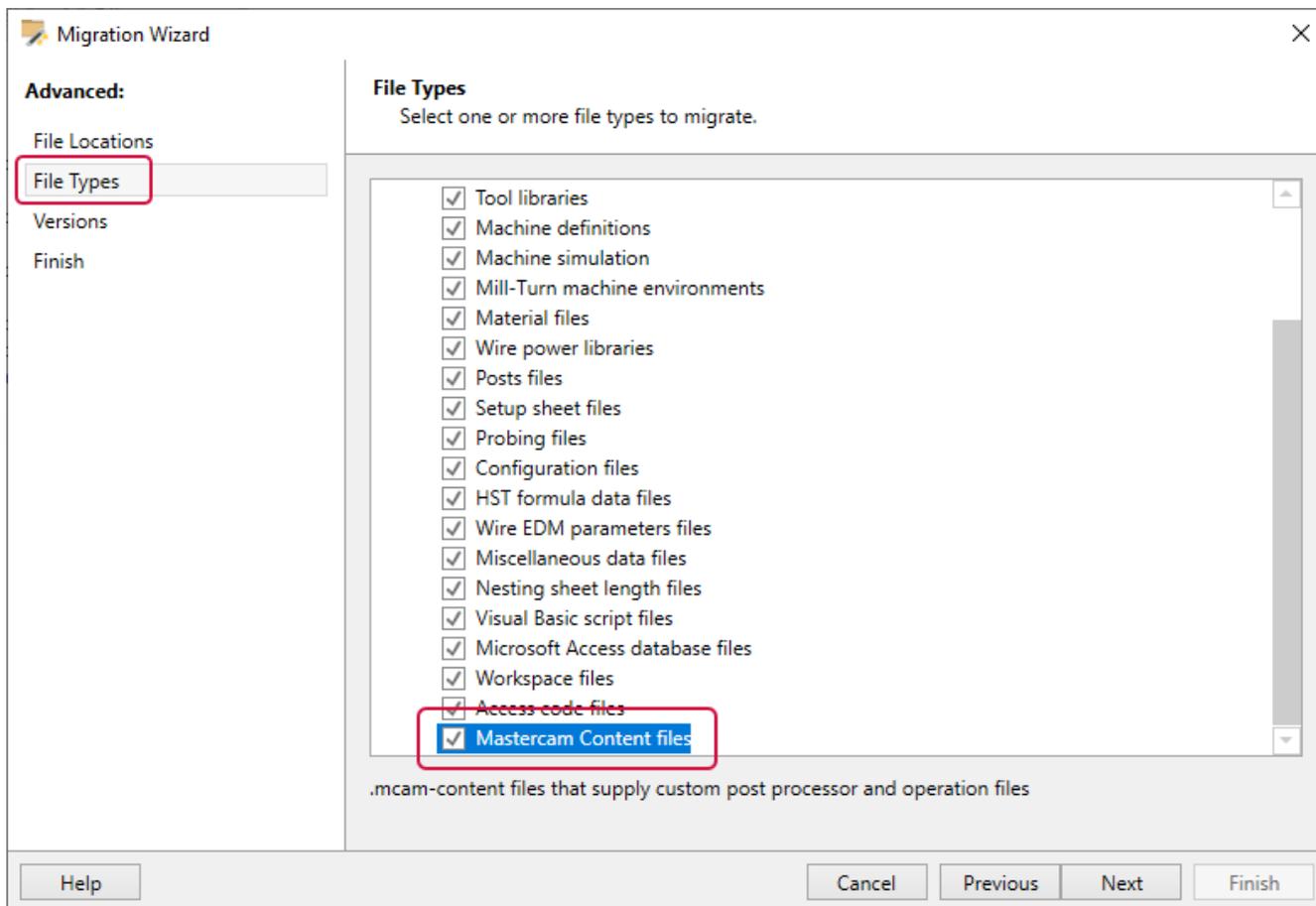
Listed below are enhancements to file usage and converters in Mastercam.

Working with Mastercam Content Files

Mastercam 2022 incorporates versioning into .mcam-content files, making it easy to check if your compressed file package is compatible with the current version of Mastercam. Like with .mcam files, the version is displayed on the file icon. You can also hover over your completed file package in Windows Explorer to view the version of the part file.



The Migration Wizard has also been enhanced to allow you to update Mastercam Content files. Content files are selected by default on the **File Types** page of the **Advanced** option in Migration Wizard. As with other files on this page, deselect **Mastercam Content files** from the list if you do not wish to migrate them.



File Translator Enhancements

Listed below are enhancements to importing and exporting other CAD file formats.

Support for 3MF Files

Mastercam now supports the import and export of 3MF files. 3D Manufacturing Format (3MF) is a file format designed specifically for additive manufacturing. It includes material, color, and other information that cannot be represented in the STL format.

Import and export 3MF files with the **File, Open** and **Save As** commands, just as you would any other Mastercam-supported CAD file.

The **Converters** page of the **System Configuration** dialog box lets you select default parameters for conversion to and from the .mcam and .3MF formats. You can also override these parameters when you convert a file to 3MF by choosing **Options** in the **Save As** dialog box.

Importing and Exporting CAD Files with New File Translators

Mastercam 2022 includes updates to the file translators in Mastercam 2022 to support these CAD file formats:

- Parasolid v32
- ACIS 2021.1
- Rhino 7
- Spatial 2021 1.0 SDK

Mastercam also no longer supports importing or exporting CATIA V4 files.

Multiple STL Files Merged as Separate Level Names

When you merge multiple STL files into a single .mcam file, Mastercam places the entities from each file onto separate levels that retain the name of the STL file. This enhancement makes the organization of your part files more manageable and should be especially helpful if you import STL files to build machines.

Merge Enhancements

Listed below are enhancements made to the **Merge** function, located on the **File** tab.

Merge Multiple Files in One Operation

When programming your parts, you may use libraries of jigs, fixtures, and clamps that are kept in different locations. In previous Mastercam versions, you needed to create separate merge operations for each file. In Mastercam 2022, you can merge several parts from many locations using one merge operation.

Import files in either of two ways:

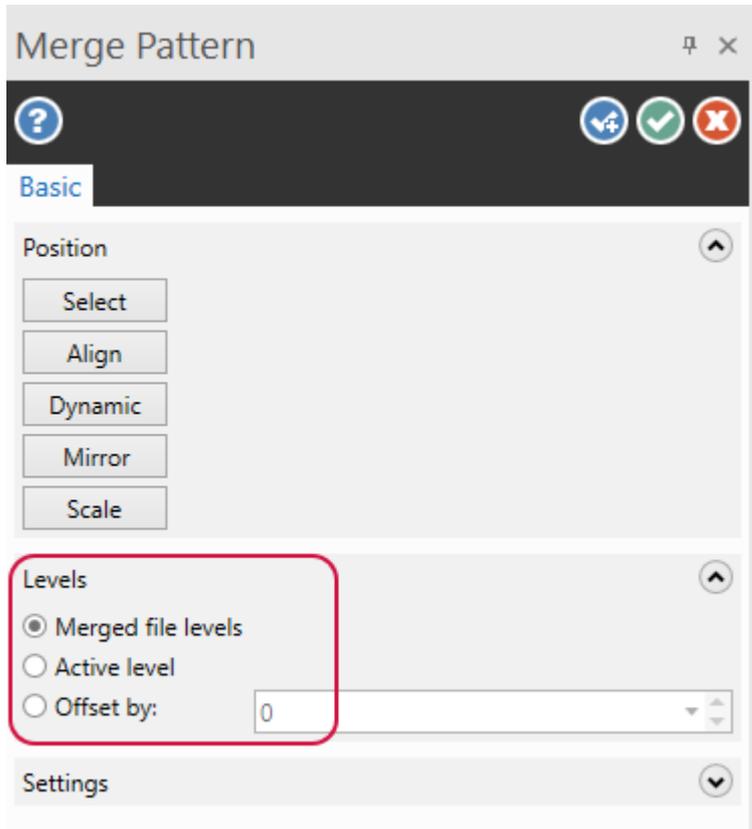
- Select **File, Merge** and choose your part files. You can also select a directory to merge all of the Mastercam files in it.
- Select the files in the Windows Explorer, and press the **[Ctrl]** key as you drag and drop the files onto the Mastercam graphics window.

NOTE

The Merge function will not import entities within sub-folders.

See Level Options Immediately Upon Merging Files

The **Merge** function now presents level information earlier in the process. In Mastercam 2022, the **Levels** options display the results in the Levels Manager while the operation is processing. In earlier releases, the results did not display until after the operation was complete.



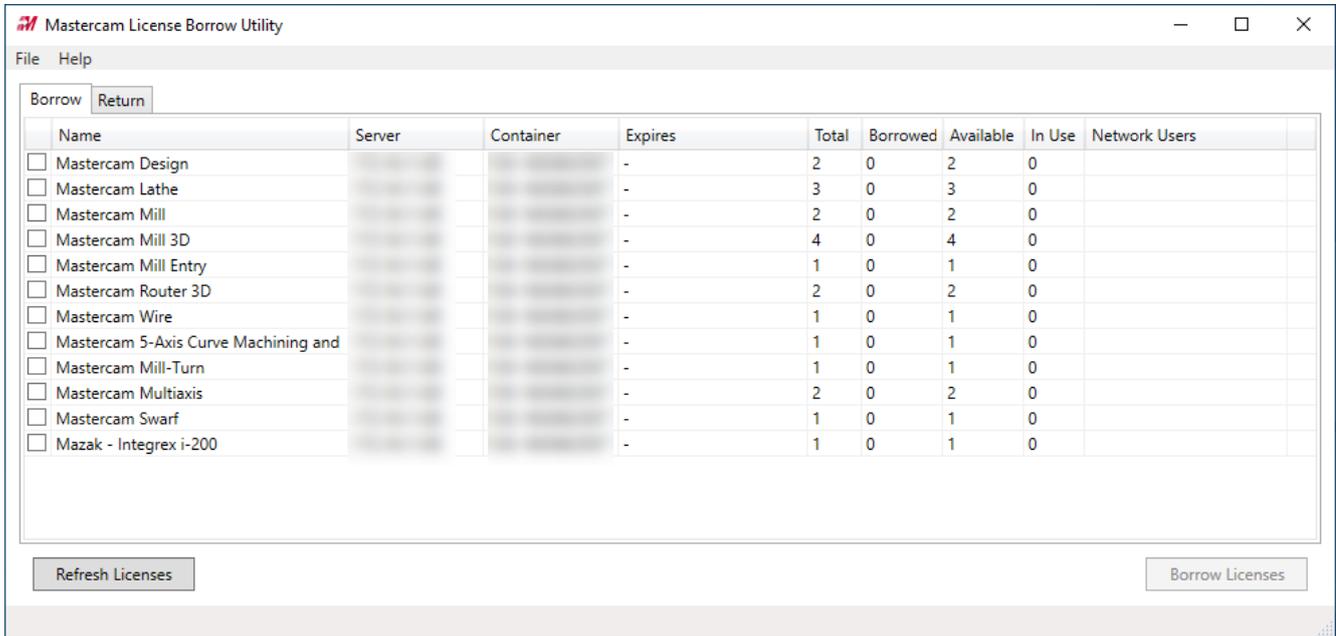
GENERAL ENHANCEMENTS

Listed below are general enhancements to Mastercam.

Borrowing Mastercam Network Licenses

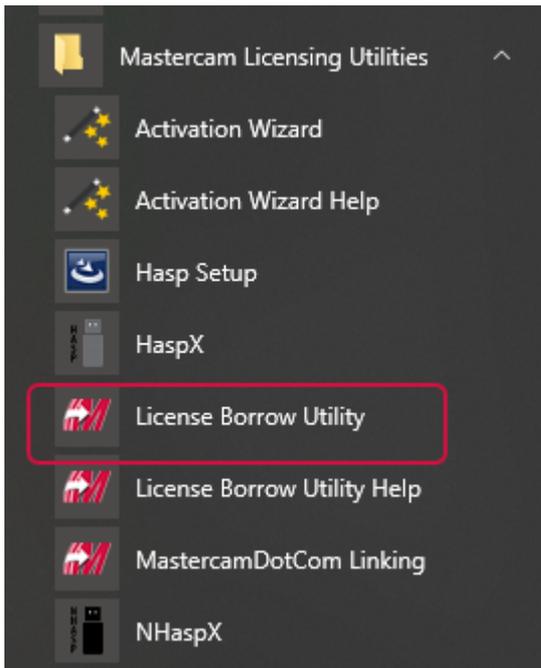
The new Mastercam License Borrow Utility allows you to use Mastercam outside of the workplace. You must be connected to the network that hosts the Mastercam licenses to borrow a license. After borrowing, you can disconnect from the network.

You can now easily borrow a Mastercam network license for a specified time period. You can return the license or trade it for a different Mastercam product as needed. To streamline your workflow, you can also borrow multiple Mastercam products simultaneously.

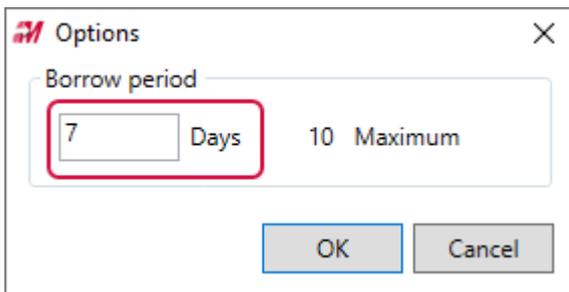


The License Borrow Utility requires that you exchange your current remote network license for a network borrow server license. This is a one-time task that your network administrator can perform by reactivating your Mastercam network license.

Launch the License Borrow Utility by opening it from the **Mastercam Licensing Utilities** folder in the Windows **Start** Menu.



Select the licenses you want to borrow, and then click **Borrow Licenses**. The default borrow period is customizable and can be accessed by clicking **File, Options**.



Licenses will automatically return to your network at the end of the borrow period. You can also choose to return a license early for others to use. Select the **Return** tab, select your license from the list, and click **Return Licenses**.

Expanding the Functionality of Mastercam Demo/Home Learning Edition

Mastercam Demo/Home Learning Edition (HLE) is Mastercam’s free CAD/CAM software for demonstration and educational purposes. It is a trial version of Mastercam that is meant for personal use.

In previous releases, Demo/HLE was a separate installation. In this release, Demo/HLE uses the same installation media as our core application in all installations of Mastercam or Mastercam for SOLIDWORKS that are governed by a software license. This version of Demo/HLE also includes more Mastercam products, add-ons, and functionality as described below.

These add-on products are now included with the new Demo/HLE license:

- Port Expert
- Blade Expert
- Productivity+™
- ProDrill (Mastercam only)
- Mill-Turn (Mastercam only)
 - Includes support for all Mill-Turn machine environments

Demo/HLE users who wish to use Mill-Turn can load any of the generic Fanuc machine environments that are installed with Mastercam 2022 without an additional license. The generic Siemens .machine files can be used with Demo/HLE, but these need to be downloaded from the [Tech Exchange](#).

Additionally, Mastercam 2022 Demo/HLE includes the following features:

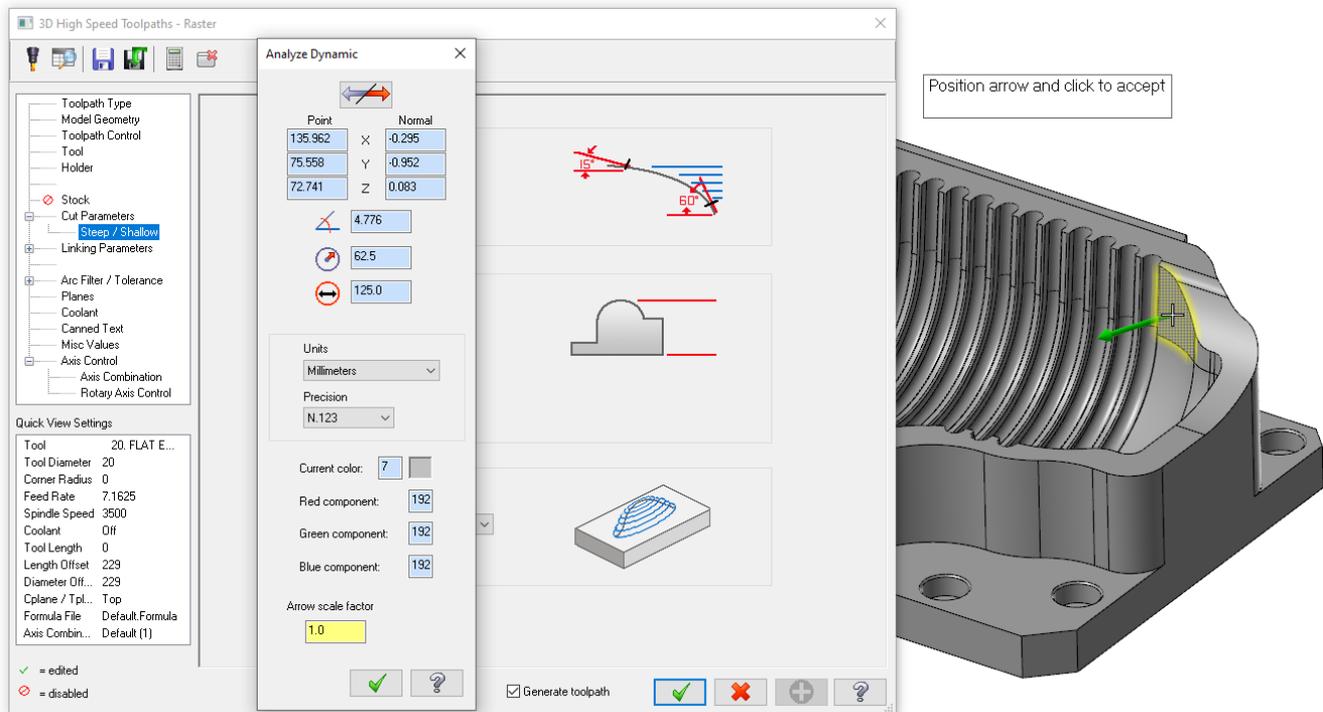
- Zip2Go
- Check for Updates
- Migration Wizard
- Batch toolpath processing
- Saving control definitions
- Network licensing

The following features are only available in the Mastercam version of Demo/HLE:

- Autosave
- Command-line arguments
- Export to STL and 3MF
- Wire auxiliary registers

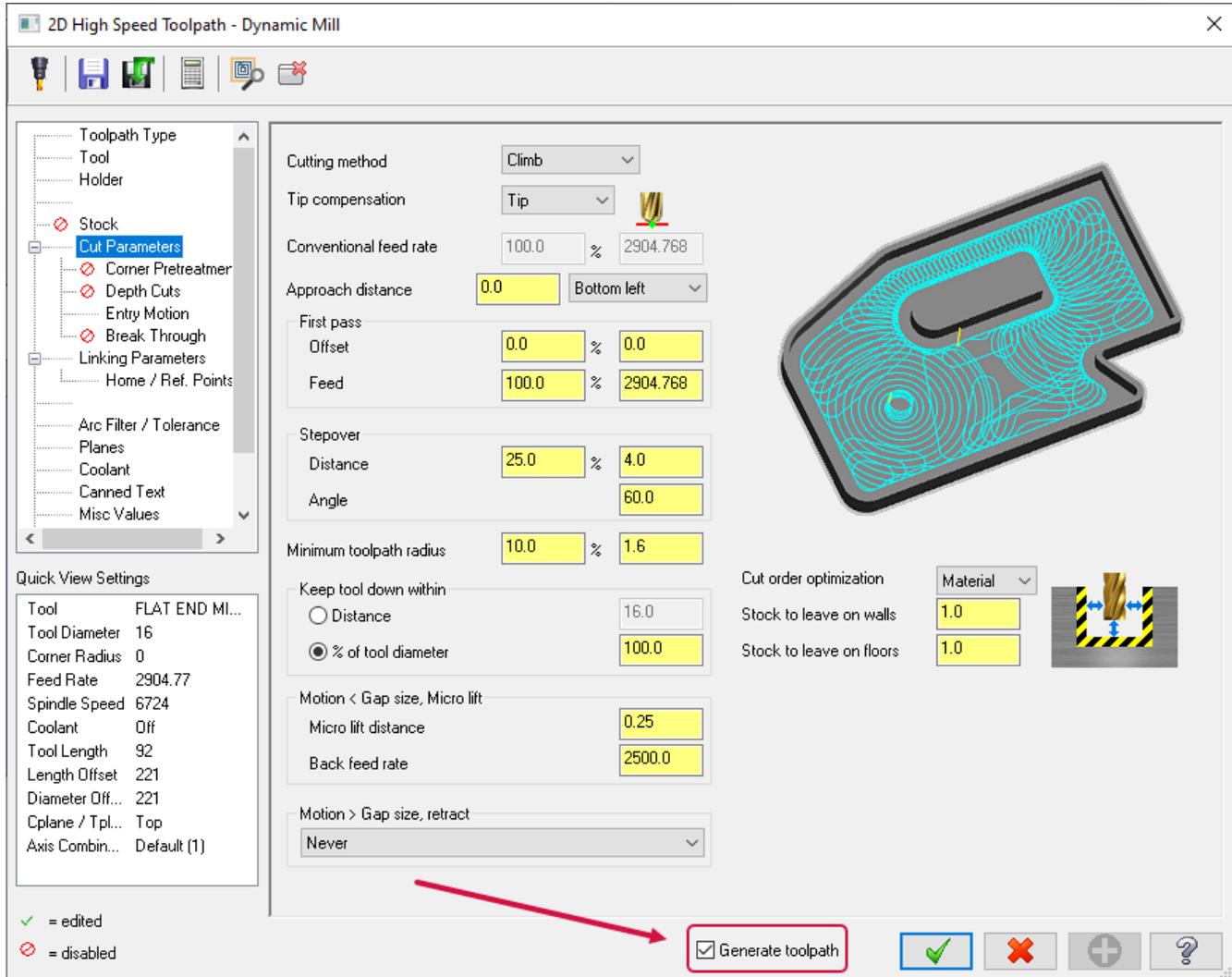
Interact with Mastercam with the Toolpath Dialog Box Open

While the toolpath dialog box is open, you can now interact with your part in the graphics window in Mastercam 2022. This includes the ability to adjust the view, make changes in the Planes Manager and Levels Manager, access the **Analyze** functions, and blank or hide entities. This helps you save time by gathering the necessary information from your part without closing and re-opening the toolpath dialog box. For example, you can use **Analyze Dynamic** to review your part while on the **Steep/Shallow** page of a 3D High Speed Raster toolpath.



Accepting and Generating a Toolpath with a Single Click

When creating or editing a toolpath, you can accept and generate the toolpath with a single click. In previous Mastercam versions, editing a toolpath would create a "dirty" operation that you would then need to regenerate. Mastercam 2022's **Generate toolpath** option automatically generates the toolpath when you click **OK** in the toolpath dialog box.



Generate toolpath applies to both new and edited operations. **To batch** settings are also respected. **Generate toolpath** is on by default for new and existing operations. Toolpaths in files loaded from previous versions of Mastercam do not have **Generate toolpath** enabled. To allow new or edited toolpaths to be marked "dirty" in the Toolpath Manager, deselect **Generate toolpath**. Your setting persists across instances of Mastercam, even after closing the program.

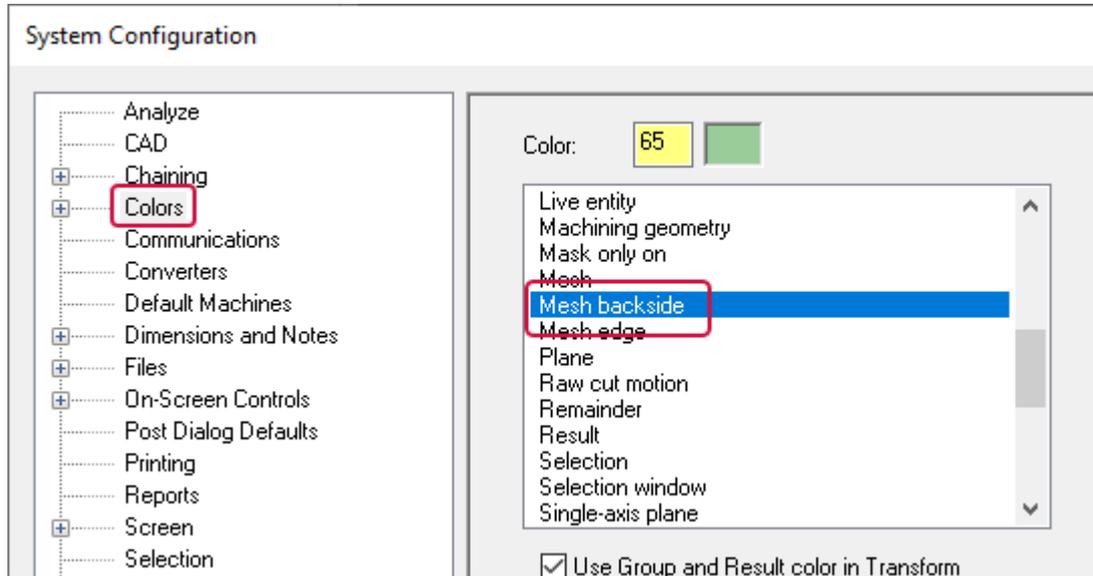
You can edit the default behavior for **Generate toolpath** on the **Toolpaths** page of the **System Configuration** dialog box.

System Configuration Enhancements

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

Customizing Mesh Color Settings

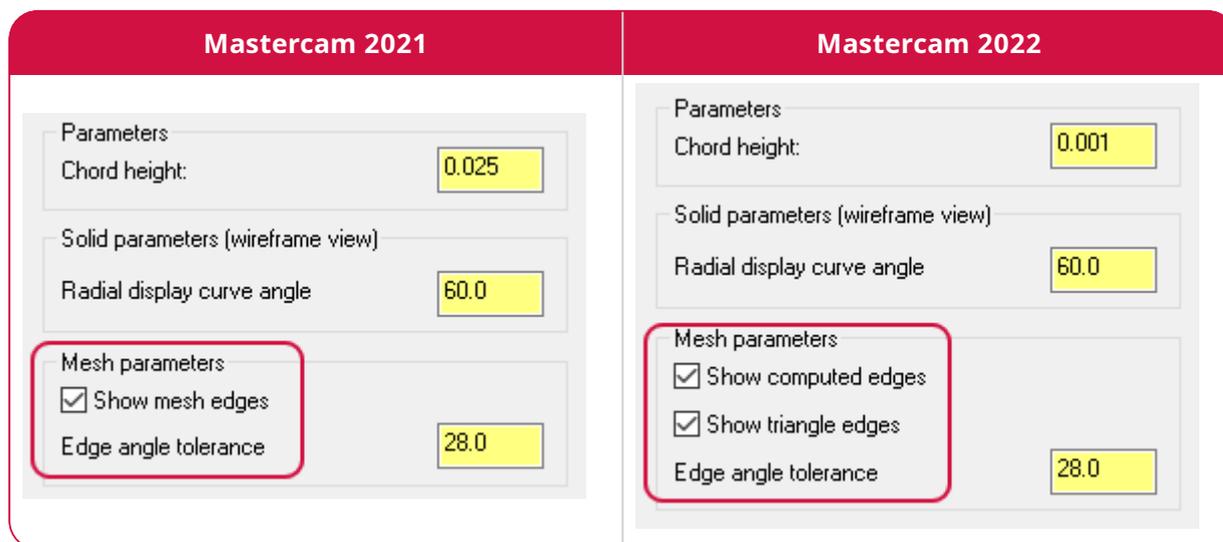
The new **Mesh backside** color option supports the new mesh features and aligns them with the functionality of surfaces.



This new color option allows you to see the back (positive) side of a mesh when **Backside** on the **View** tab is enabled. **Mesh backside** and **Surface backside** use the same default color found in the **Colors** page of the **System Configuration** dialog box.

Showing Mesh Edges with a Greater Degree of Control

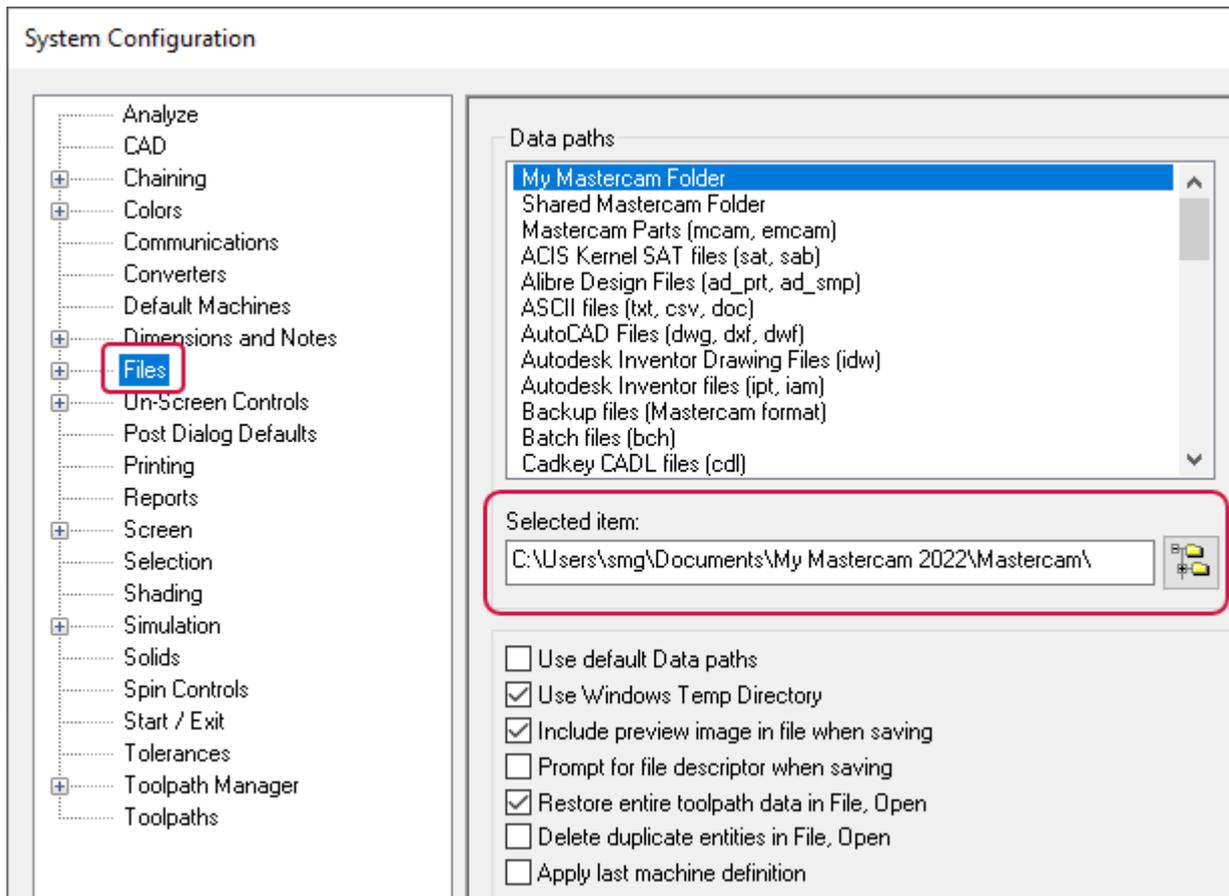
The **Mesh parameters** group, located in the **Shading** page of the **System Configuration** dialog box, has been enhanced to give you greater control over how polygonal meshes (STLs) are displayed in **Outline Shaded** display mode.



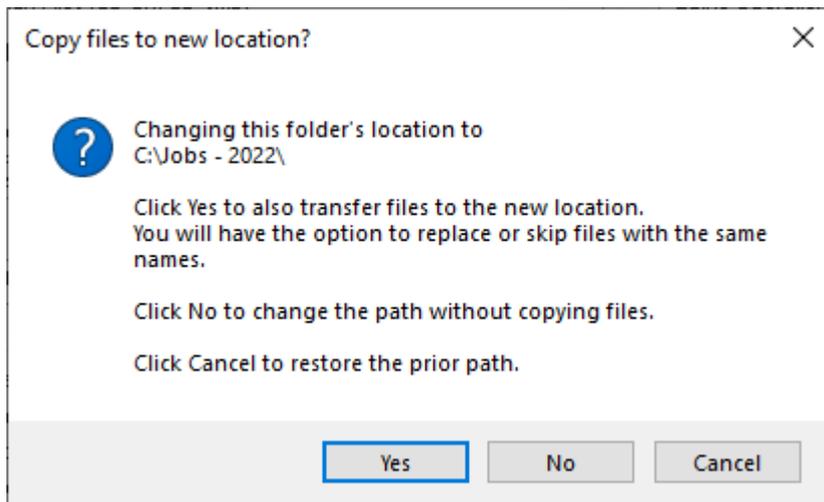
Previously, the **Show mesh edges** option created edges that were "synthetic" because they were computed by comparing the angles of adjacent polygons. The STL format does not define edges. An edge was generated if the angle was greater than the **Edge angle tolerance**. Individual triangle edges were shown when the setting was deselected. This enhancement allows you to see both computed edges and triangle edges at the same time, instead of one or the other.

Moving a Shared Folder Without Copying Files

File management in Mastercam 2022 has been enhanced. Previously, when you moved a data path location, you had two options for files that had the same name in both locations: replace them or skip them from being copied.

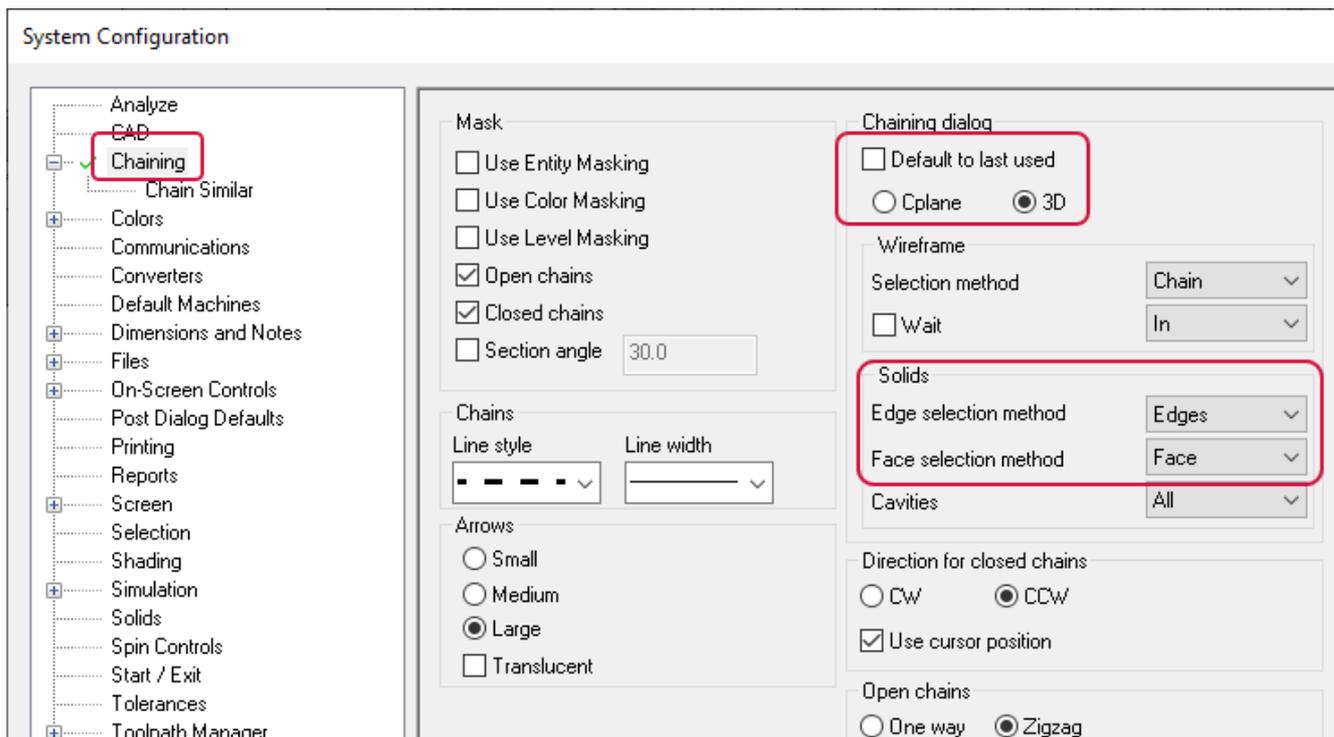


Now Mastercam allows you to opt out of copying any files to the new data path location. This feature is useful when multiple users share a network folder. A message appears that lets you opt out of copying files after you set a new file path in the **Files** page of the **System Configuration** dialog box.



Setting Chaining Mode Defaults

Mastercam 2022 allows you to set the chaining dialog box defaults, with the option to default to the last mode used, in the **Chain dialog settings** group of the **Chaining** page. Two new options for solids have also been added to give you greater control over your default chain dialog settings: **Edge selection method** and **Face selection method**.

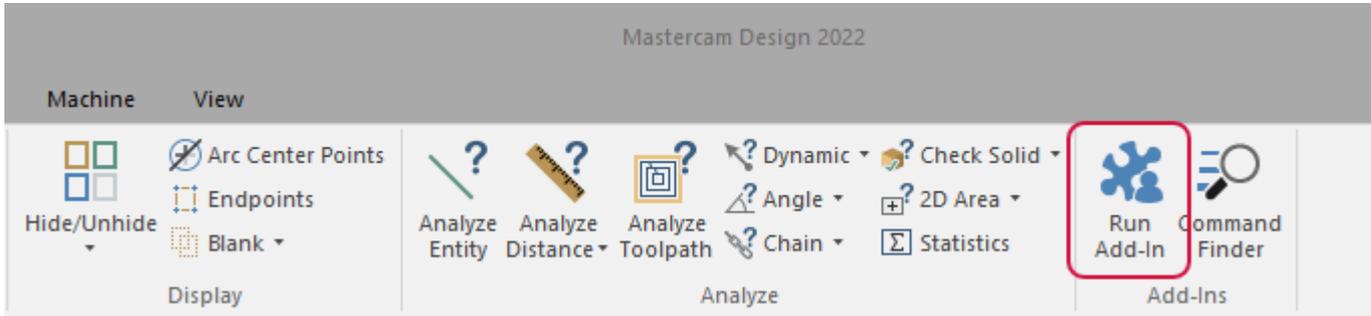


Creating and Editing .NET Scripts in Mastercam

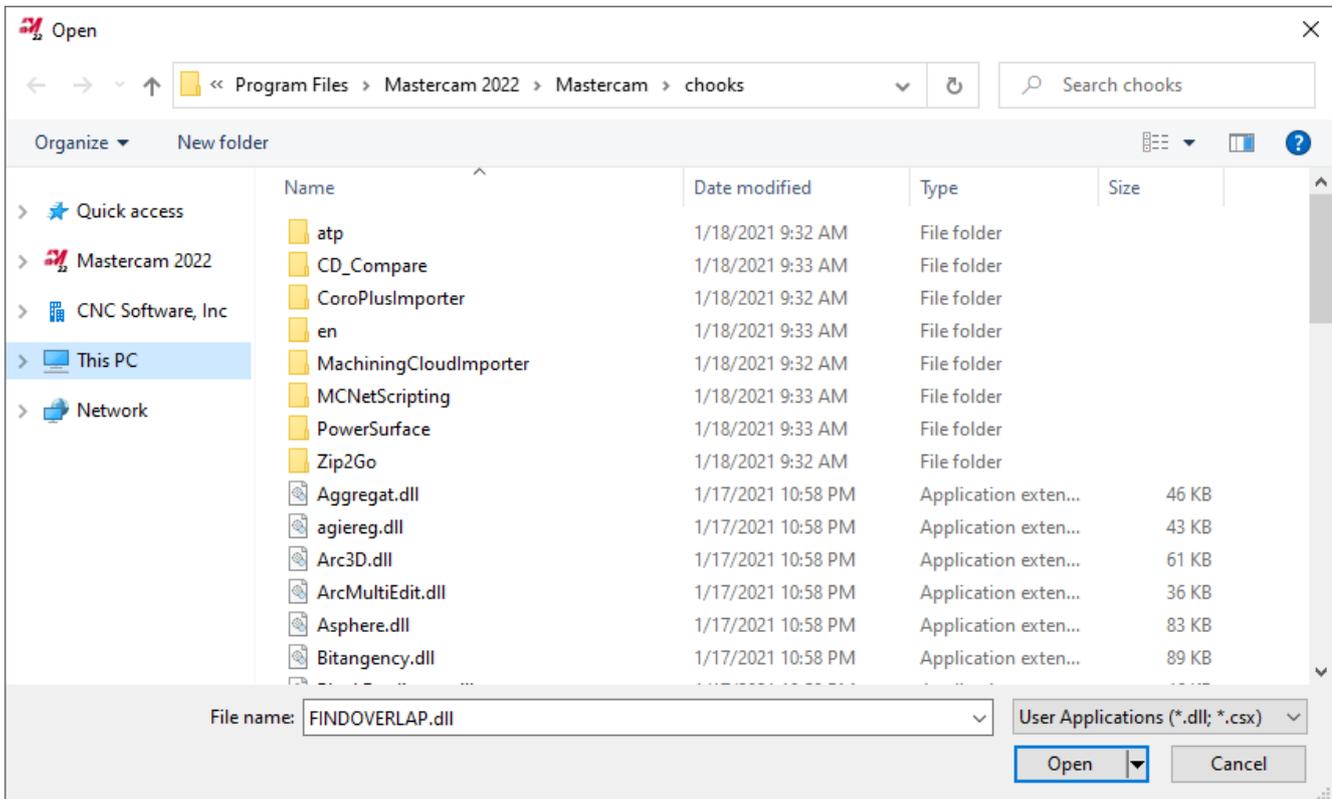
Mastercam's new NET-Script Editor highlights a suite of improvements for script developers. Use it to create, edit, compile, and run .NET scripts within Mastercam. Developers can access the entire .NET API and even the entire .NET libraries.

The NET-Script Editor includes built-in error checking when you compile or run your script. Double-click on an error to jump to that line in your code. You can add the NET-Script Editor to the Mastercam interface, such as the Quick Access Toolbar.

In addition to the ability to load and run scripts from within the NET-Script Editor, Mastercam 2022 also lets you run them directly by selecting **Run Add-In** on the **Home** tab.



After selecting **Run Add-In**, you can select and load either a DLL or script. The default location for add-ins and other resources that appear is been moved to `..My Mastercam 2022\Mastercam\chooks`.



Developers with a valid My Mastercam account can access additional documentation at <https://nethookdocs.mastercam.com>.

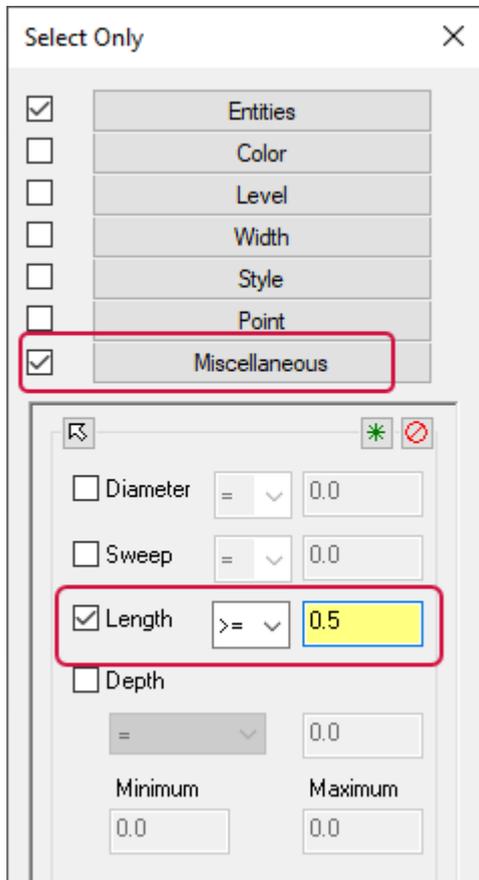
In connection with the new NET-Script Editor, Mastercam’s existing VB script functionality is being deprecated as of Mastercam 2022:

- VB scripting and the VB Script Manager are still present in Mastercam 2022.
- They will not be included in Mastercam 2023 but will be available upon request.

- They will not be available in Mastercam 2024.

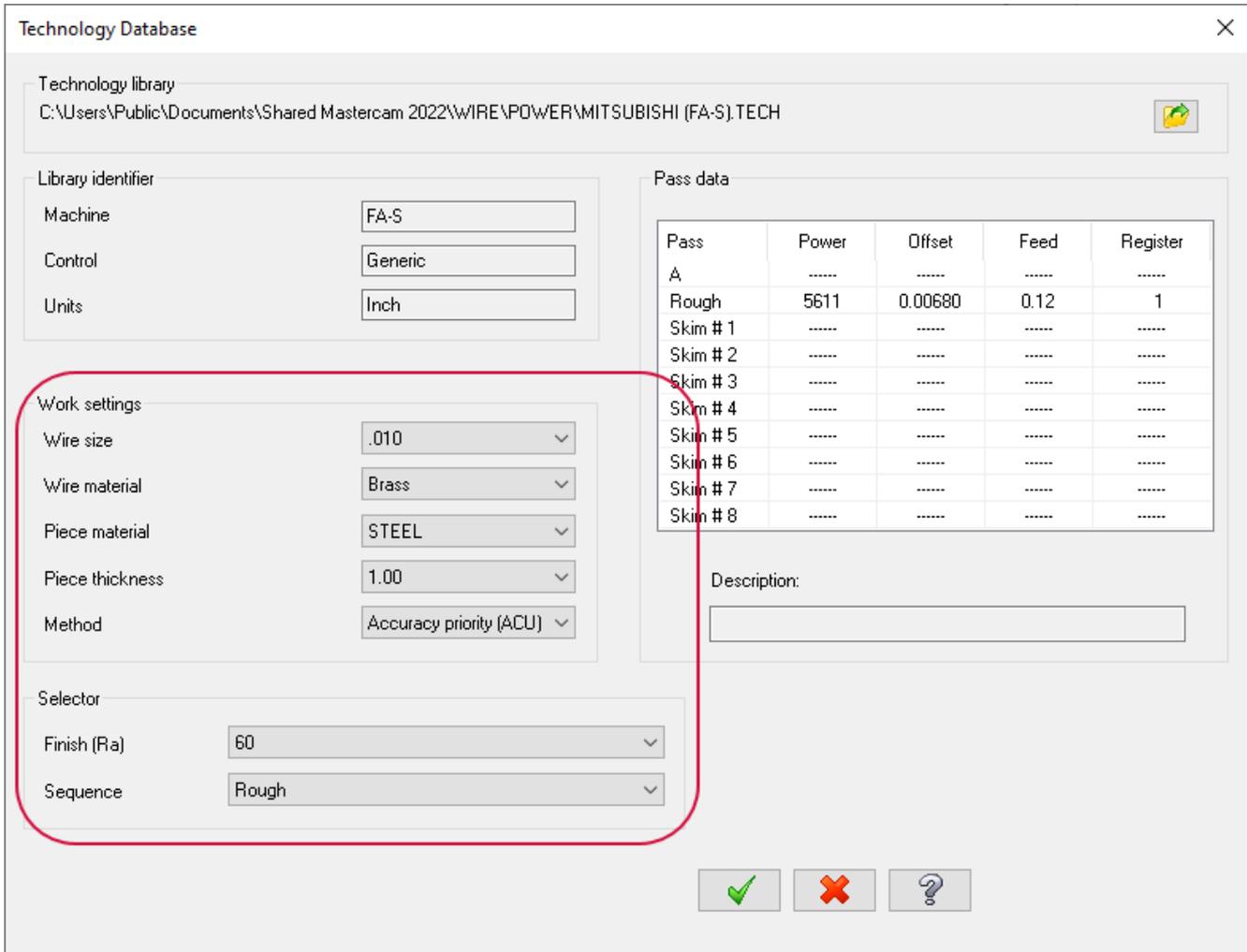
Finding Arcs and Splines by Length

Previous versions of Mastercam allowed you to use **Quick Mask** to select lines by their **Length**. Mastercam 2022 extends this ability to splines and arcs. Using the **Select Only Quick Mask**, select **Miscellaneous** in the **Select Only** dialog box. Then, use the **Length** option to find arcs, lines, and splines.



Retaining Wire Work Settings

When programming a Wire machine using a TECH-based Machine, the **Technology Database** dialog box values in the **Work settings** and **Selector** groups now remain active so that you no longer have to reselect them each time you create a new operation.



POSTS AND MACHINE ENVIRONMENTS

Listed below are the new post and machines for Mastercam 2021 and can be downloaded from the Mastercam Tech Exchange website (<https://community.mastercam.com/techexchange>).

Lathe Machine Environments

The following machine environments are now available for Mastercam 2021 Lathe. These machines can be run by a user with a Lathe and Mill licenses. They do not require a full Mill-Turn product license.

NOTE

Posts that are compatible with Mastercam 2022 will be uploaded to the Tech Exchange when they are available. Check the [Tech Exchange](#) for the latest updates.

Machine	Control	Configurations
Spinner TC 800-85 MCY	Siemens 840D	Single Spindle Single Turret Tailstock
DMG Mori Seiki NLX <ul style="list-style-type: none"> • NLX3000MC 3000TSY_12st BOT • NLX2000MC 500TSY_12st BOT - Gen2 • NLX3000MC 1250TSY_12st BOT • NLX1500MC 500TSY_12st BOT - Gen 2 • NLX1500MC 500TSY_20st BOT - Gen 2 • NLX2500MC 1250TSY_12st BOT 	Mitsubishi (CELOS)	Single Spindle Single Turret Tailstock
Doosan Lynx <ul style="list-style-type: none"> • 220LSYC • 2600SY_BMT55x24 	Fanuc i series	Dual Spindle Single Turret

Machine	Control	Configurations
<p>Doosan PUMA simple lathes</p> <ul style="list-style-type: none"> • 2100SY_BMT55 • 2100SY_BMT55x24 • 2100SY_BMT65 • 2100SY_BMT65x24 • 2600MS_BMT65x24 • 2600SYBII_BMT65x24 • 2600SY_BMT65 • 2600SY_BMT65x24 • 2600Y_BMT65 • 2600Y_BMT65x24 	<p>Fanuc i series</p>	<p>Dual and Single Spindle Single Turret</p>
<p>Haas ST v2</p> <ul style="list-style-type: none"> • Haas ST-20 Y_C_BMT65_v2 • Haas ST-20 Y_S_BMT65_v2 • Haas ST-20_C_BOT_v2 • Haas ST-20_C_VDI40_v2 • Haas ST-20_S_BMT65_v2 • Haas ST-20_C_BMT65x24_v2 • Haas ST-20_S_BMT65x24_v2 • Haas ST-20 Y_C_BMT65x24_v2 • Haas ST-20 Y_S_BMT65x24_v2 • Haas ST-15 Y_S_Hybrid_v2 	<p>Haas CNC</p>	<p>Dual and Single Spindle Tailstock Single Turret</p>
<p>DMG Mori Seiki</p> <ul style="list-style-type: none"> • NLX3000MC • 1250TSY_12st BOT 	<p>Mitsubishi (CELOS)</p>	<p>Single Spindle Single Turret Tailstock</p>
<p>DMG Mori Seiki</p> <ul style="list-style-type: none"> • NL2000SY 500 • NL2000Y 	<p>Fanuc 31i-A</p>	<p>Dual Spindle Single Turret</p>

Machine	Control	Configurations
DMG Mori Seiki <ul style="list-style-type: none"> NLX1500MC 500TSY_12st BOT - Gen 2 DMG Mori Seiki NLX1500MC 500TSY_20st BOT - Gen 2 	Mitsubishi (CELOS)	Single Spindle Single Turret Tailstock
Nakamura-Tome SC-300II_MSyX24	Fanuc 0i-TD	Dual Spindle Single Turret
Mazak Quick Turn <ul style="list-style-type: none"> Mazak Quick Turn 250MSY Mazak Quick Turn Universal 350MY 500U 	Nexus 2	Dual Spindle Single Turret

Mill-Turn Machine Environments

The following machine environments are now available for Mastercam Mill-Turn. These machines require a full Mill-Turn product license. Mill and/or Multiaxis licenses might also be required to access advanced Milling and Multiaxis features. You must request these machine environments from your Mastercam Reseller.

NOTE

Posts that are compatible with Mastercam 2022 will be uploaded to the Tech Exchange when they are available. Check the [Tech Exchange](#) for the latest updates.

Machine	Control	Configurations
Index RatioLine G220 VDI25x18	Index C200 SL	Dual Spindle Tool Spindle Dual Turret
Okuma LU7000 EX-MY 2SC x2000_V10W	OSP-P300L	Single Spindle Dual Turret Tailstock

Machine	Control	Configurations
Doosan PUMA TT Series <ul style="list-style-type: none"> • 1300SYY • 1300SYYB • 2100SYY • 2100SYYB 	Fanuc 31i (A5 and B5)	Dual Spindle Dual Turret Lower Turret
Doosan PUMA TT1800SY	Siemens 840D	Dual Spindle Dual Turret
Nakamura Tome NTRX-300L_S_S-Type	Fanuc 35i-B5	Dual Spindle Tool Spindle
DMG Mori Seiki NZX4000 3000Y	Fanuc 31i-A	Dual Spindle Dual Turret
DMG Mori Seiki NTX2000 1500SZY – Gen 2	Fanuc 31i-A	Dual Spindle Dual Turret B-axis
DMG Mori Seiki CTX <ul style="list-style-type: none"> • CTX beta 1250 TC_S – Gen 1 • CTX beta 800 4A_S • CTX beta 1250 4A_S • CTX beta 1250 TC 4A 	Siemens 840D / DMG Structure	Dual Spindle Single Turret Dual Turret B-axis
Nakamura Tome NTY3-150	Fanuc 31i-B	Dual Spindle Triple Turret
Kovosvit MAS Multicut 500i S	Siemens 840D	Dual Spindle Single Turret B-axis
Eurotech Trofeo B465-SY2	Fanuc 31i-TT	Dual Spindle Dual Turret

In addition, the following component libraries are available, containing chucks and jaws. These can be used with either Mill-Turn or Lathe machine environments. These machines can be downloaded from the [Mastercam Tech Exchange](#) website.

Type	Maker	Details
Chuck and Jaws	Kitagawa	B200 Series
		BB200 Series
		BR Series
Chuck and Jaws	Royal	Royal Quick-Grip Collet Chucks

Multiaxis Posts

The following multiaxis posts are now available Mastercam 2021. These machines can be downloaded from the [Mastercam Tech Exchange](#) website.

NOTE

Posts that are compatible with Mastercam 2022 will be uploaded to the Tech Exchange when they are available. Check the [Tech Exchange](#) for the latest updates.

Machine	Control	Configurations
Siemens 5X Mill	Siemens 840D	Table-table AC
		Table-table BC
DMG DMU 50 Gen2	Siemens 840D	Table-table BC
Heidenhain TNC 5X Mill	TNC530, 620, 640	Various

Wire Posts

The following wire posts are available for GFMS CUT Series 4-axis machines. The versions listed in the table below are available for download; however, other versions can be requested (2.8.1, 2.8.0, 2.7.1, 2.7.0, 2.5.1, 1.3.7, and 1.3.5).

NOTE

Posts that are compatible with Mastercam 2022 will be uploaded to the Tech Exchange when they are available. Check the [Tech Exchange](#) for the latest updates.

Machine	Control	Versions
GFMS CUT P1250	HMI2	2.8.4
		2.8.5
		2.8.2
GFMS CUT P800	HMI2	2.8.4
		2.8.5
		2.8.2
GFMS CUT P550	HMI2	2.8.4
		2.8.5
		2.8.2
GFMS CUT P350	HMI2	2.8.4
		2.8.5
		2.8.2
GFMS CUT E600	HMI2	2.8.4
		2.8.5
		2.8.2
GFMS CUT C600	HMI2	2.8.4
		2.8.5
		2.8.2
GFMS CUT C350	HMI2	2.8.4
		2.8.5
		2.8.2
GFMS CUT E350	HMI2	2.8.4
		2.8.5
		2.8.2

**ATTENTION! UPDATES MAY BE AVAILABLE.
PLEASE REFER TO [MASTERCAM.COM/SUPPORT](https://www.mastercam.com/support)
FOR THE LATEST DOWNLOADS.**

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



www.mastercam.com

Mastercam®

Mastercam® is a registered trademark of CNC Software, Inc. All other trademarks are property of their respective owners.
©1983-2021. All rights reserved.