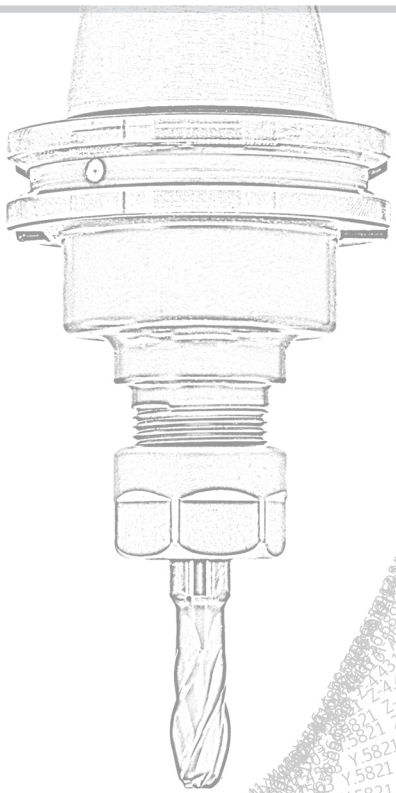
Mastercam®



What's New in Mastercam 2023

MARCH 2022

WHAT'S NEW IN MASTERCAM 2023

March 2022

Software: Mastercam 2023

Terms of Use

Be sure you have the latest information!

Information might have changed or been added since this document was published. The latest version of the document is installed with Mastercam or can be obtained from your local Reseller. The Resolved Issues website—installed with each release—includes the latest information about Mastercam features and enhancements.

TABLE OF CONTENTS

INTRODUCTION

Welcome to What's New in Mastercam 2023! Mastercam 2023 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:

- "Detecting Undercut Stock When Machining" on page 15
- "Consolidating Multiaxis Toolpaths " on page 27
- "New B-Axis Contour Turning Toolpath" on page 35
- "Check Mesh Enhancements" on page 57
- "Appending Operations to a Current Mastercam Simulator Session " on page 83
- "Adding Stock Awareness to Holder Checking" on page 24
- "New Workflow and Interface for Managing Your Machine Groups " on page 103
- "Creating Consistent Flowlines with Surface Ruled/Lofted" on page 72
- "Creating Chamfers with Push-Pull " on page 71

Mastercam Resources

Enhance your Mastercam 2023 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 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 our many 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 (https://www.instagram.com/mastercam/) 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, LLC, 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.

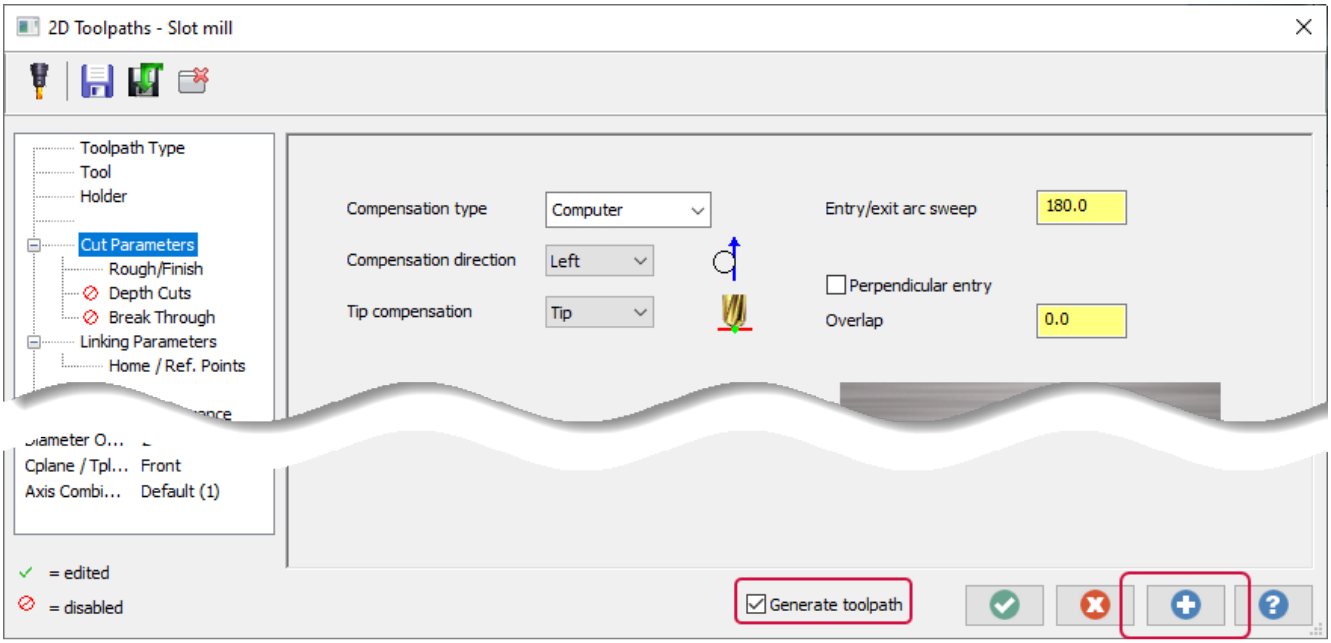
MILLING ENHANCEMENTS

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.

New Behavior for the Apply Button in Milling Toolpaths

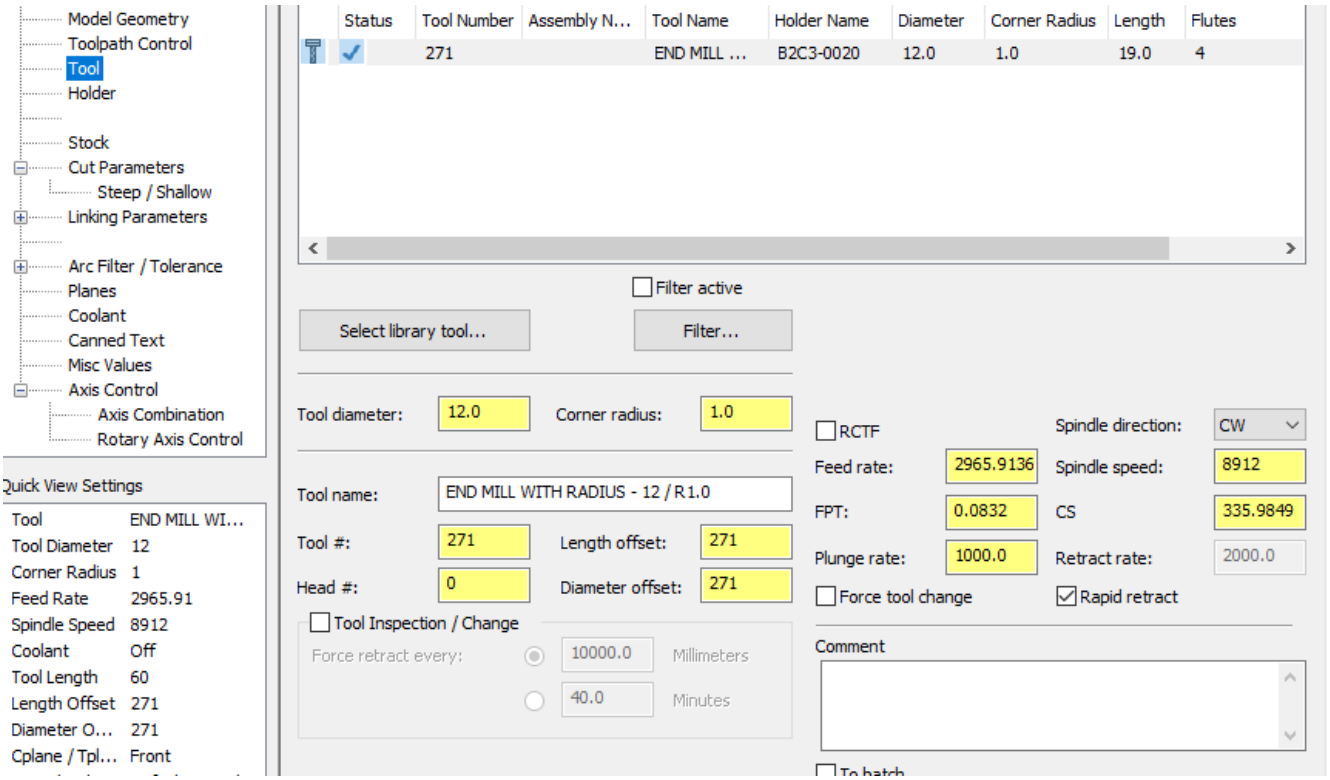
The **Apply** button for tree-style milling toolpaths has been modified to be similar to the **OK** button. When you click **Apply**, Mastercam processes and applies your toolpath changes without closing the dialog box. If **Generate toolpath** is selected, your changes are applied to the toolpath and the toolpath is automatically regenerated. If **Generate toolpath** is deselected, your toolpath is marked as dirty in the Toolpaths Manager to regenerate at a later time.

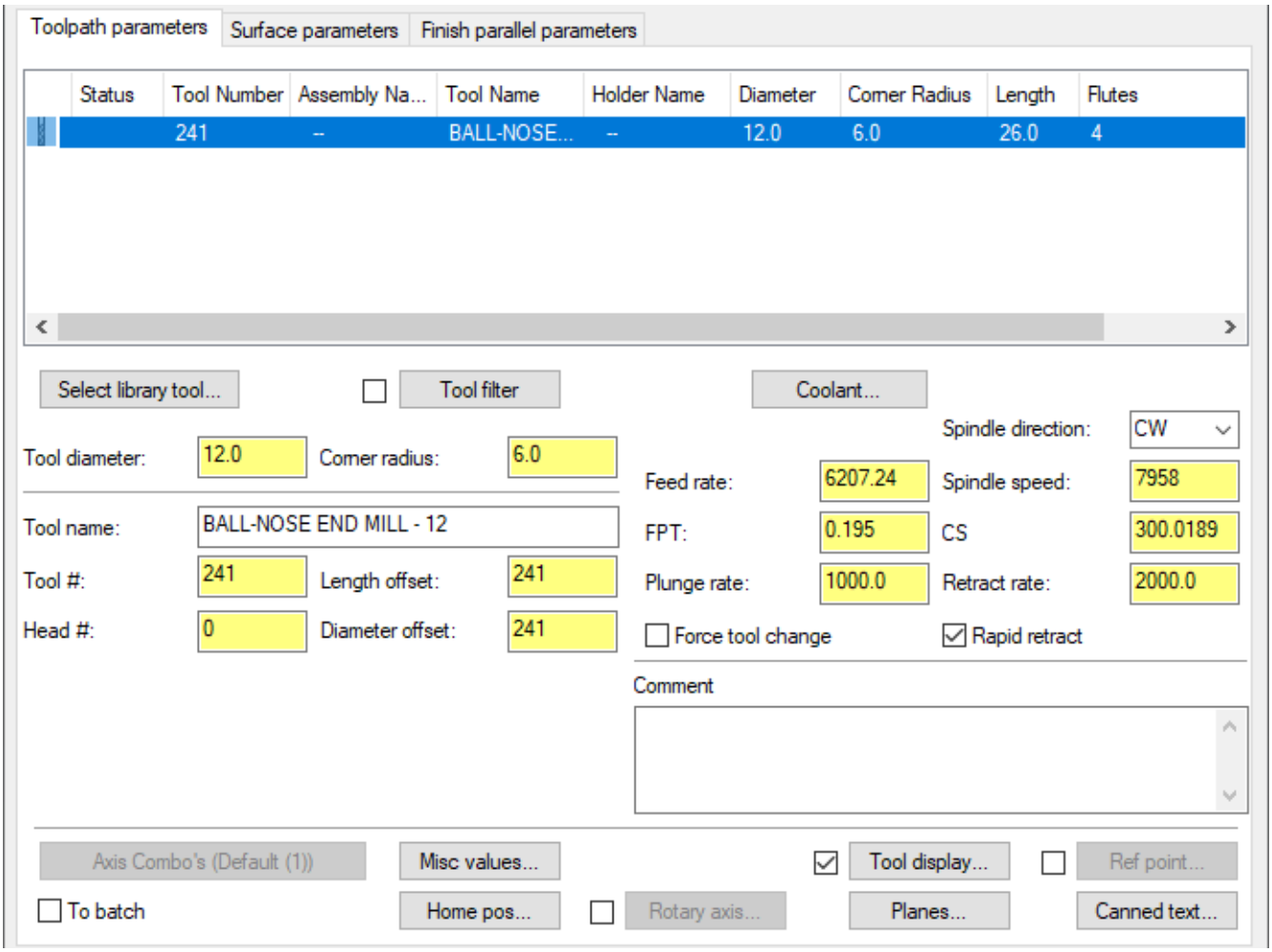


Due to this change, the **Preview** button is no longer necessary and has been removed from the affected toolpaths.

Re-designed Tool Pages and Tabs for Milling Toolpaths

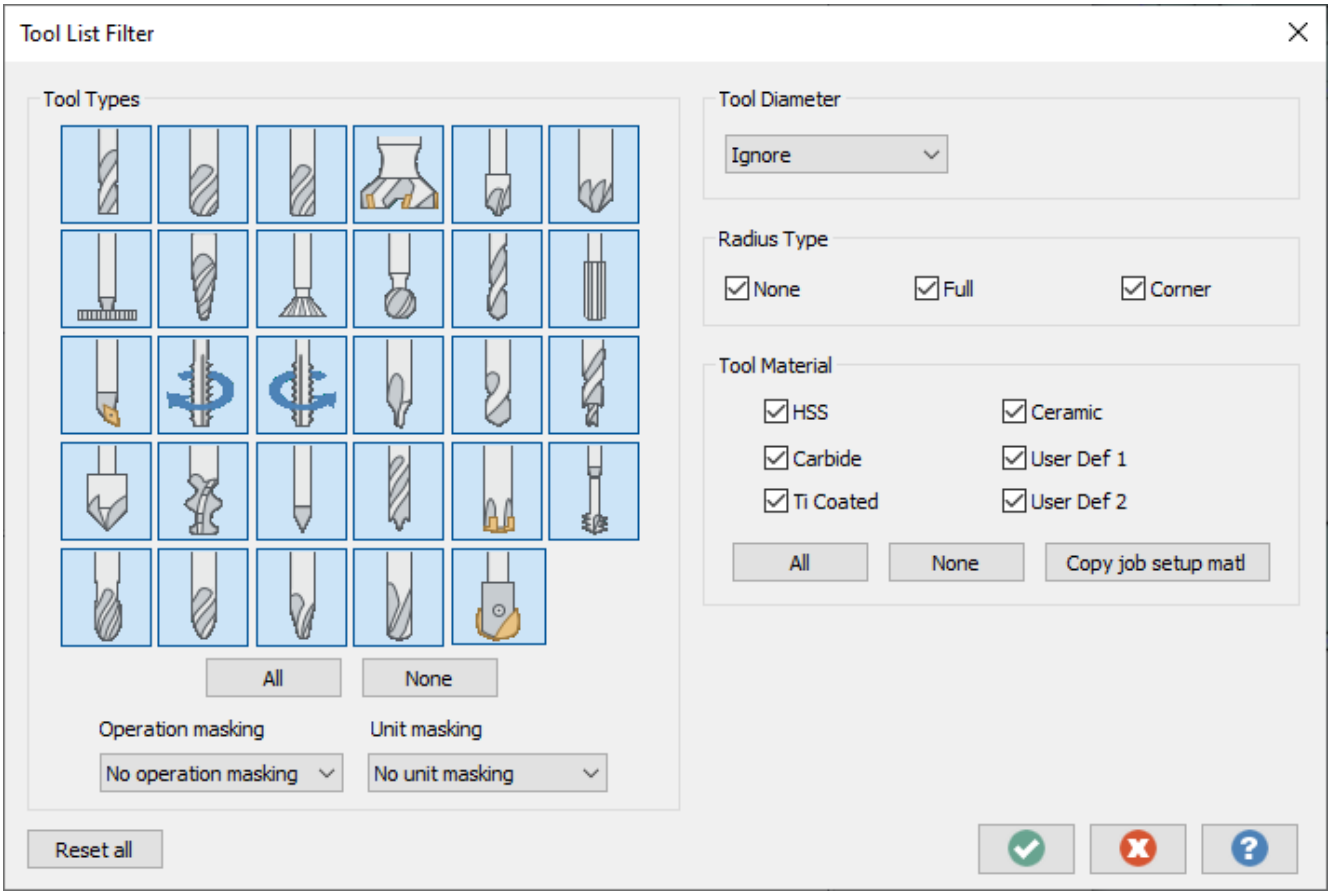
The **Tool** page and **Toolpath parameters** tab for milling toolpaths have been redesigned to provide a better, user-friendly layout.




Toolpath parameters
Surface parameters
Finish parallel parameters
Status
Tool Number
Assembly Na...
Tool Name
Holder Name
Diameter
Corner Radius
Length
Flutes
241
--
BALL-NOSE...
--
12.0
6.0
26.0
4
Select library tool...
Tool filter
Coolant...
Spindle direction:
CW
Tool diameter:
12.0
Corner radius:
6.0
Feed rate:
6207.24
Spindle speed:
7958
Tool name:
BALL-NOSE END MILL - 12
FPT:
0.195
CS
300.0189
Tool #:
241
Length offset:
241
Plunge rate:
1000.0
Retract rate:
2000.0
Head #:
0
Diameter offset:
241
Force tool change
Rapid retract
Comment
Axis Combo's (Default (1))
Misc values...
Tool display...
Ref point...
To batch
Home pos...
Rotary axis...
Planes...
Canned text...

New Mill Tool Icons

The new Mill tool icons better express tool functionality and contribute to a more realistic tooling experience in Mastercam 2023. You can view these icons while working in the **Tool List Filter** dialog box, the Tool Wizard, or the stand-alone Tool Manager.



Overall Process Improvements

Listed below are general improvements made to Mill toolpaths and processing times.

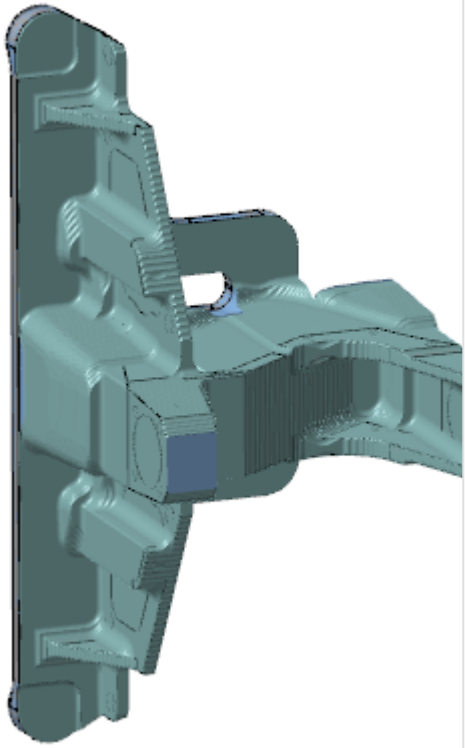
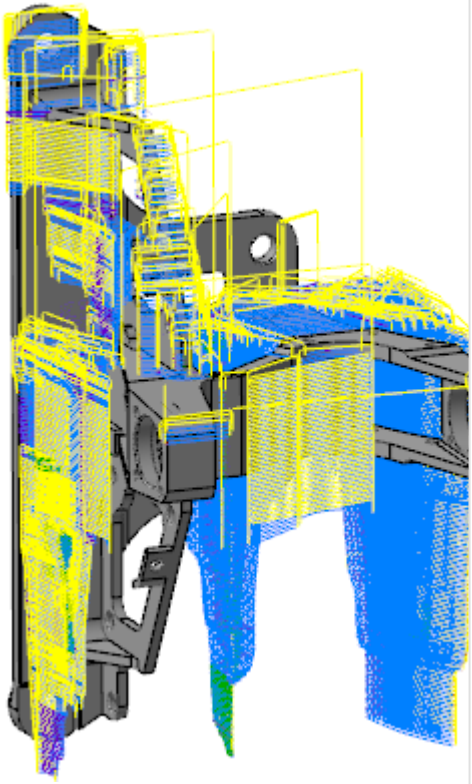
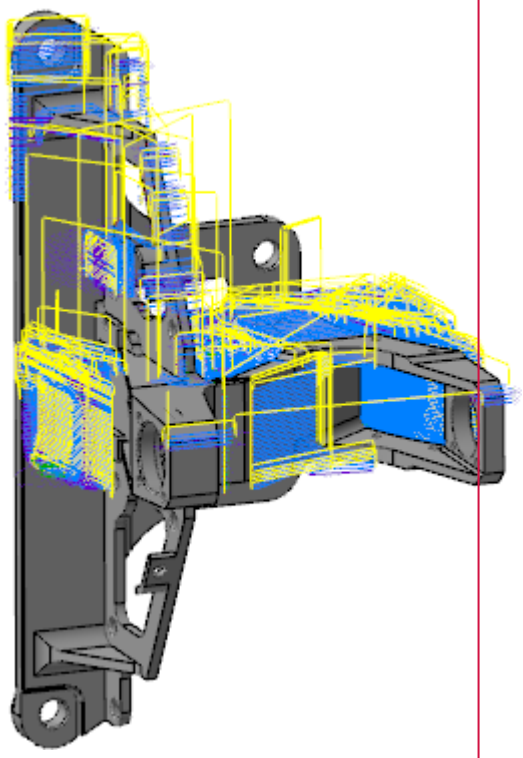
- Equal Scallop's processing time has improved when making changes to existing operations.
- General improvements have been made to the linking steps for 3D high speed toolpaths, improving processing time and quality of motion.

3D Toolpath Enhancements

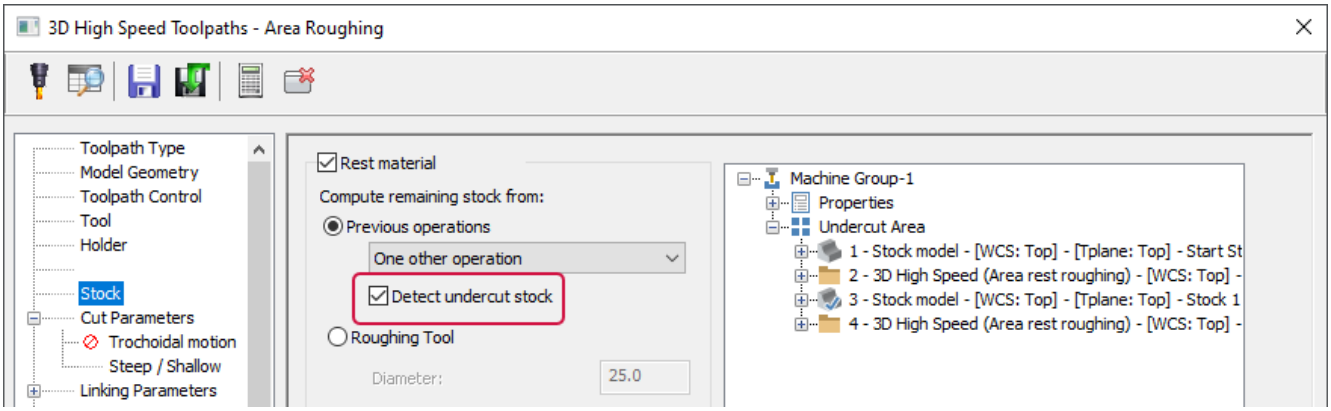
Listed below are enhancements to 3D milling toolpaths.

Detecting Undercut Stock When Machining

The Dynamic OptiRough and Area Roughing toolpaths can now be aware of undercut stock conditions resulting in improvement to the toolpath motion including less air cutting.

Stock Model	Detect Undercut Stock Disabled	Detect Undercut Stock Enabled
		

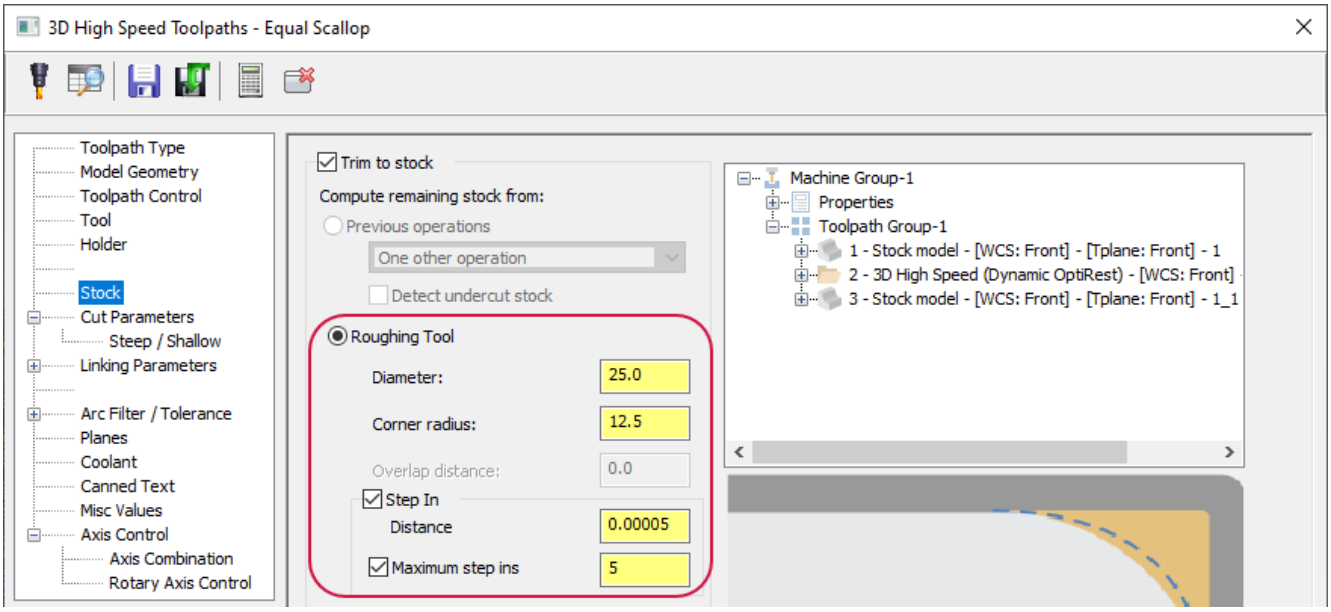
This new **Detect undercut stock** option is located on the **Stock** page and is only available when **Previous operations** is set to **One other operation**.



Identifying and Machining Material in Corners

The Equal Scallop toolpath now identifies material left in corners and will machine it with a single pass or with multiple passes to ease into the material. The new **Step in** option allows you to control the removal of material by the step ins you define. **Distance** specifies the amount of material to remove with each step in. **Maximum step ins** specifies the maximum number of step ins. The actual number of step ins performed might be less than the maximum specified here.

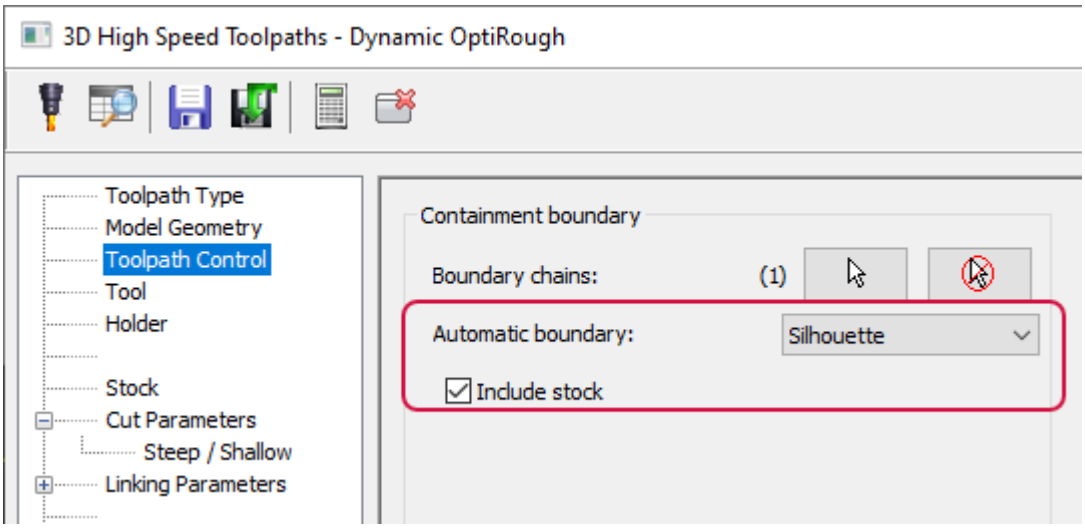
The **Step in** options are for semi-finishing applications and not intended for finishing. If Mastercam does not detect enough stock to engage, then no more passes are created. Mastercam leaves small amounts of stock so that small segments of motion are not created.



Creating Automatic Boundaries for Toolpath Containment

A new option, **Automatic boundary**, has been added to the **Toolpath Control** page for 3D high speed toolpaths. This new option allows Mastercam to create a boundary around the selected **Machining geometry** and uses it as a containment boundary in addition to any selected **Boundary chains**. The new **Include stock** option adds the stock settings from the **Stock** page to the selected **Automatic boundary**. You can select from the following options:

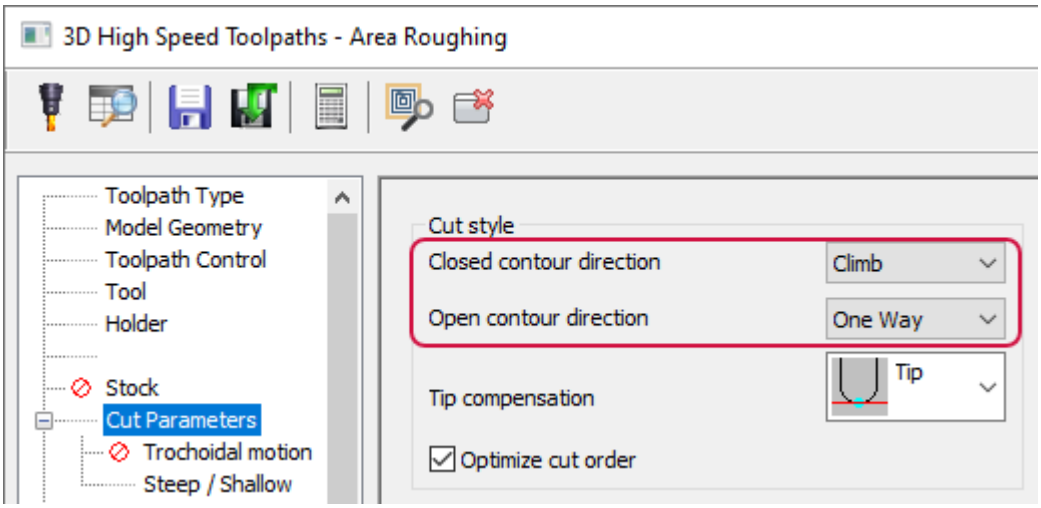
- **None**—No boundary is created. The **Include stock** option is automatically applied.
- **Silhouette**—Creates a boundary that is a silhouette of the selected **Machining geometry** and **Boundary chains**.
- **Bounding rectangle**—Creates a rectangle as a boundary.
- **Bounding circle**—Creates a circle as a boundary.



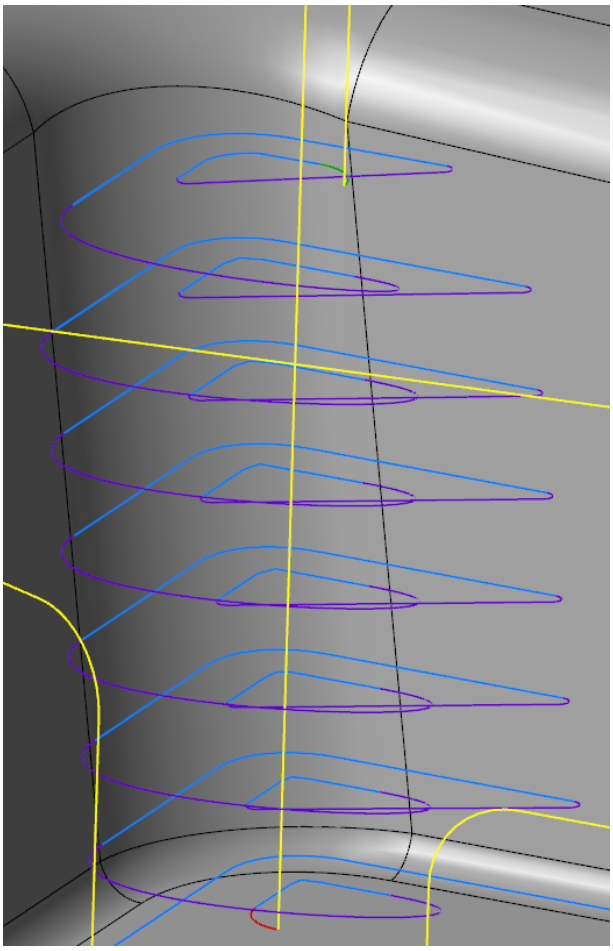
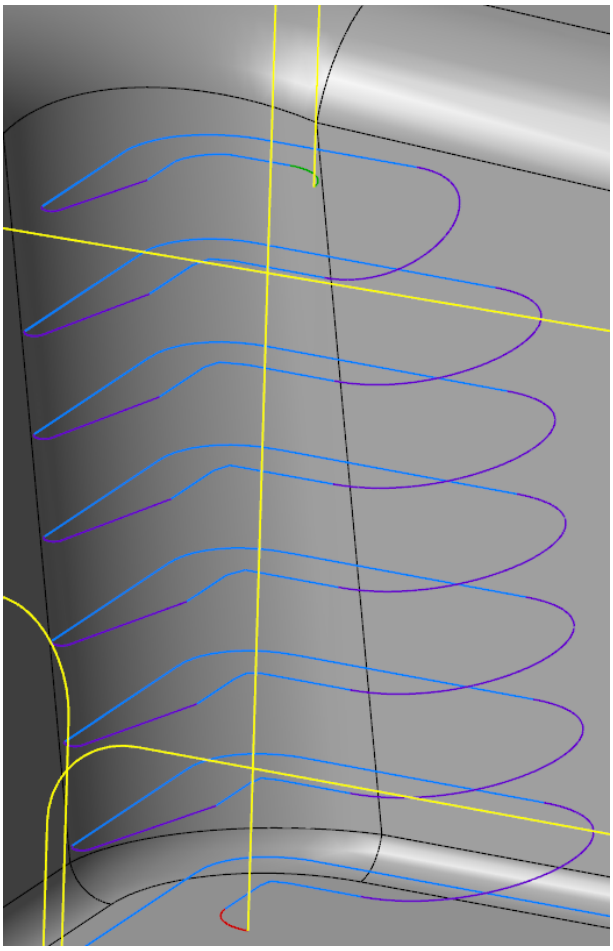
Due to these changes, the **Silhouette boundary** option has been removed and added to the **Automatic boundary** drop-down as **Silhouette**.

Machining with Zigzag Motion

The **Cut Method** drop-down on the **Cut Parameters** page for Area Roughing toolpaths 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.

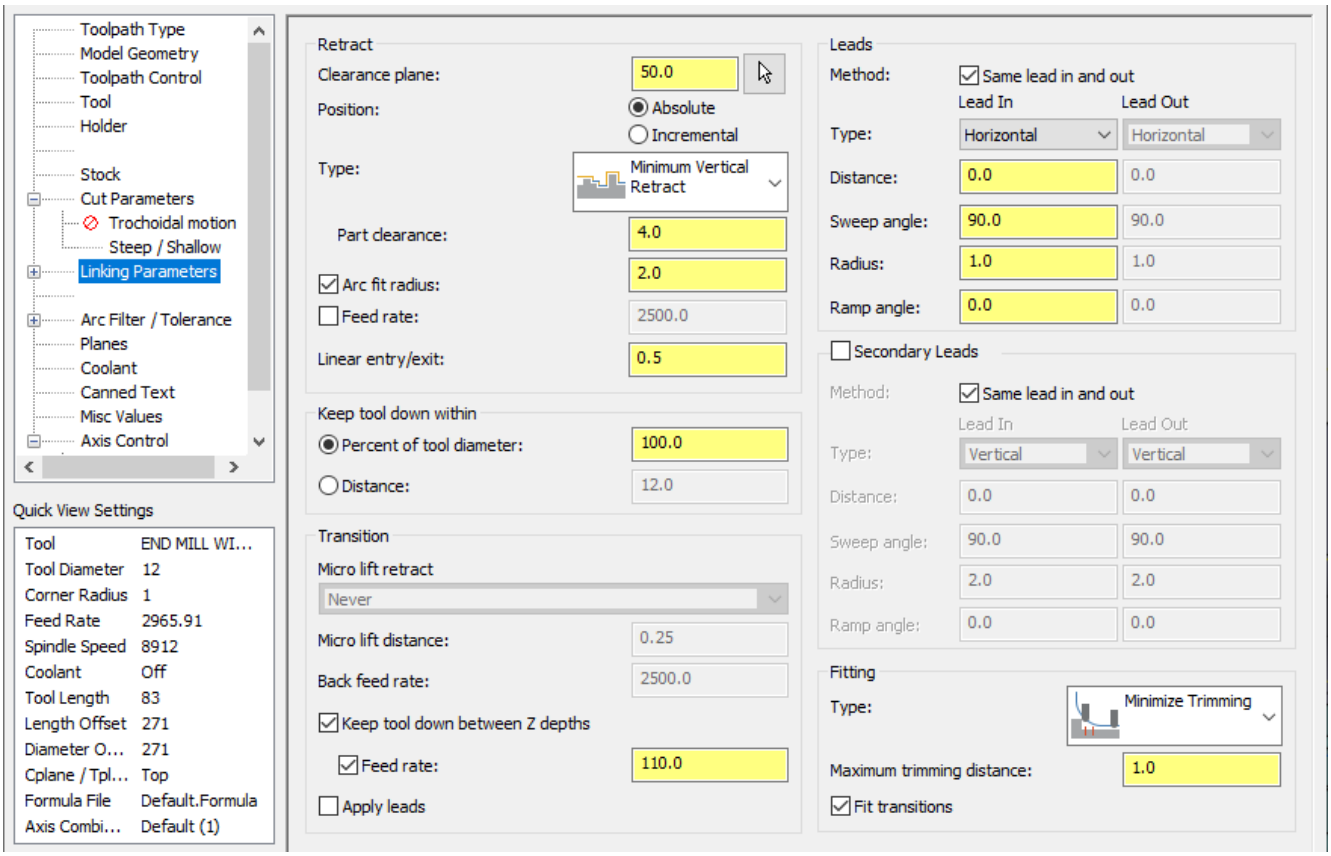


Area Roughing also includes a new **Open contour direction** option, **Zigzag**. **Zigzag** cuts each pass in the opposite direction to the previous pass. A short linking motion connects the two ends.

One Way	Zigzag
	

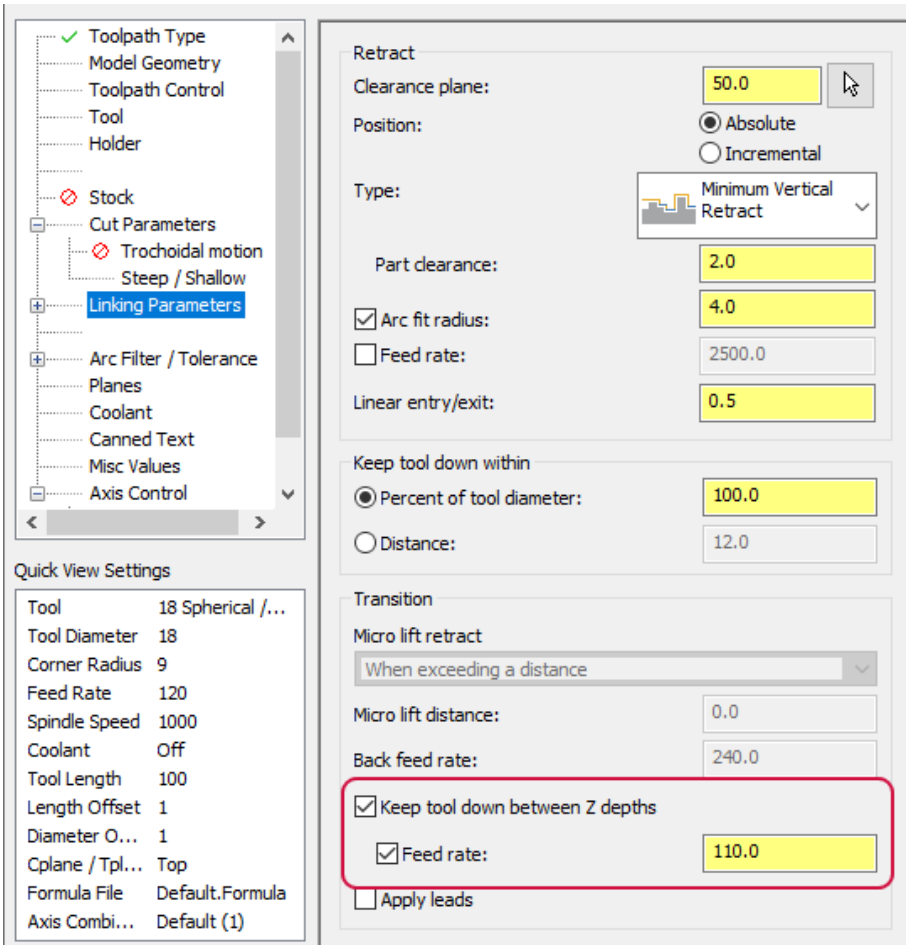
More Powerful and Flexible 3D Roughing Linking

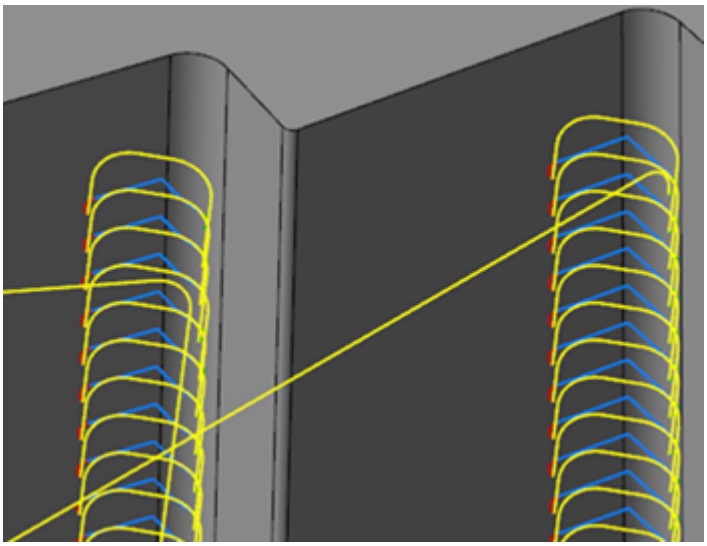
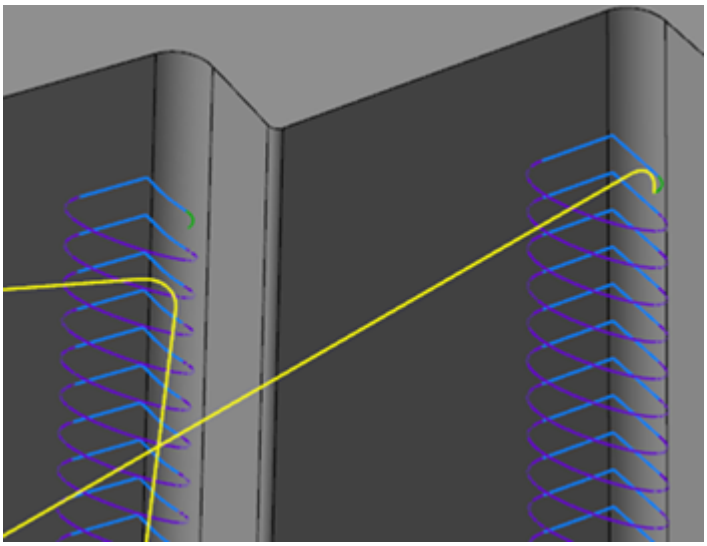
Mastercam 2022 introduced the new **Linking Parameters** page for finishing 3D high speed toolpaths, which includes new and improved functionality. Mastercam 2023 has expanded this effort to the Dynamic OptiRough, Area Roughing, and Horizontal Area toolpaths.



Preventing the Tool from Retracting Between Passes

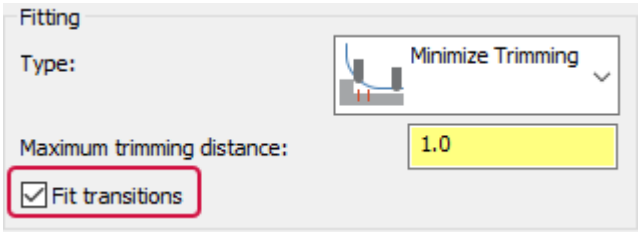
You now control the moves between Z depths using the new **Keep tool down between Z depths** option. In previous releases, Mastercam always retracted to move from Z depth to Z depth on open passes. You can now create smooth linking moves between Z depths, with an optional feed rate control. You can also specify a separate **Feed rate** for these moves. Additionally, the **Keep tool down option** has been moved from the **Cut Parameters** page.

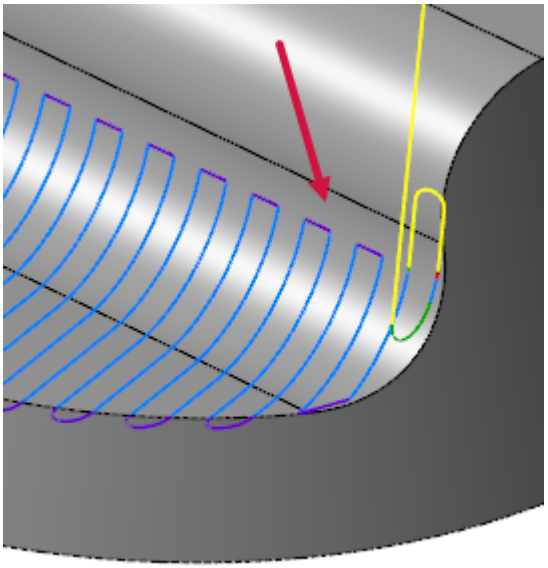
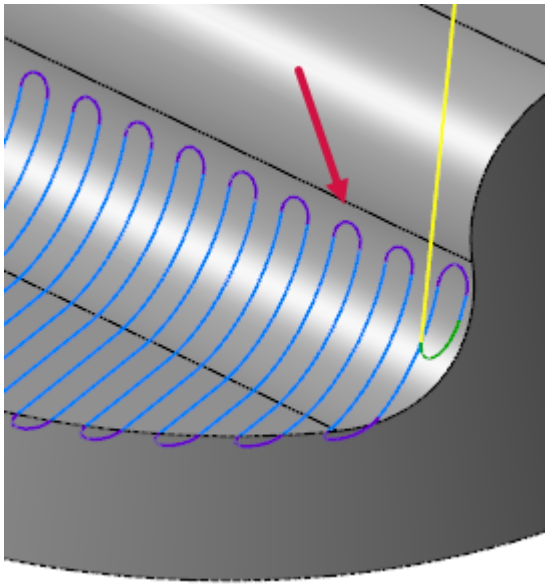


Keep tool down off	Keep tool down on
	

Trimming Passes to Fit Transition Moves

Dynamic OptiRough, Area Roughing, and Horizontal Area now have the ability to trim passes to safely fit the transition moves, using the **Fit transitions** option. This option was previously only available to 3D high speed finishing toolpaths.



Fit transitions off	Fit transitions on
	

Rest Roughing Against Stock Model Improvements

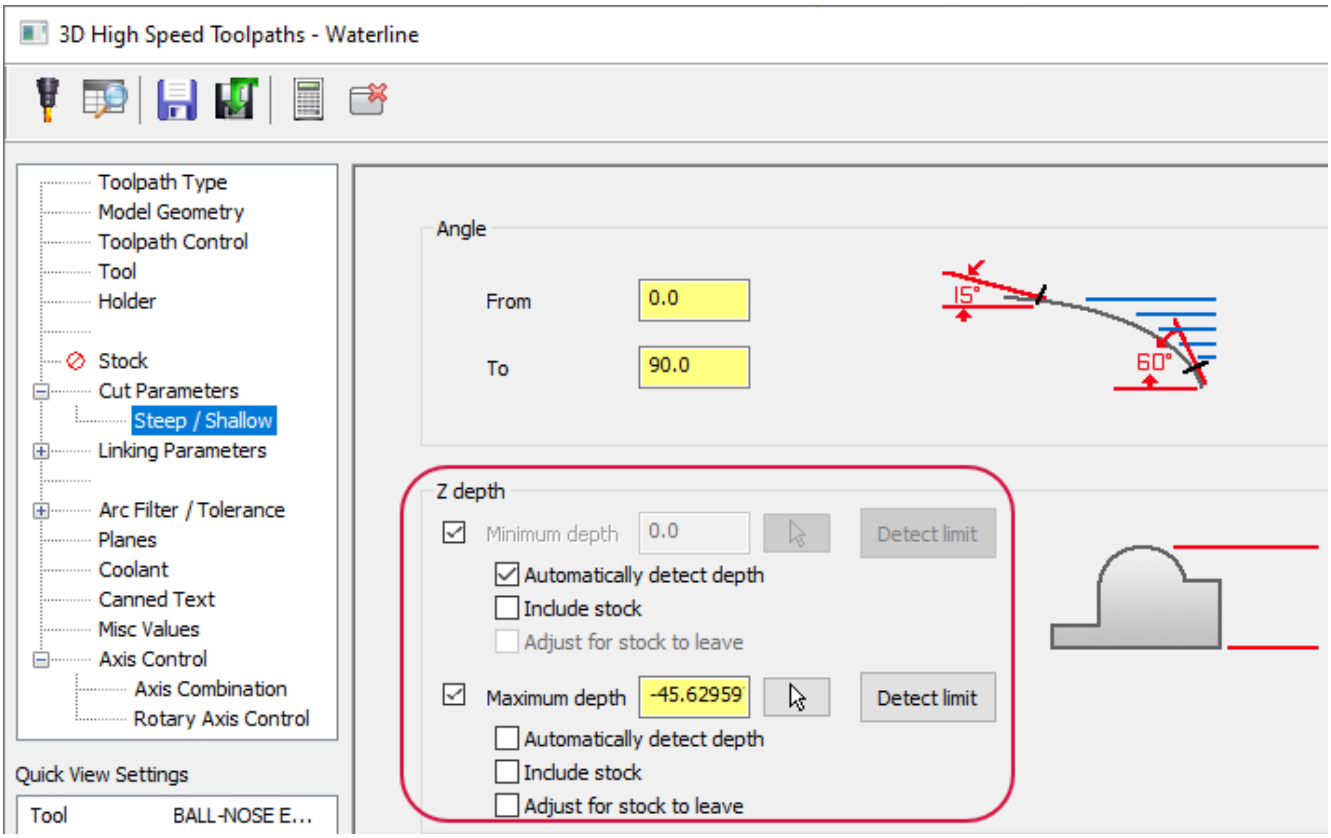
When rest machining against one other operation that references a stock model for Dynamic OptiRough and Area Roughing toolpaths, the following improvements have been made:

- **Machining geometry** is no longer required. Mastercam now ensures that the stock model can be reached from all sides in the machining XY plane when no machining geometry has been defined.
- A minimum Z depth set above the stock model is no longer required. Mastercam now ensures all stock is machined in the Z direction when no Z depth limits are defined.
- Mastercam takes a stepdown at the defined maximum Z depth when set below the stock model to aid in breakthrough.
- If collision checking is selected on the **Holder** page, Mastercam is now aware of all stock material that resides outside the Z depth limits and/or selected **Containment boundaries**.

Set Z depth options on the **Steep/Shallow** page and **Containment boundaries** on the **Toolpath Control** page.

Automatically Detecting and Including Stock for Z Depths

The **Steep/Shallow** page for 3D high speed toolpaths now includes two new options, **Automatically detect depth** and **Include stock**. Selecting **Automatically detect depth** allows you to automatically detect the Z limits when the toolpath is processed. This means you do not need to select **Detect limit** each time you modify the toolpath.



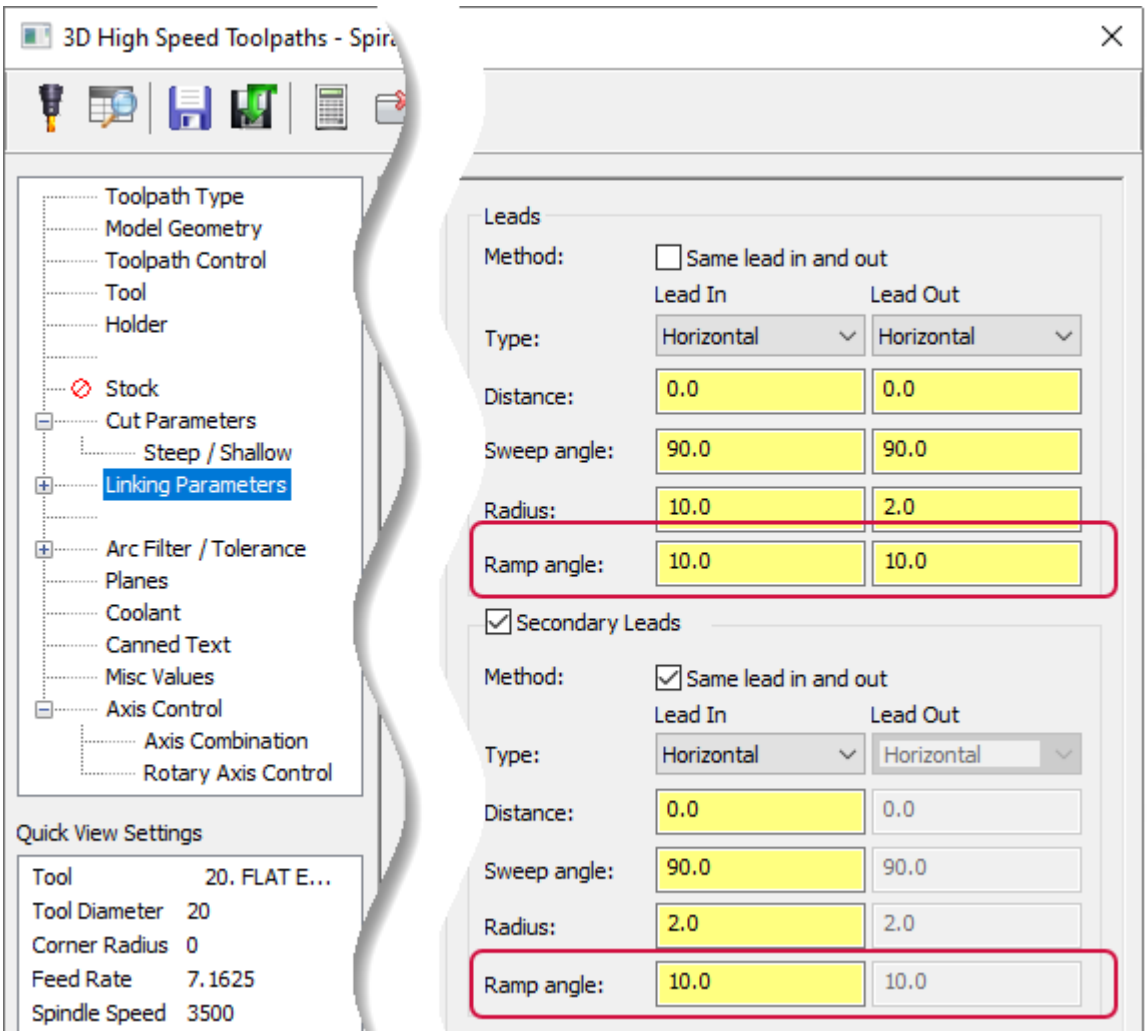
NOTE
Depending on the 3D toolpath type, some of these new options may be unavailable.

If you select **Include stock**, Mastercam includes the stock model defined on the **Stock** page in addition to the geometry defined on the **Model Geometry** page when manually or automatically detecting the minimum and maximum Z limits.

You can also manually select the **Minimum depth** and **Maximum depth**. In previous releases, you had to manually enter a value or use data entry shortcuts. Now, you can use the **Select** buttons to return to the graphics window to select a depth. These values determine the highest point and the lowest (deepest) point on your part that you want to machine.

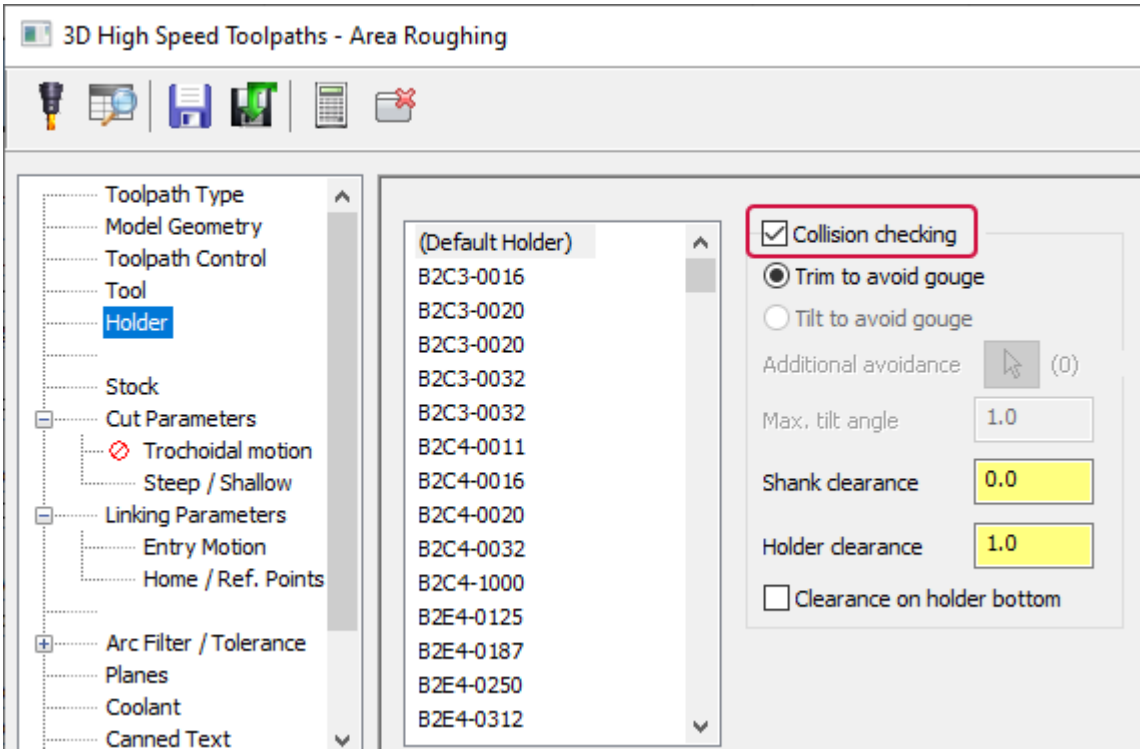
Setting the Entry and Exit Move Helix Arc Height

The **Linking Parameters** page of 3D high speed toolpaths now includes the **Ramp angle** parameter. **Ramp angle** was previously only available to Dynamic OptiRough, Area Roughing, and Horizontal Area toolpaths. This option is now included with all 3D high speed toolpaths. When you enter a value other than zero for the **Horizontal arc entry** parameter, the entry move becomes a small helix when moving into a new region. **Ramp angle** sets the height of this helix.



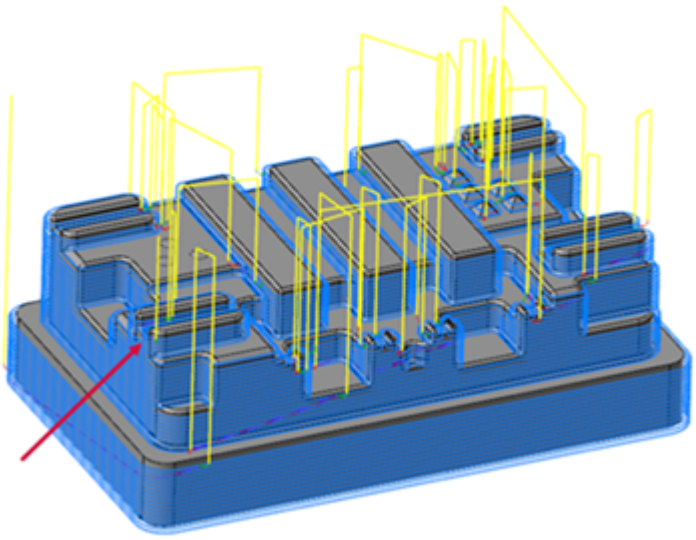
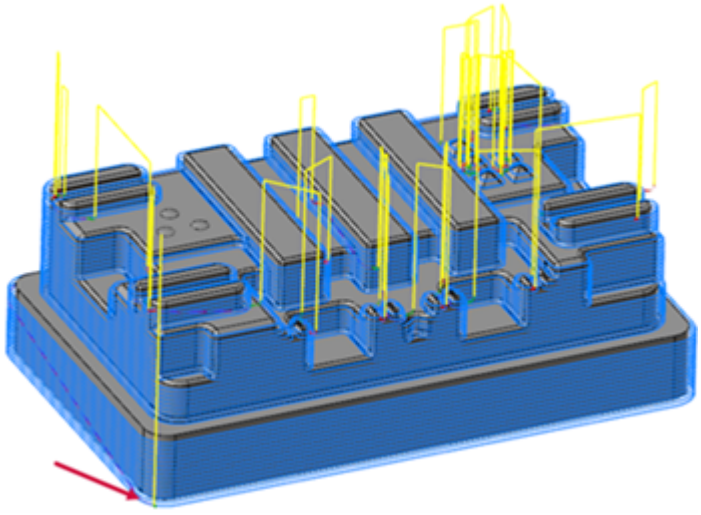
Adding Stock Awareness to Holder Checking

Dynamic OptiRough, Area Roughing, and Horizontal Area toolpaths with **Rest material** enabled on the **Stock** page are now aware of stock outside of the containment boundary when **Collision checking** is enabled on the **Holder** page. Previous versions of Mastercam did not check against any portion of the stock model that existed outside of the containment boundary.



Machining from Bottom to Top

You can now machine bottom to top with a 3D Waterline toolpath with a separate **Cut order**. In previous releases, **Bottom to top** was part of the **Cut order** parameter. In Mastercam 2023, **Bottom to top** is now a selectable option that respects both **By depth** and **Optimize**. The images below show a Waterline toolpath with **Cut order** set to **Optimize**. Notice the starting locations for both toolpaths.

Bottom to top off	Bottom to top on
	

Machining at Maximum Depth

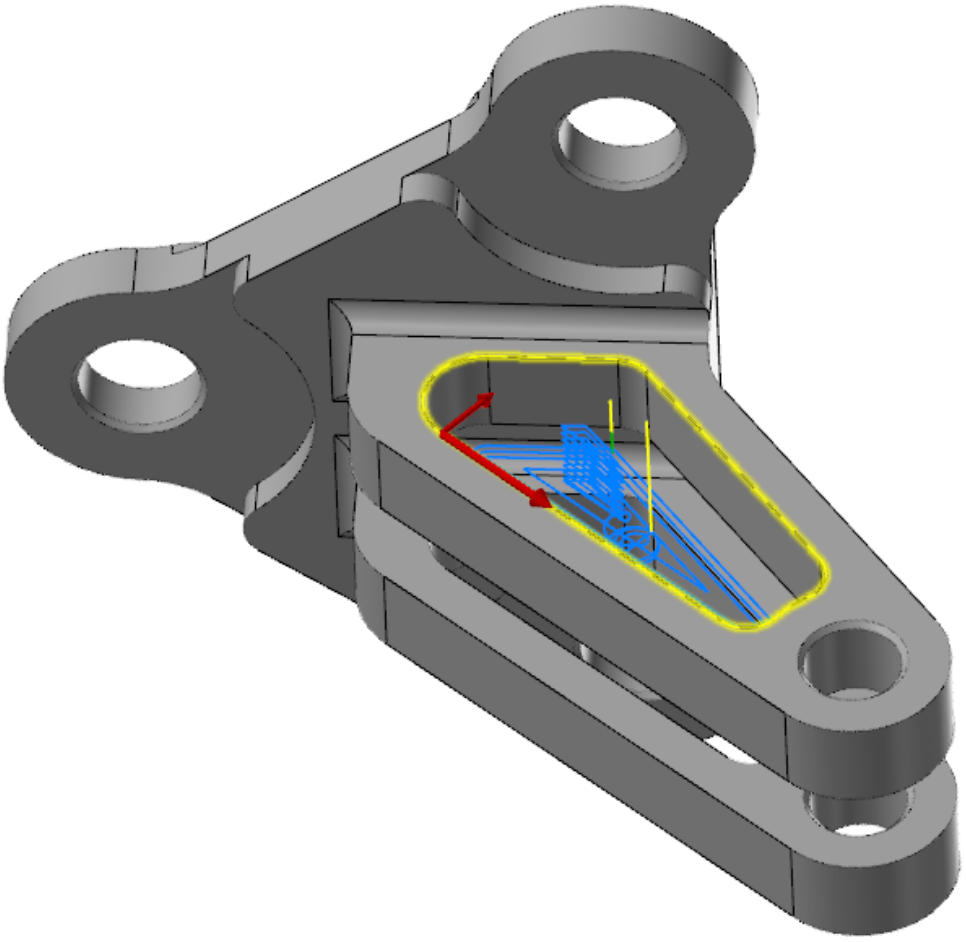
The 3D high speed Dynamic OptiRough toolpath now machines a pass at the defined **Maximum depth**, set on the **Steep/Shallow** page when **Rest material** is selected on the **Stock** page. In previous releases, this pass was not machined when rest roughing against stock models if the **Maximum depth** was defined below the stock model shape.

2D Toolpath Enhancements

Listed below are enhancements to 2D milling toolpaths.

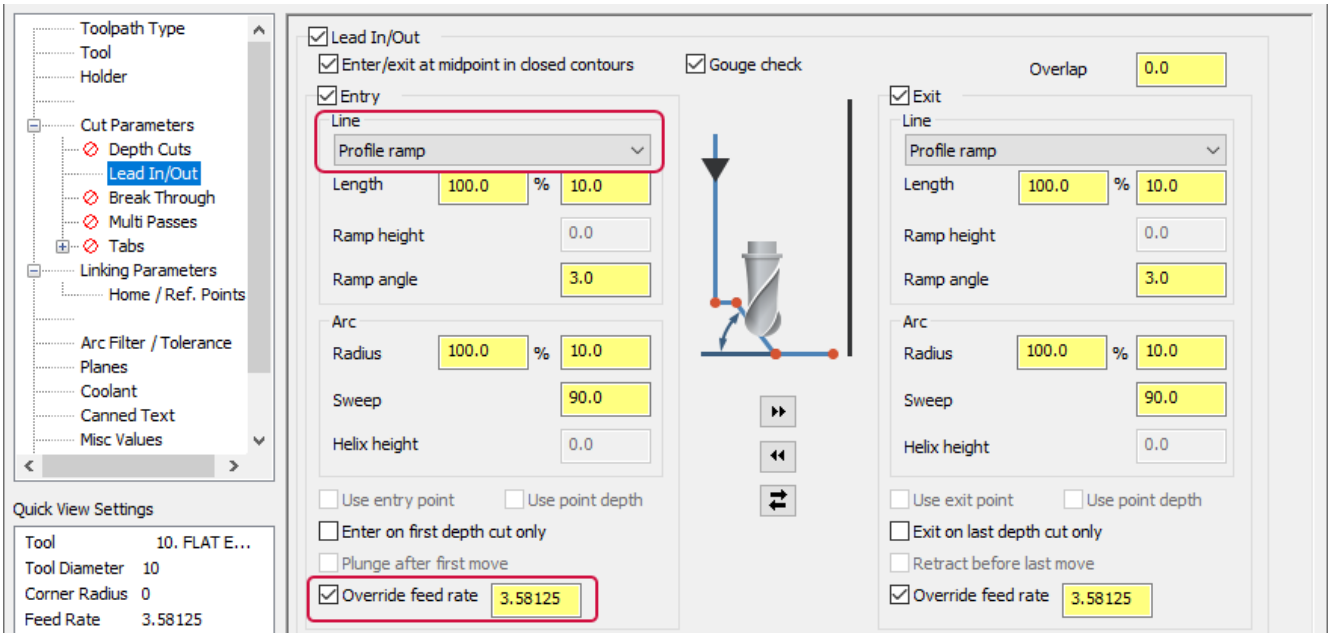
Removing Restrictions with Slot Mill Geometry

The Slot Mill toolpath no longer requires two parallel walls when selecting the machining geometry. You can select any closed chain. In previous releases, Mastercam would display an error stating that the boundary must be closed and include two parallel, straight sides.



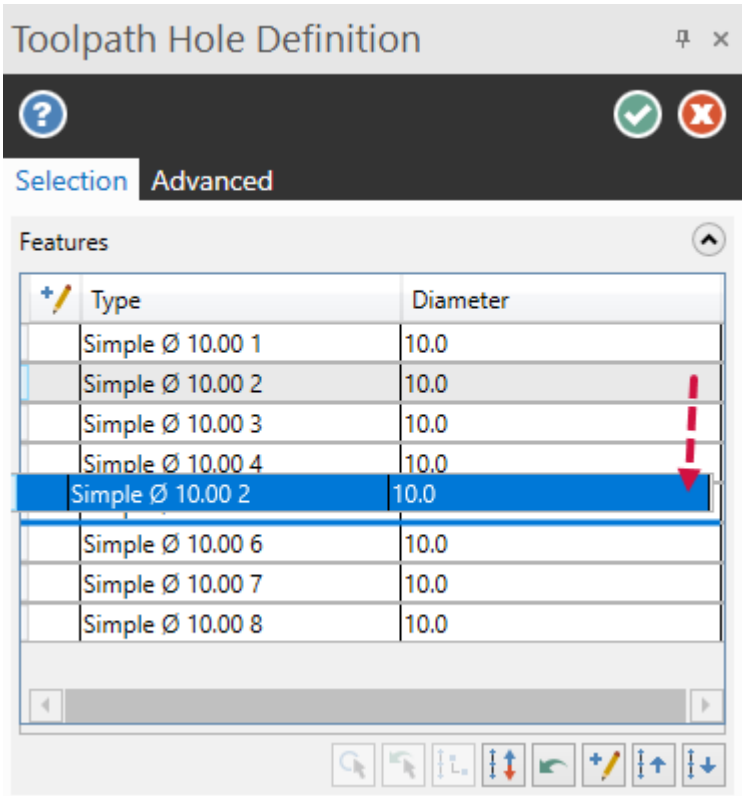
Overriding the Feed Rate for Profile Ramp Entry and Exit

2D Contour now supports the **Override feed rate** option, which allows you to specify a feed rate for the entry or exit move, when **Entry** or **Exit** is set to **Profile ramp**.



Dragging and Dropping for the Hole Making Toolpaths

When you drag-and-drop items in the **Toolpath Hole Definition** panel, Mastercam now provides visual cues for the drop location. Also, the drag-and-drop icon now indicates whether you are dragging a single item or multiple items.

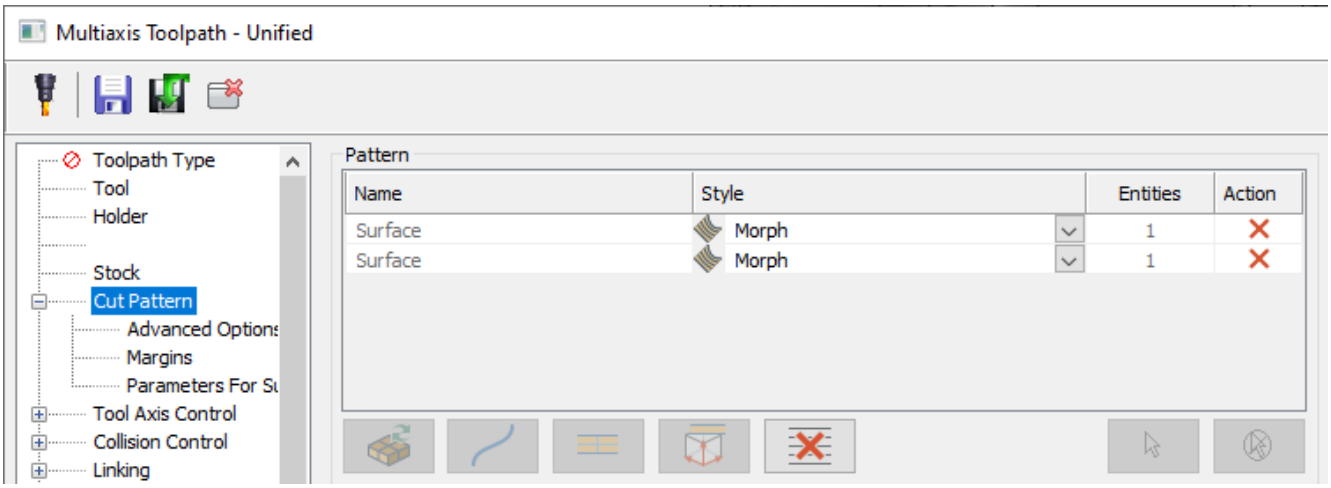


Multiaxis Toolpath Enhancements

Listed below are enhancements to Multiaxis toolpaths.

Consolidating Multiaxis Toolpaths

Morph, Parallel, Along Curve, and Project Curve are no longer individual toolpaths in Mastercam. Instead, the Unified toolpath allows access to these cut patterns when you add the appropriate curves. For example, to create a toolpath that morphs between two surfaces, select **Unified** from the **Multiaxis** toolpaths, and then set the **Cut Pattern** to two surfaces with the **Morph** style, as shown in the following image:



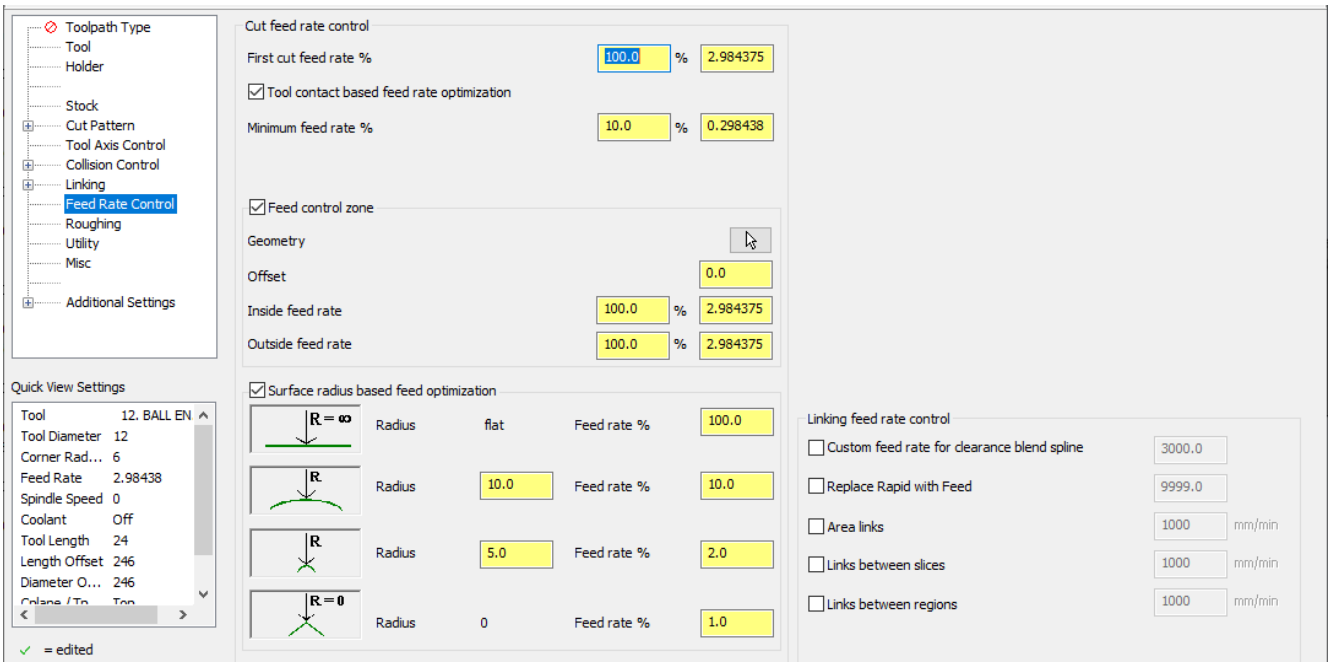
Toolpaths in files from previous releases are automatically converted to a Unified toolpath as follows:

- **Morph**—Becomes the Unified toolpath with a **Cut Pattern** of two curves or surfaces with the **Morph** style.
- **Parallel**—(set to **Curves** or **Surface**): Becomes the Unified toolpath with a **Cut Pattern** of one curve or surface with the **Parallel** style.
- **Parallel**—(set to **Angle**): Becomes the Unified toolpath with a **Cut Pattern** of **Plane** with the style set to the appropriate plane.
- **Along Curve**—Becomes the Unified toolpath with a **Cut Pattern** of **Curve** with the **Perpendicular** style.
- **Project Curve**—Becomes the Unified toolpath with a **Cut Pattern** of **Curve** with the **Project** style.

New Feed Rate Control Page

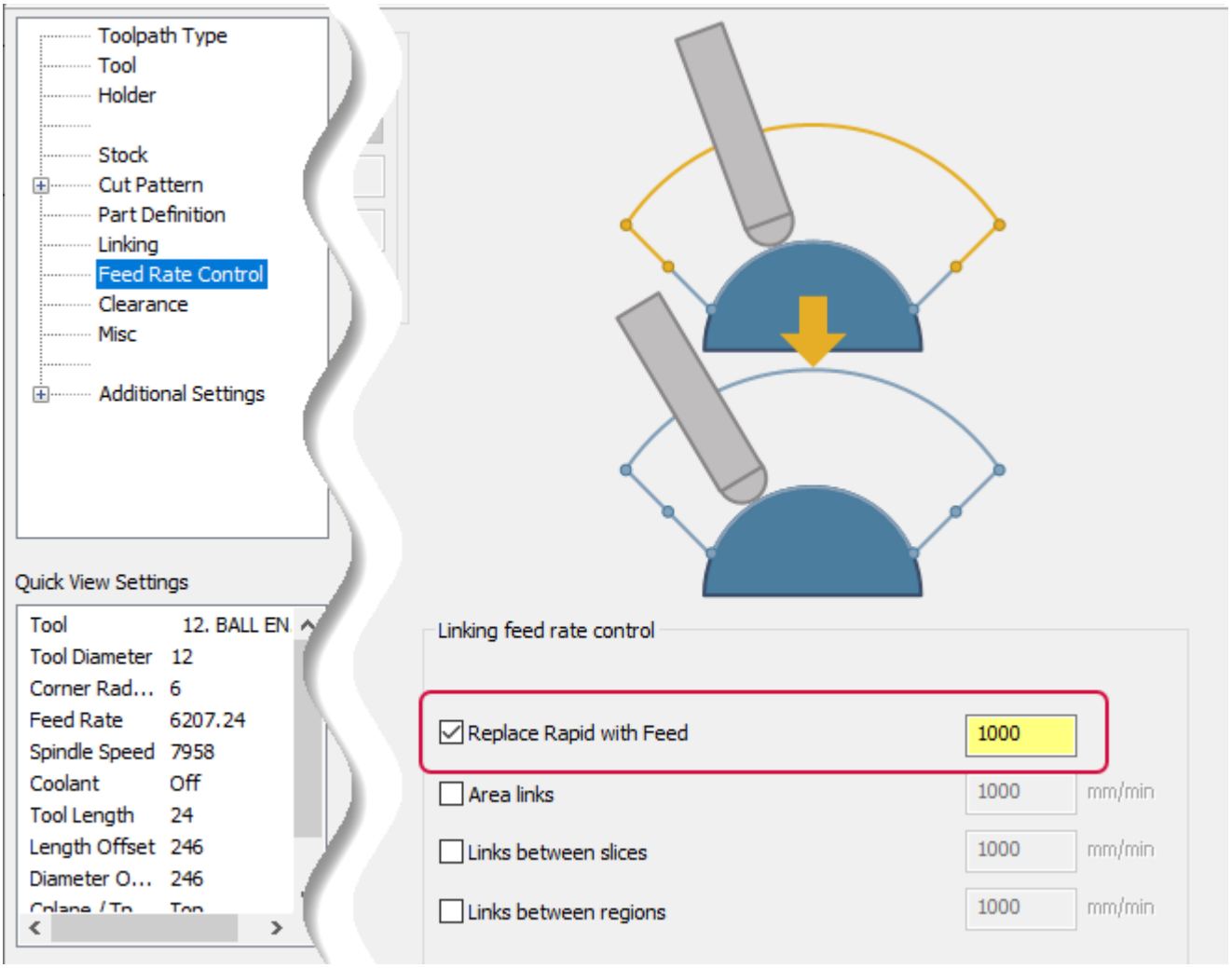
Mastercam 2023 features a new **Feed Rate Control** page for Mulitaxis toolpaths. This page consolidates the feed rate control options found in the **Linking**, **Miscellaneous**, and **Utilities** pages, making the options easier to find and simplifying your workflow. You can find this page in a toolpath's parameters dialog box, as shown in the following picture.

NOTE
The contents of this page may change, based on the selected toolpath type.



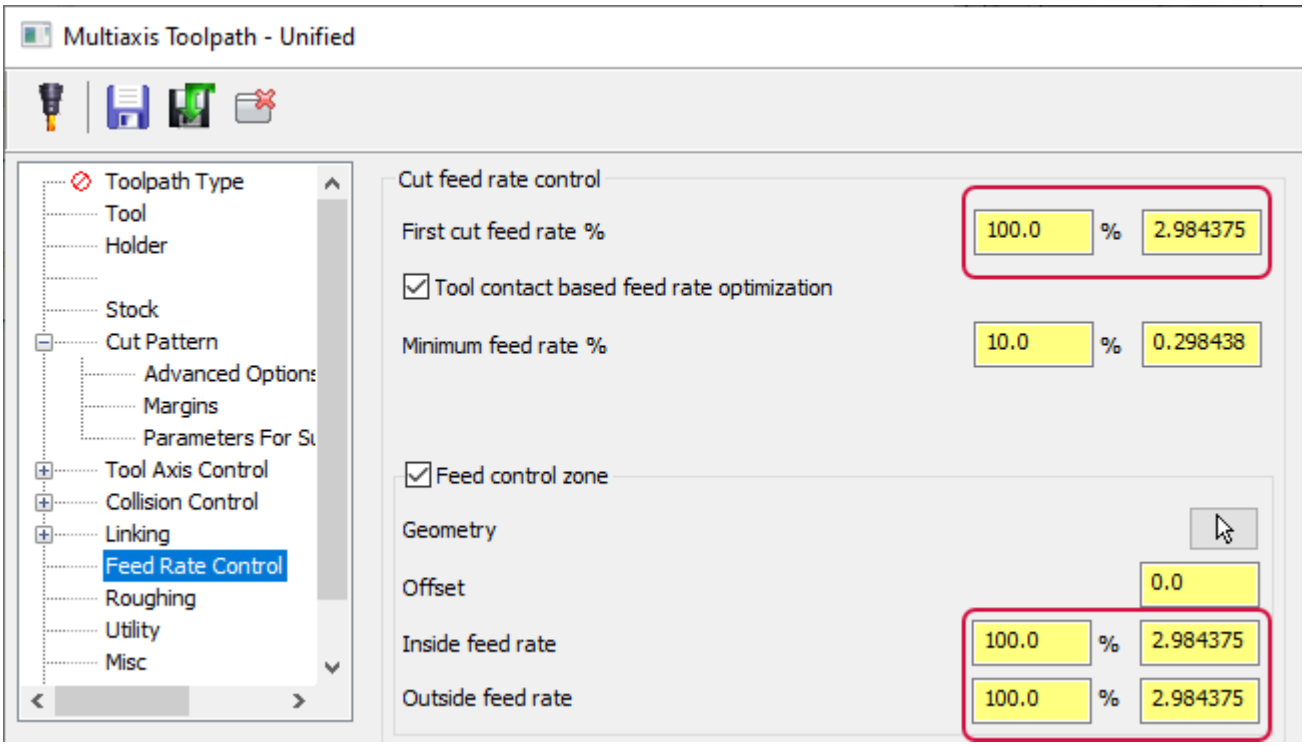
Replacing Rapid Motion with Feed Rate

Rotary Advanced toolpaths now include a new **Replace rapid with feed** option on the **Feed Rate Control** page, providing better control over rapid conditions by setting a feed rate for clearance area and rapid distance moves.



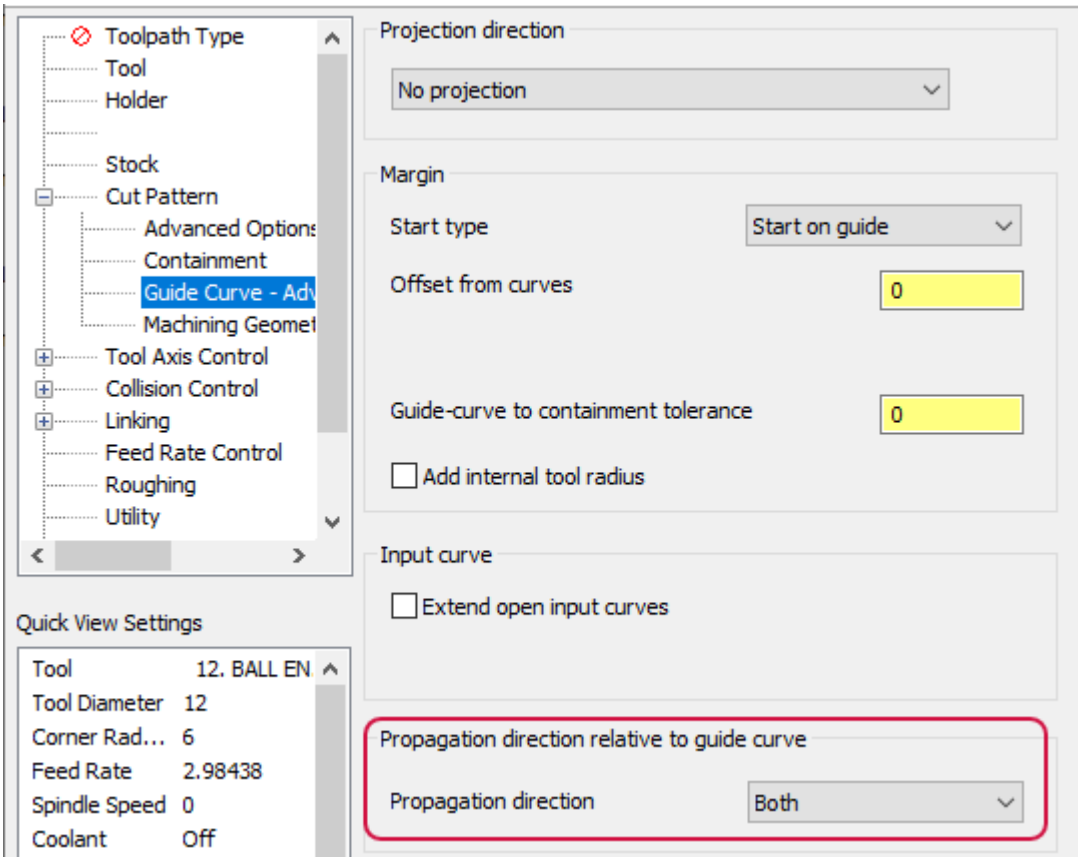
Setting Feed Rates as a Percentage or Direct Value

You can now set feed rates by either the percentage of the machining feed rate or the feed rate itself, allowing easier control of these values. Previously, you could adjust the feed rates only by percentage. Entering a value in the percentage parameter or the feed rate parameter automatically updates the other.



Controlling the Guide Curve Across Machining Geometry

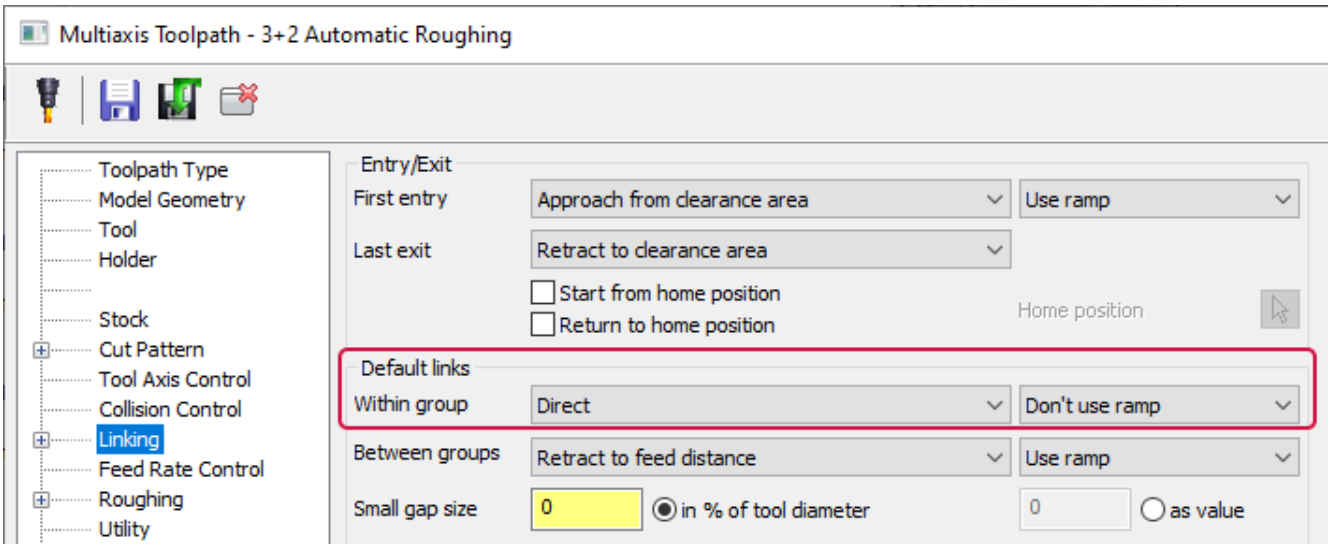
The new **Propagation direction** parameter on the **Guide Curve - Advanced Parameters** page for Unified toolpaths allows you to control which way the guide curve propagates across machining geometry.



The **Guide Curve - Advanced Parameters** page is available when the **Pattern** is set to **Automatic** or to **Curve, Guide**.

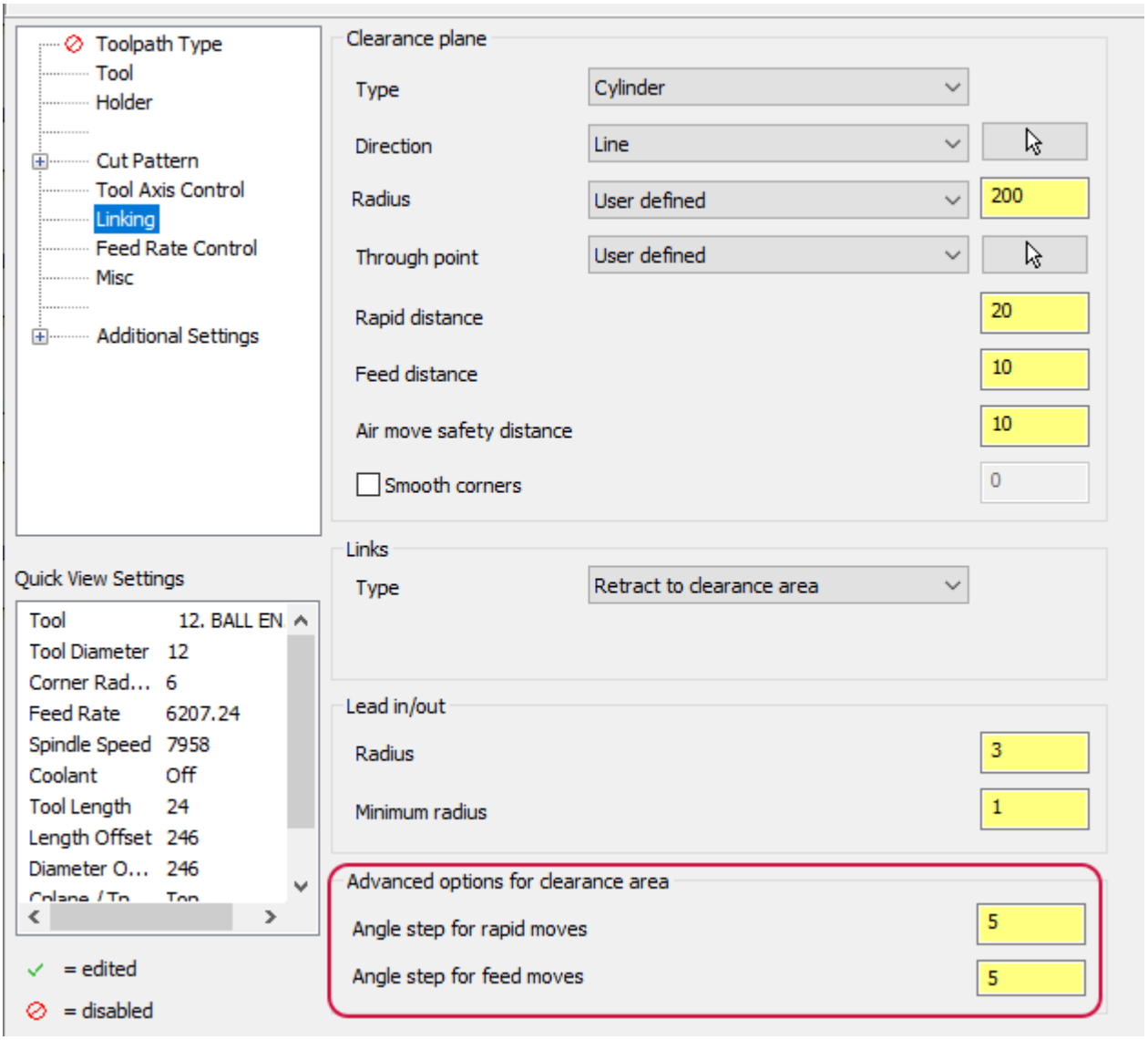
Specifying Connection Moves Between Cuts

The **Linking Parameters** page for Pocketing and 3+2 Automatic Roughing toolpaths now includes the **Within group** option. Use this option to specify connection moves between the offset cuts in a single group, providing easier manual control over some retract behaviors.



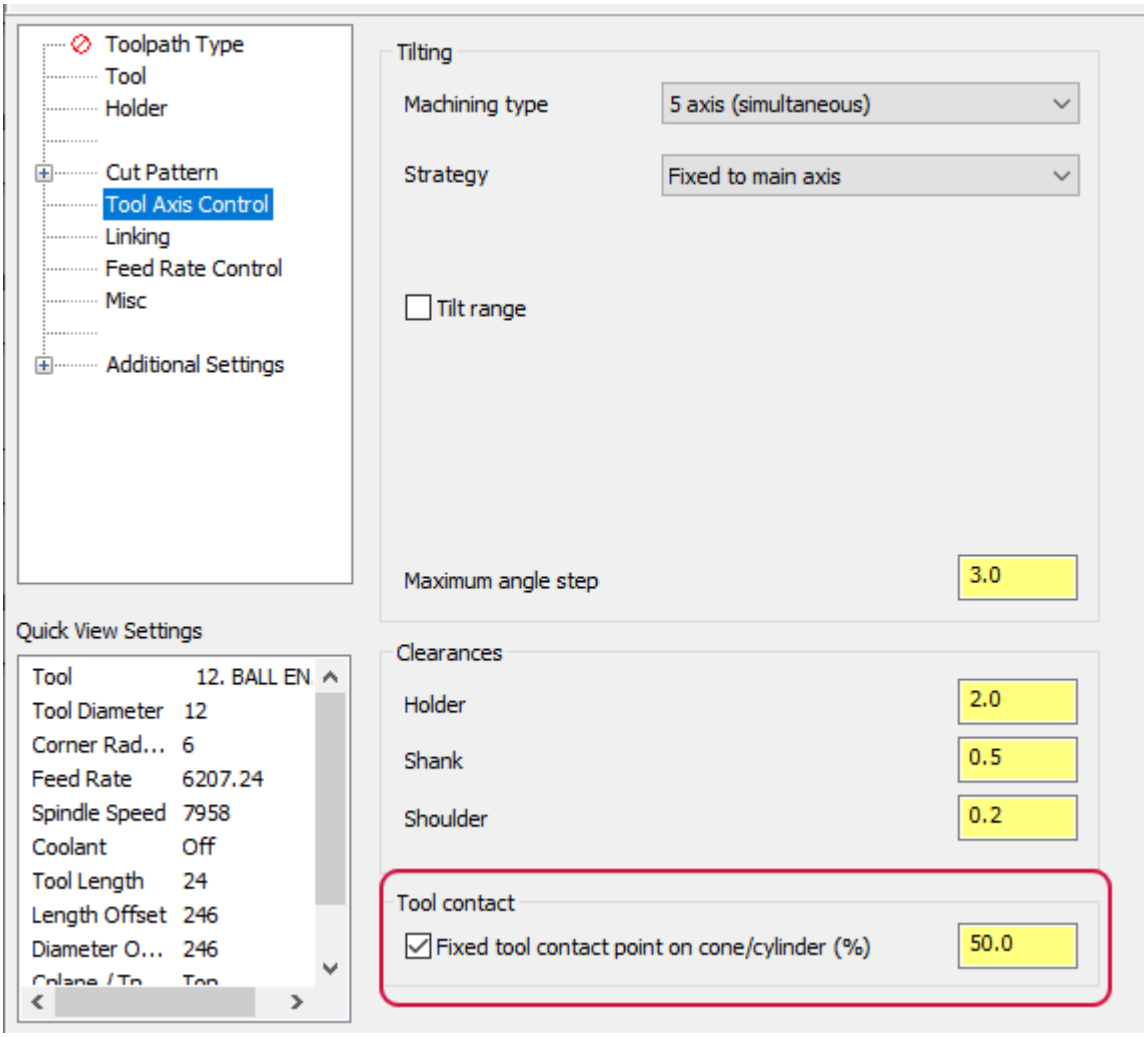
Controlling Rapid and Linking Moves

The new **Angle step** options for Pocketing and Deburr toolpaths allow you to control the angle step of rapid and linking moves. In Pocketing, you can find these options on the **Linking** page when you select **Interpolation tilt angle**. In Deburr, select the **Tool Axis Control** page and set **Machining type** to **5 axis (simultaneous)**. Then go to the **Linking** page and set **Type** to **Cylinder** or **Sphere**.



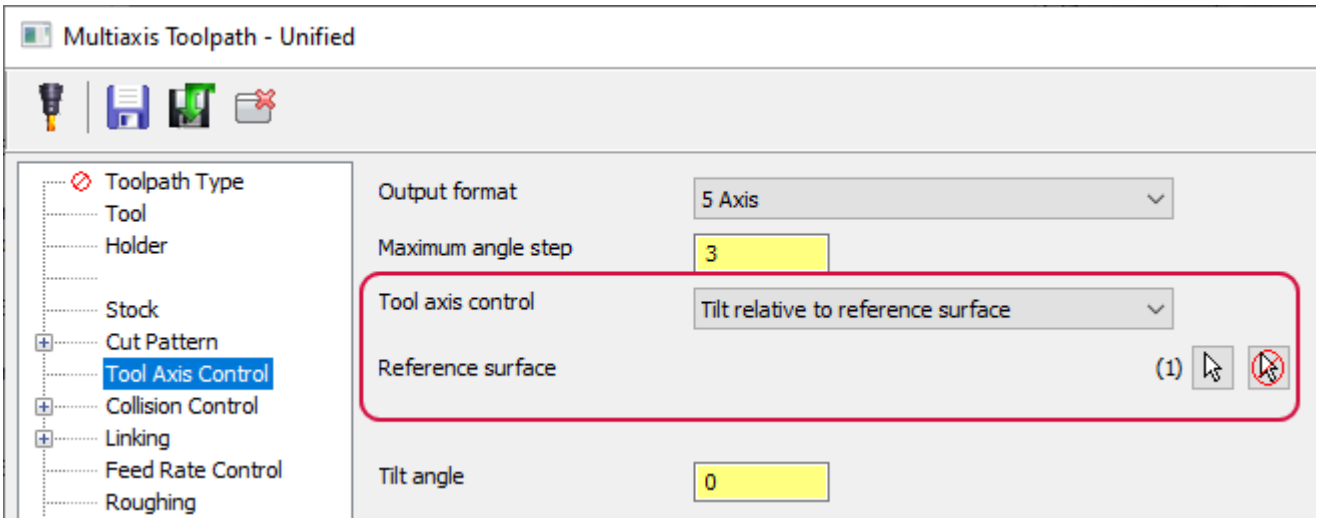
Controlling the Tool Contact Point

The Deburr toolpath now includes a new **Fixed tool contact point on cone/cylinder** option, which controls the tool's contact point during machining. This new option is on the **Tool Axis Control** page. Deburr attempts to use the specified contact point, which is a percentage of the flute length of the cylindrical portion of the flute, on any supported ball, taper, or end mill tool.



Driving the Tool Axis Control from Another Surface

The **Tool Axis Control** page for Multiaxis toolpaths now includes the **Tilt relative to reference surface** option in the **Tool axis control** drop-down. This option lets you drive the tool axis control from a another surface, including surfaces that are shaped differently than the machining geometry.



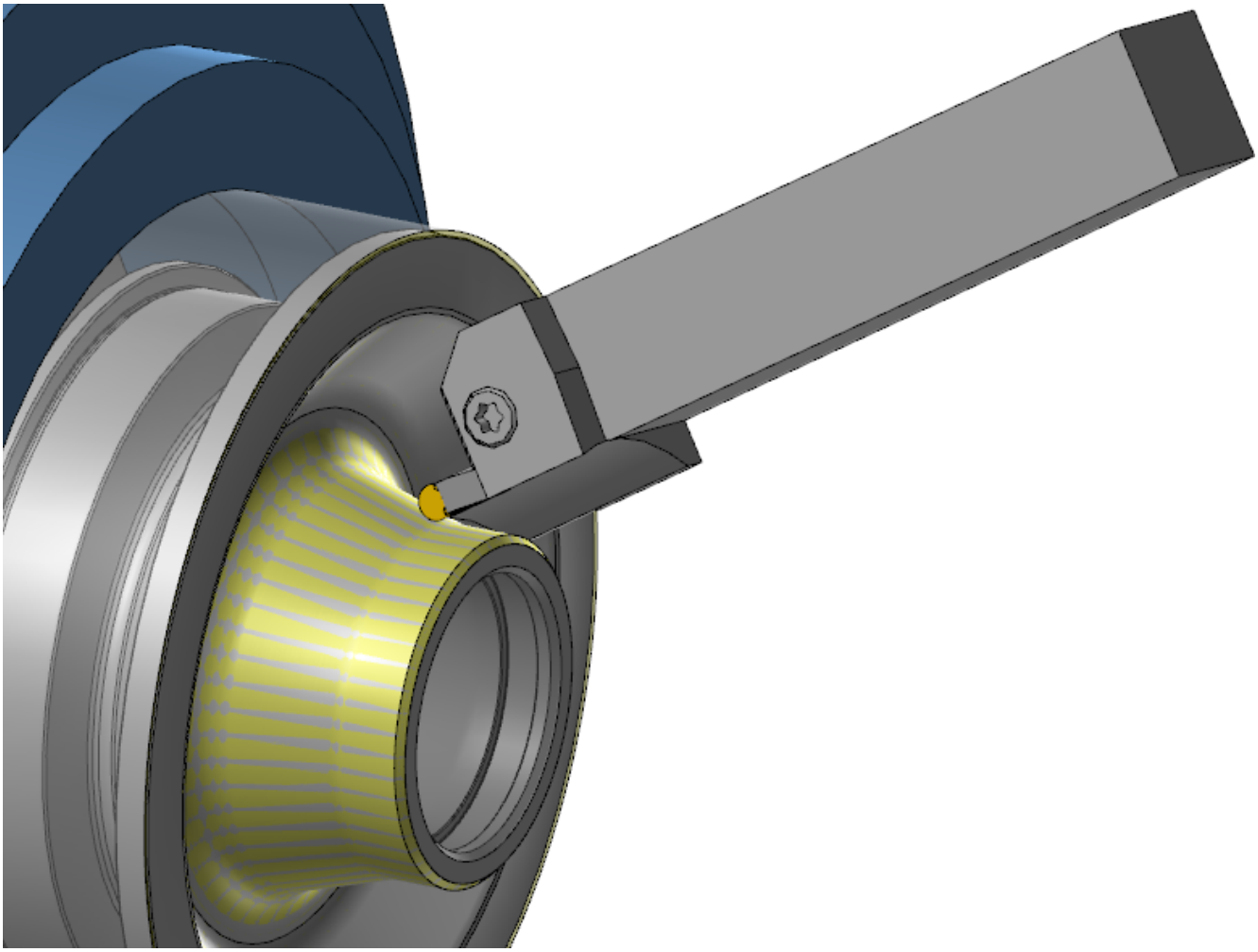
TURNING ENHANCEMENTS

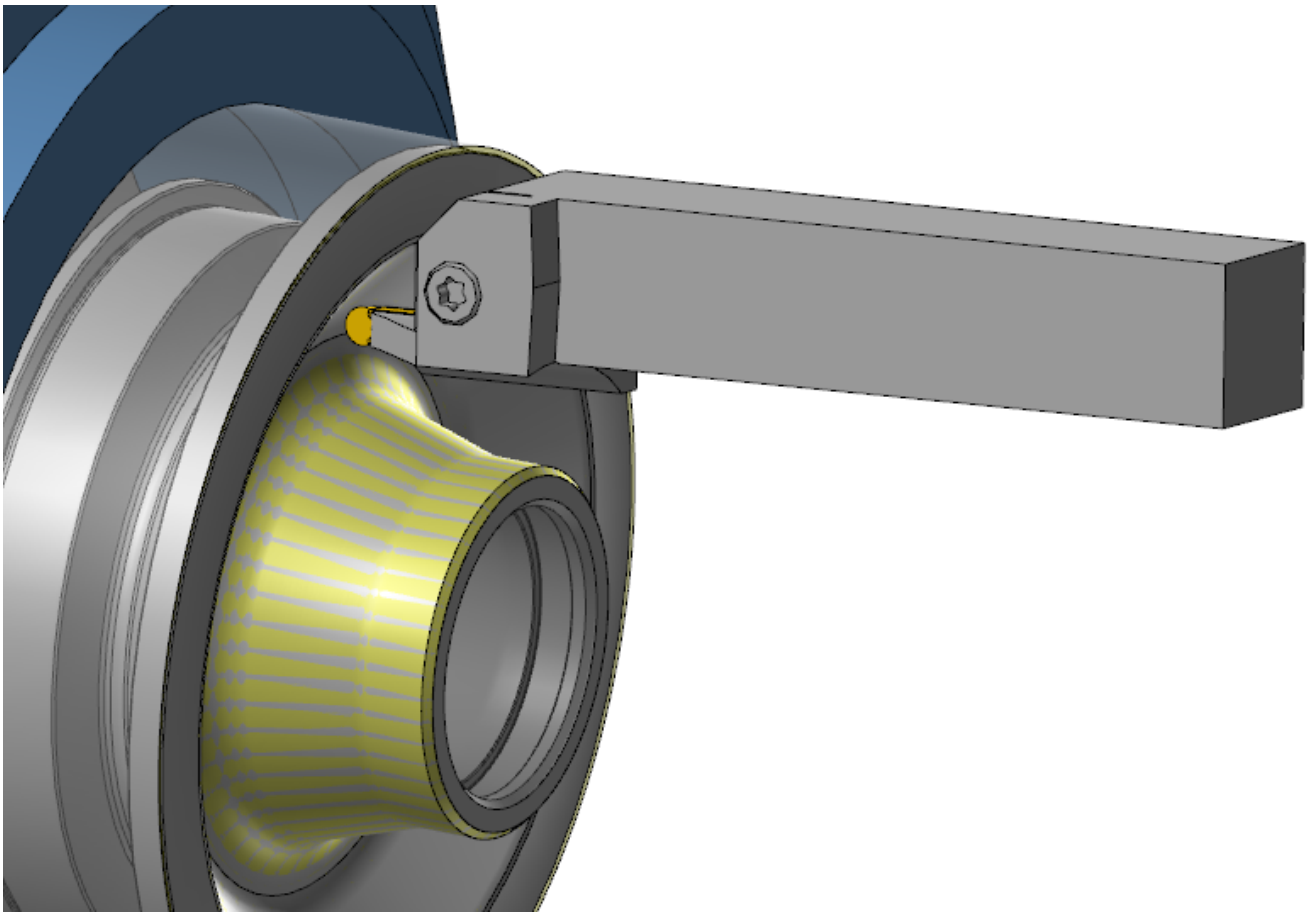
Listed below are major enhancements to the Lathe and Mill-Turn products.

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

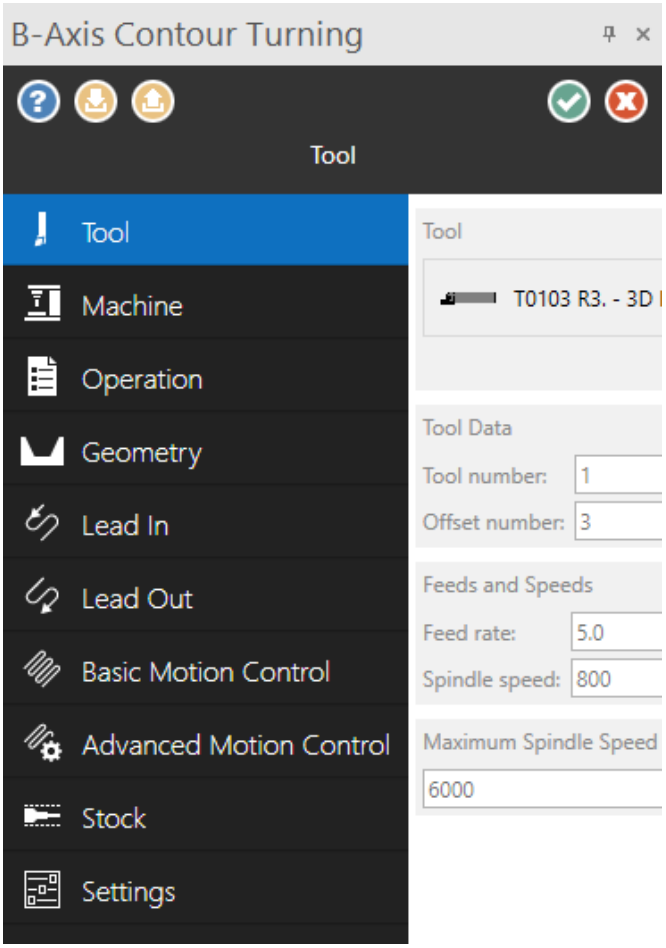
New B-Axis Contour Turning Toolpath

Mastercam 2023 introduces a new toolpath to the Turning suite for the Mill-Turn product. B-Axis Contour Turning is a finishing toolpath that allows for rotation of the B-axis while the tool is cutting. This new toolpath can be found in the **General** gallery on the **Mill-Turn Turning** contextual tab.





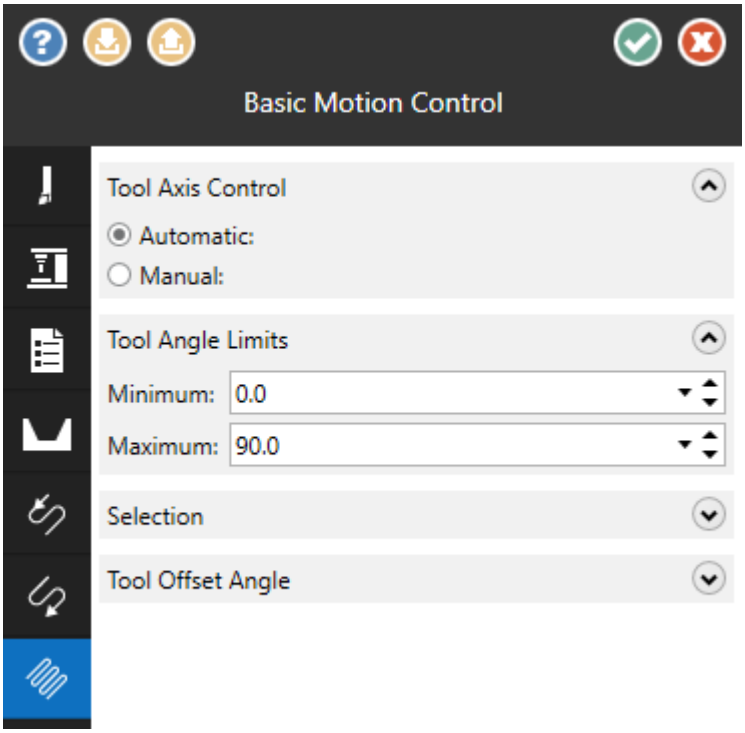
The B-Axis Contour Turning toolpath features a top-down workflow, represented by the icons on the left side of the panel.



Select from two toolpath types on the **Basic Motion Control** page: **Automatic** or **Manual**.

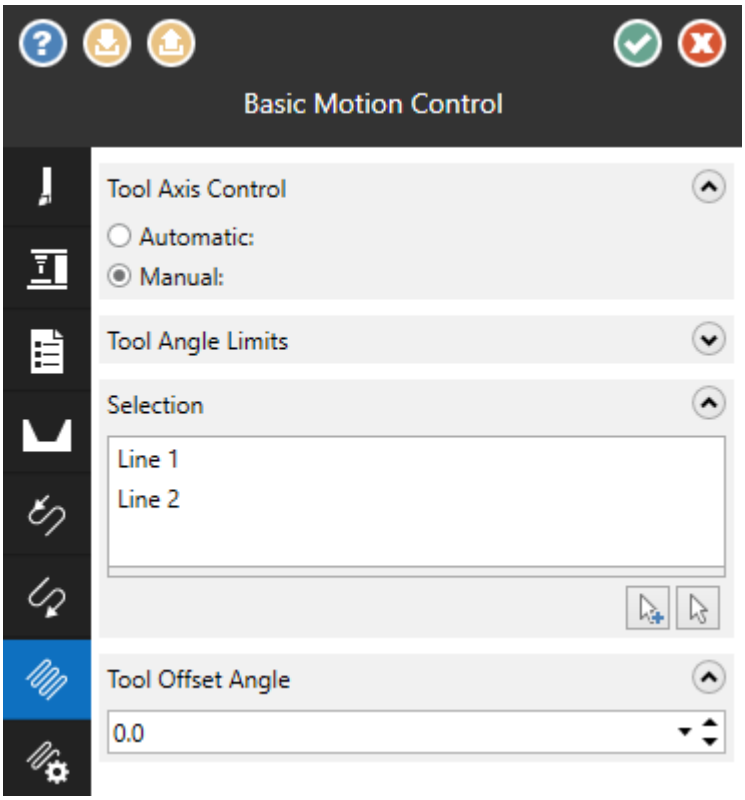
Automatically Create the Toolpath Motion

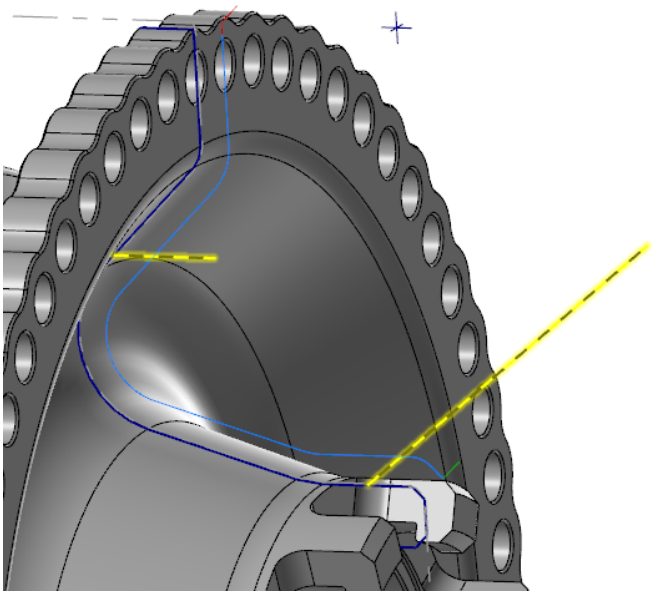
Automatic mode produces safe toolpath motion that keeps the insert in contact with the contour. The **Tool Angle Limits** expander offers quick restriction of the angles that the toolpath is allowed to travel within.

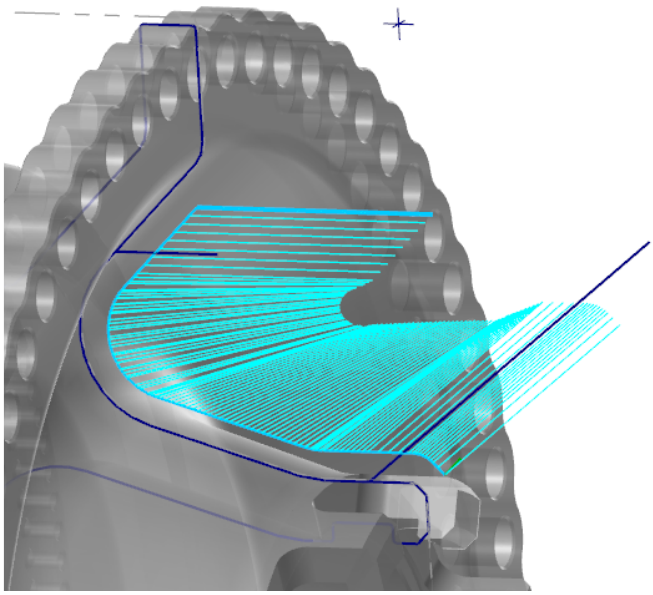


Manually Control the B-Axis Angles

Manual mode offers the user full control over the B-axis angles along the contour. After choosing **Manual**, a selection list box automatically expands to quickly select tool axis control lines. If necessary, you can use the **Tool Offset Angle** parameter to define an offset from the lines.

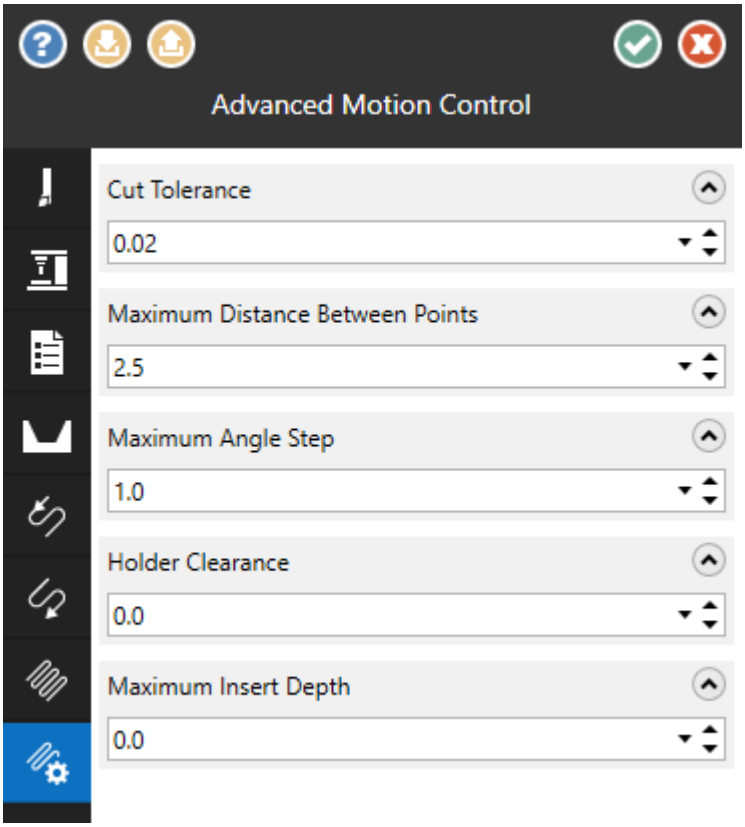






Adjusting the Toolpath Motion for Both Modes

For both **Manual** and **Automatic** mode, use the **Advanced Motion Control** page to control the finer details of toolpath generation.

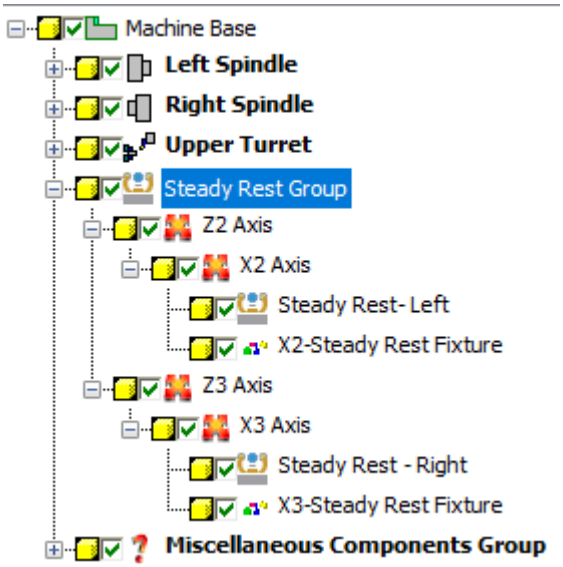


Supporting Multiple Steady Rests

Mastercam 2023 extends the steady rest support in Mill-Turn to include multiple standalone, machine-mounted steady rests. Mastercam's modular machine definition architecture means that you can organize your steady rests in several different configurations.

Using Multiple Standalone Steady Rests

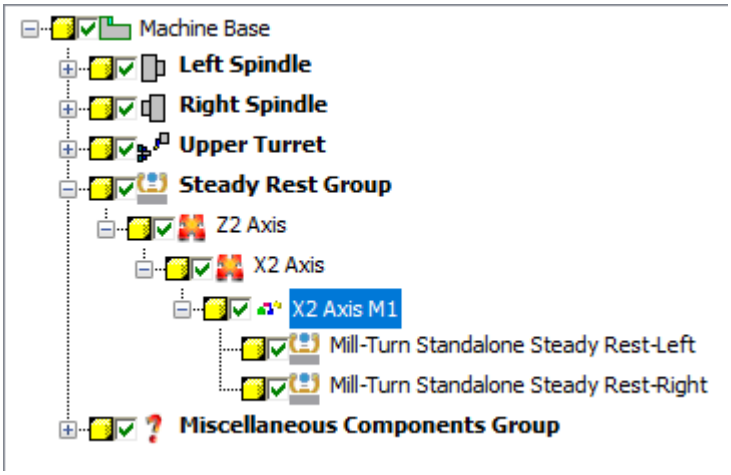
Large CNC mill-turn machines often include multiple steady rests that can be individually positioned. These can support individual work pieces at multiple points. This configuration is represented in Mastercam by two separate steady rest components each attached to their own linear axis components, letting you program and position them independently.

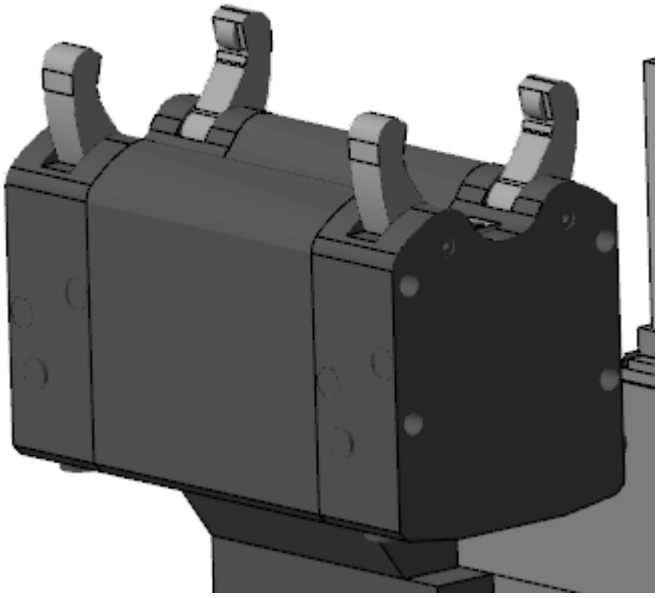


You are not limited to two steady rests; machine developers can create as many as needed to support very long parts. Developers can also select in which stream the commands for each steady rest will be output.

Using Tandem Steady Rests

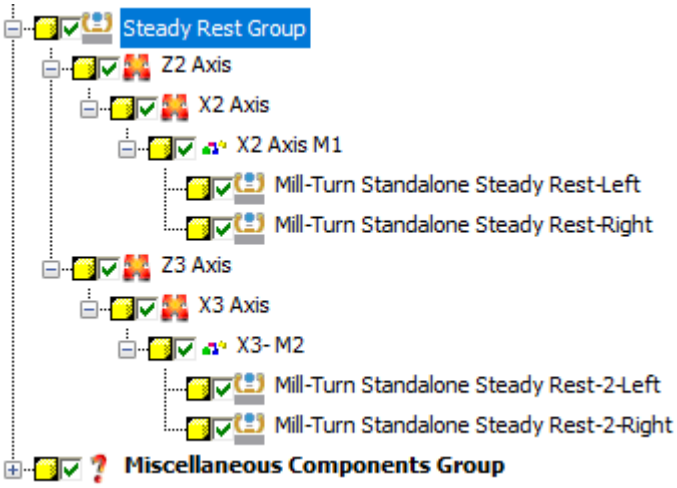
Tandem steady rests are another common configuration; these consist of several steady rests mounted so that they move together. This configuration is represented in Mastercam with multiple steady rest components connected to the same linear axis components. Even though the steady rests move together, while programming your part you can still clamp/unclamp each steady rest independently with individual steady rest operations for each component.

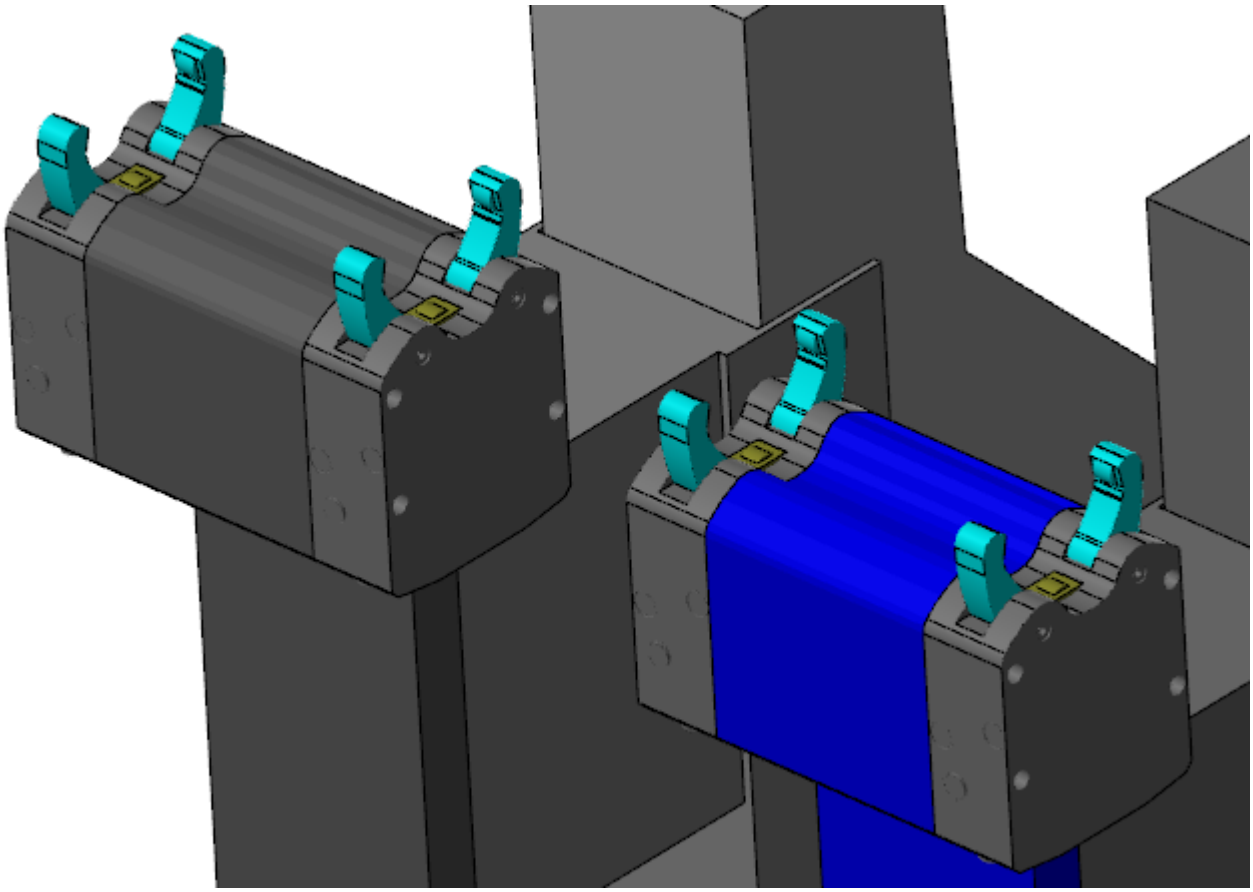




Using Multiple Tandem Steady Rests

Mastercam's modular architecture means that even more complicated applications can be supported. For example, you can include multiple tandem steady rests.



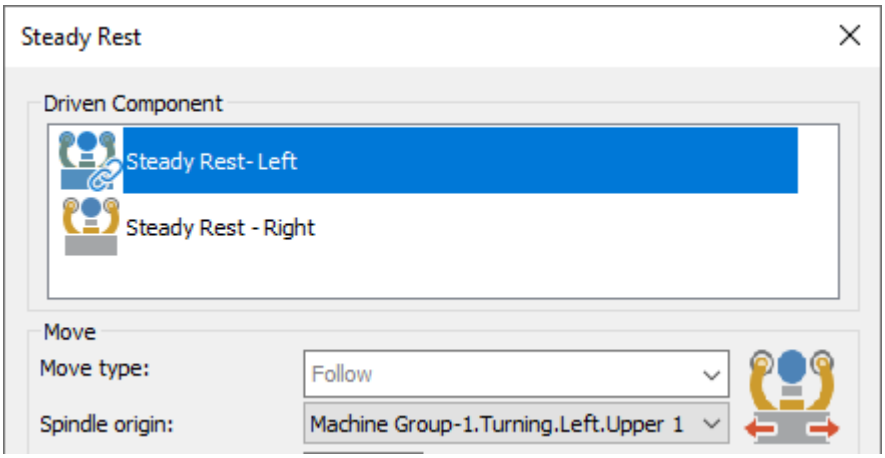


Again, organizing the components in Mastercam's machine definition is straightforward. In this case, create multiple sets of linear axis components and link multiple steady rests to each one. This lets you move each set of tandem steady rests independently of the other.

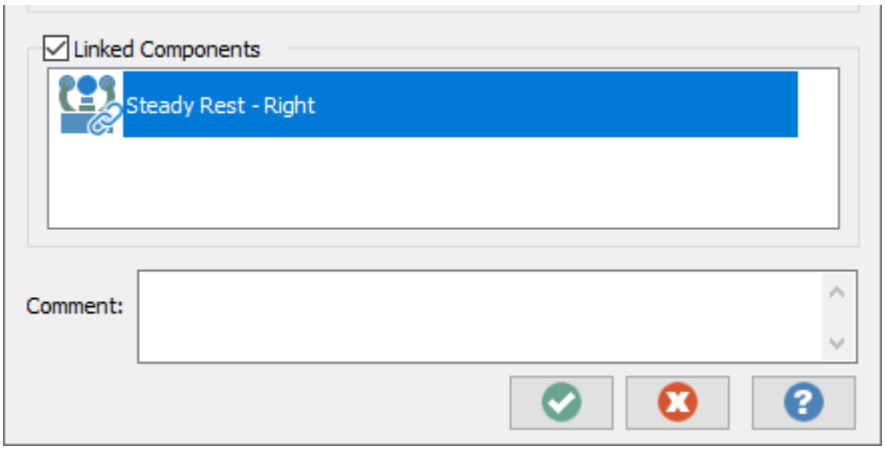
Programming Multiple Steady Rests

Machines with multiple standalone steady rests use the same steady rest operation introduced for single steady rests in Mastercam 2022. However, the interface has been enhanced in a couple of ways.

The **Driven Component** list lets you select which steady rest is being programmed when multiple steady rests are available.



Once you select the primary steady rest for the operation, you can select additional steady rests that are linked to the primary steady rest and move with it. Mastercam automatically detects the available steady rests and displays them here. You can select as many as eight linked steady rests.

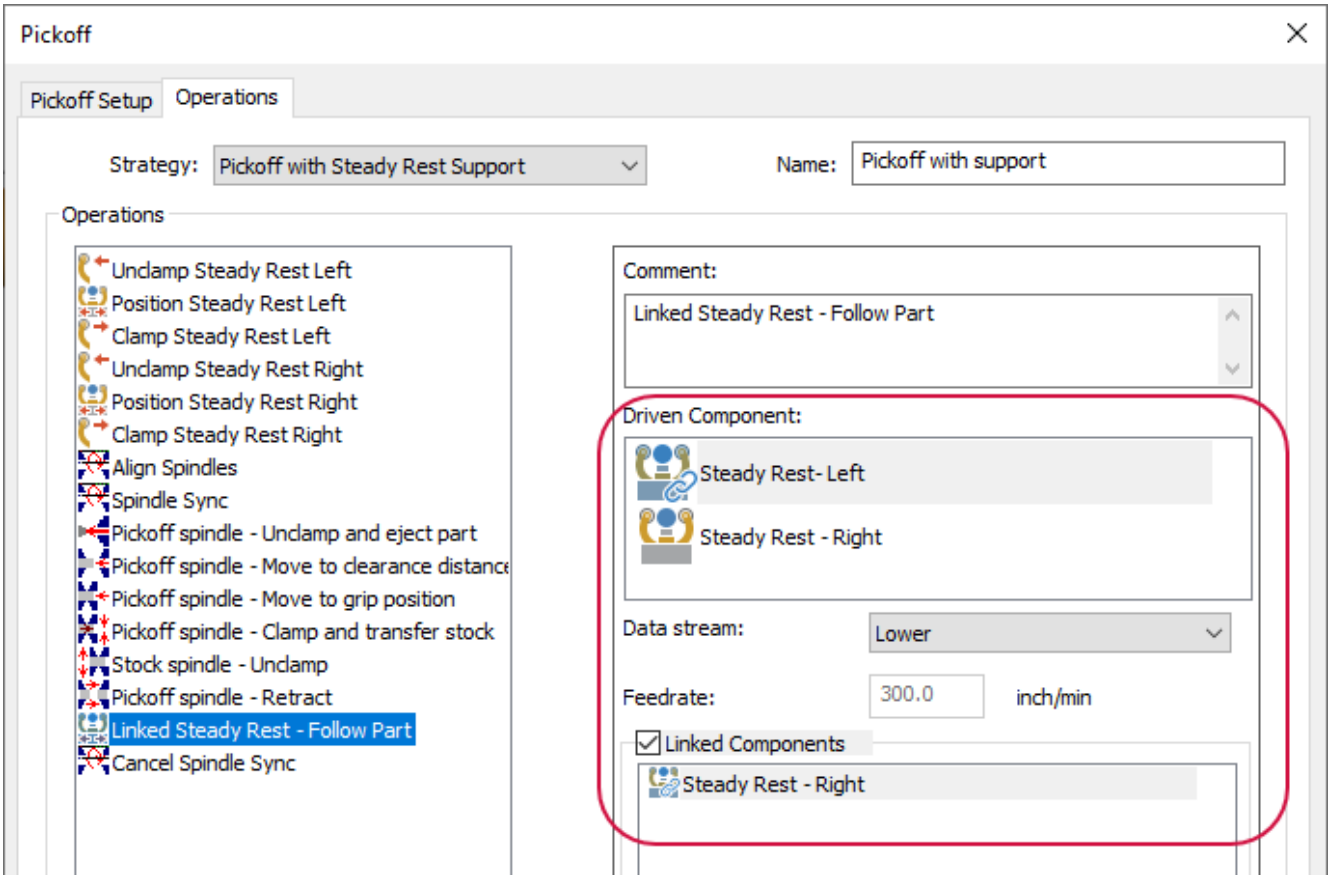


Steady rests that are output in the same stream can be programmed in this way. If the steady rests are in different streams, the operations can be synced in the Sync Manager.

Incorporating Multiple Steady Rests in Operation Strategies

Mastercam 2023 also lets machine developers incorporate multiple steady rests in Mill-Turn part handling strategies. The interface for adding steady rest operations to a part handling strategy has been enhanced so that developers can select which steady rest will be driven by the operation as well as any linked steady rest components. Developers can select steady rests from any stream. The Operation Strategies editor has also been enhanced with a new **Active Spindle** setting; see "Setting a Base Spindle for Steady Rest Operations" on the next page.

End users can access these same settings when they select the strategy while programming their part. The initial settings come from the strategy but the user can edit them if desired:

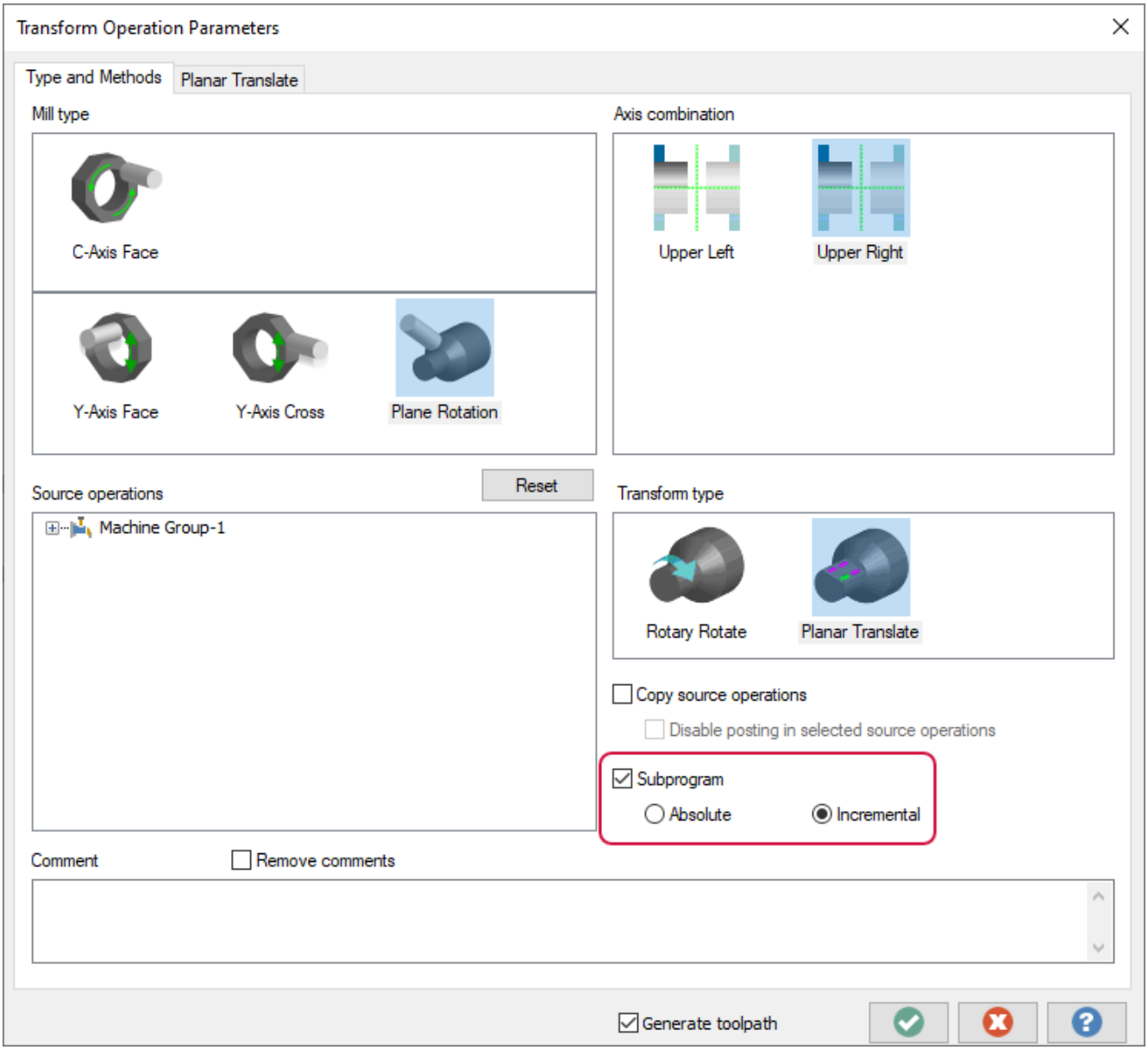


Setting a Base Spindle for Steady Rest Operations

The Operation Strategies editor now lets machine developers select a base spindle for outputting steady rest operations. The steady rest positions will be output relative to the selected spindle origin.

Supporting Transform Subprograms in Mill-Turn

Mastercam 2023 includes support for transform subprograms in Mill-Turn, building on the general subprogram support introduced in Mastercam 2022. The same subprogram options that are available in Mill and Lathe are now available in Mill-Turn when creating your transform operations:

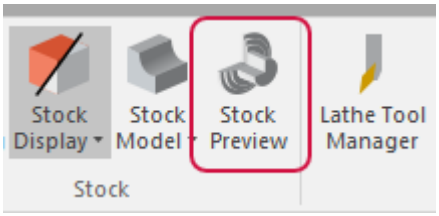


Creating Part Handling Operations That Use the Active Tool Plane

In Mill-Turn, manually created part handling operations now use the active tool plane by default. This provides a more consistent Mastercam experience by matching the behavior of part handling operations in Lathe.

Displaying the Stock Preview Directly from the Ribbon

Mastercam 2023 features a secondary location for the **Stock Preview** button on the **Turning** contextual tab for Lathe and Mill-Turn. Use this function to display the machined stock, chuck, tailstock, and steady rest boundaries with a single click.



As with earlier versions of Mastercam, you can still access **Lathe stock preview** and **Turning stock preview** from the right-click menu of the Toolpaths Manager.

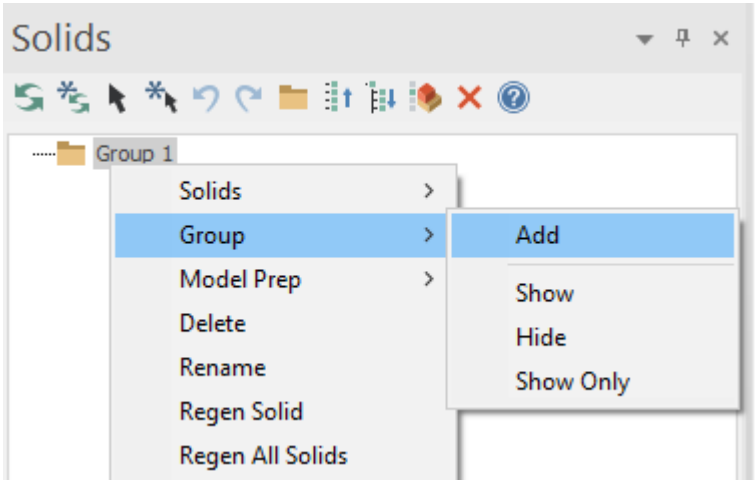
DESIGNING ENHANCEMENTS

Listed below are major enhancements to designing functions.

Organizing Your Solid Models into Groups

Mastercam 2023 enhances groups functionality in the Solids Manager with a new button in the toolbar and support for nested groups. The new **Add group** button lets you quickly create a group at the top level of the Solids Manager without using right-click menus.

The Solids Manger also includes support for nested groups to help you organize your work with solid models. To create a sub-group folder, right-click an existing group folder and select **Group**, **Add** from the menu. Then, click on the nested group to create a new body, or drag and drop existing items into the sub-group. Right-click a nested group folder and select **Group**, **Move to top level** from the menu to promote the group in the Solids Manager.



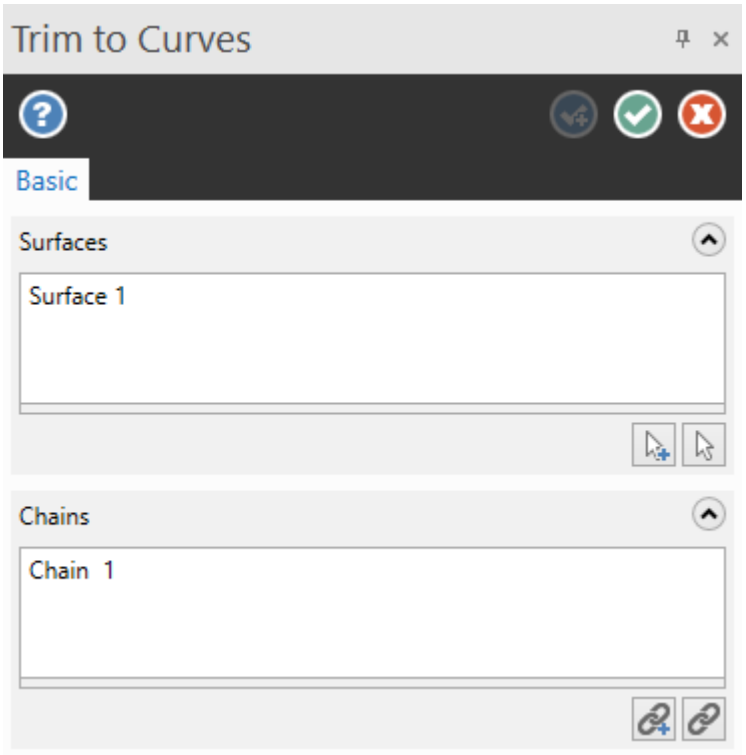
When you import solid models that include assemblies, sub-assemblies, and component parts, Mastercam 2023 automatically creates groups and sub-groups. These groups and the intuitive naming conventions they use will help you better manage the relationships between the solids in Mastercam.

Improving the Selection Workflow

Multiple **Wireframe** and **Surfaces** function panels now include a new selection grid for surfaces and chains, providing easier editing during the creation process. This applies to the following functions:

Surfaces functions

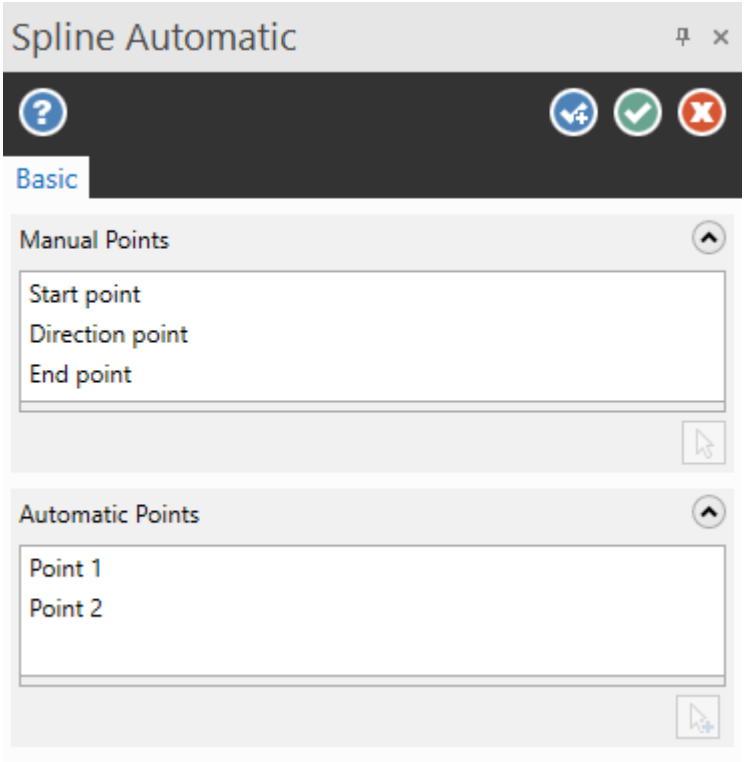
- Trim to Plane
- Fillets to Plane
- Trim to Curves
- Fillets to Curves
- Trim to Surface
- Fillet to Surface



Wireframe functions

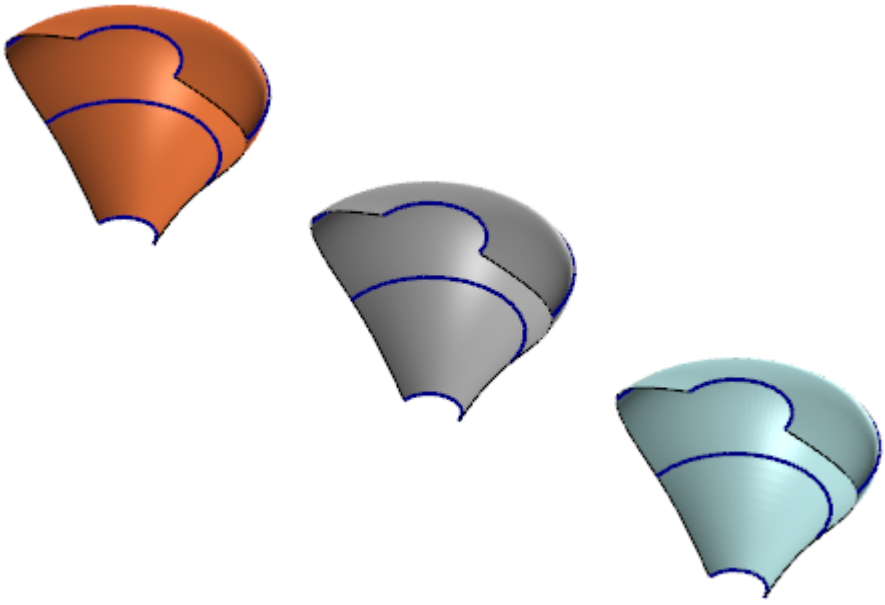
- Spline Automatic
- Curve at Intersection

Spline Automatic includes a selection grid for manual and automatic points. **Manual points** are defined by you to set the pattern of the spline. This includes the start point, direction point, and end point. **Automatic points** are used by Mastercam to define the spline. You can remove or add to these points. In previous releases, you had to exit the function and start over to make modifications.



Creating Additional Geometry Types

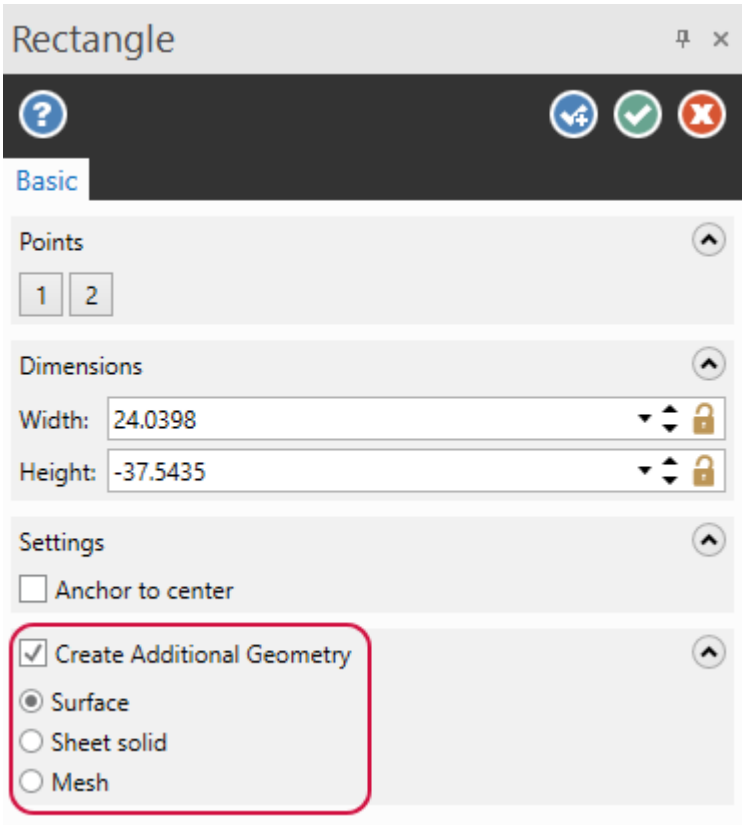
You can now create a mesh, surface, or sheet solid in **Wireframe** and **Surfaces** functions. In previous releases, you could only create surfaces inside wireframe functions which resulted in a closed shape or you had to create a surface and convert it.



This applies to the following functions:

Wireframe functions

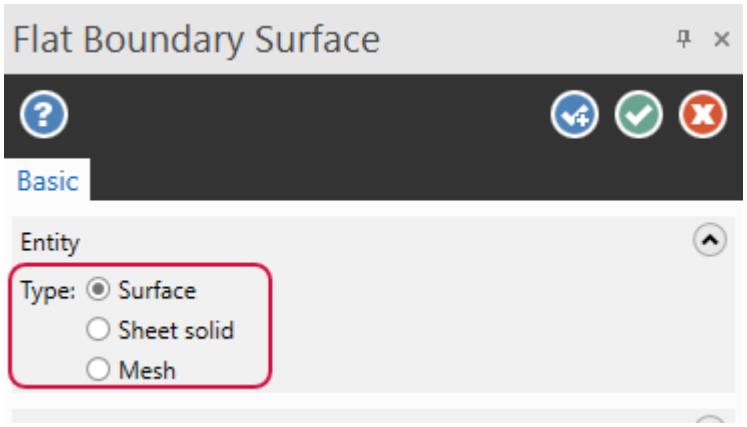
- Circle Center Point
- Circle Edge Point
- Rectangle
- Polygon
- Ellipse
- Rectangular Shapes



Surfaces functions

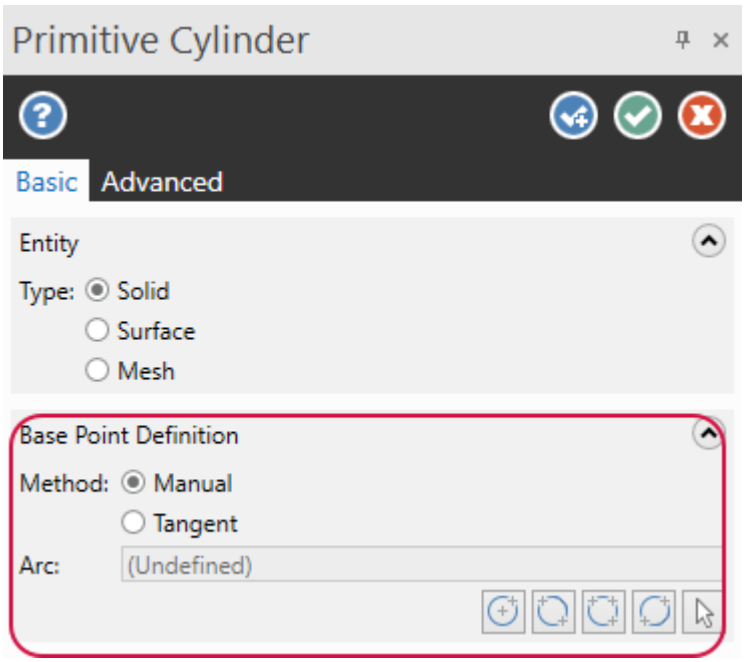
- Draft
- Lofted/Rules
- Extrude
- Revolved
- Flat Boundary
- Two Surface Blend
- Three Surface Blend
- Swept
- Net
- Fence

- Offset
- Three Fillet Blend
- Power Surface
- Fill Holes



Creating a Cylinder with Unknown Base Points

Primitive Cylinder now includes **Base Point Definition** options. These new options allow you to create a cylinder without knowing in advance where the center point is.

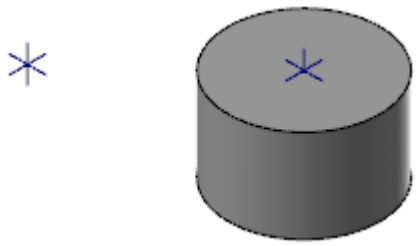


You choose between two **Methods**:

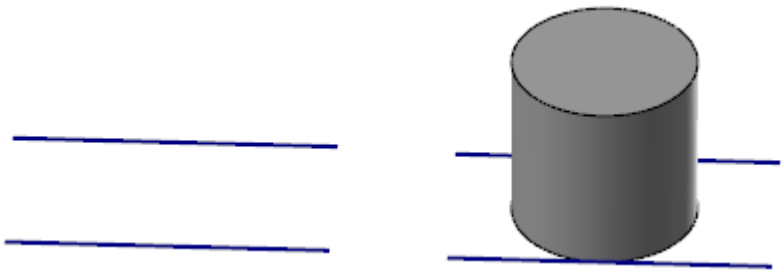
- **Manual**: Creates the cylinder according to your selected options and positions.
- **Tangent**: Creates the cylinder tangent to existing entities, according to the selected options.

Then, you can choose between the following:

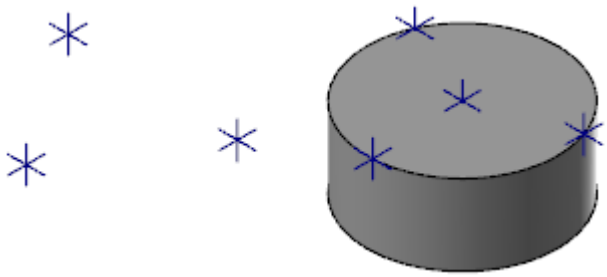
- **Center point**: Select a center point and set the radius. This is the default arc method.



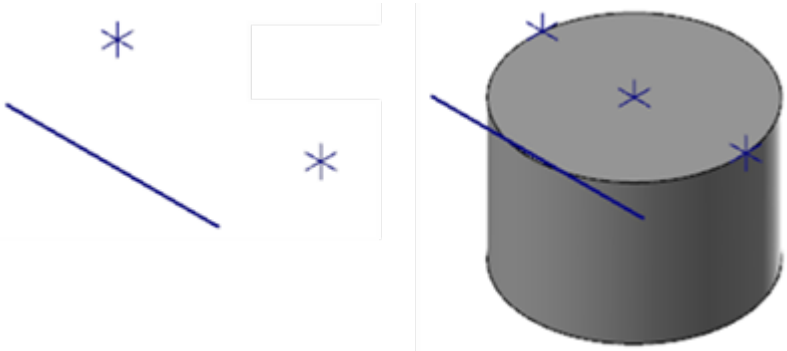
- **Two edges**: Select two edge positions that the cylinder's diameter passes through. The edge positions determine the center point location.



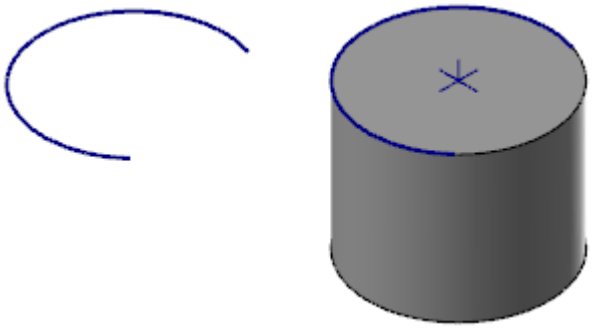
- **Three edges**: Select three edge positions that the cylinder's diameter passes through. The edge positions determine the center point location.



- **Endpoints**: Select two edge positions and an edge point to define the cylinder's center point and edges.



- **Geometry**: Select an arc or radial face that defines the center and the radius of the cylinder.

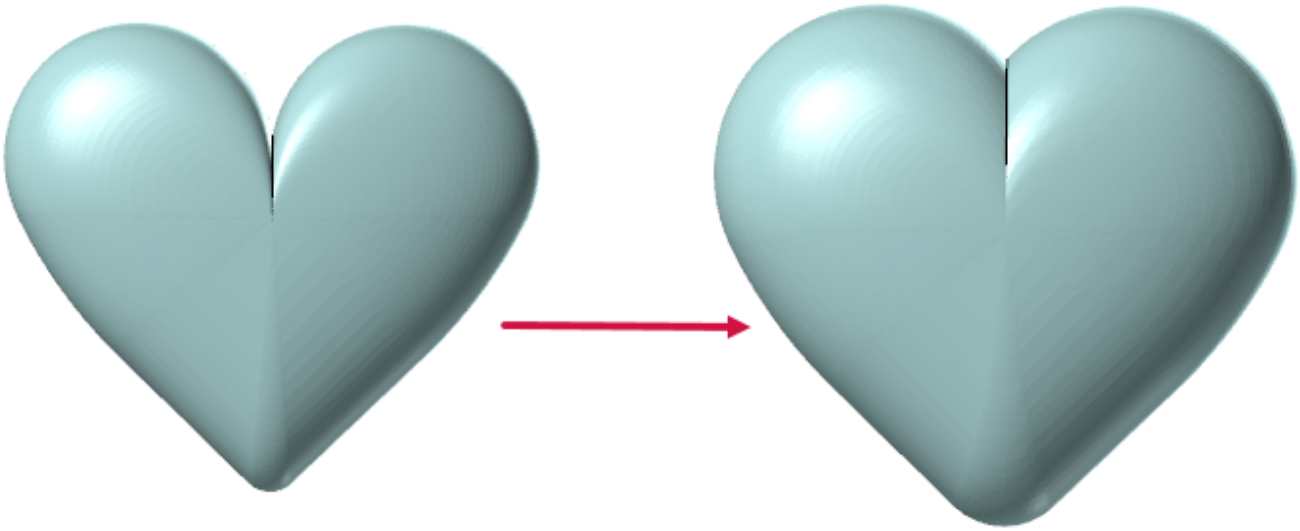


Mesh Enhancements

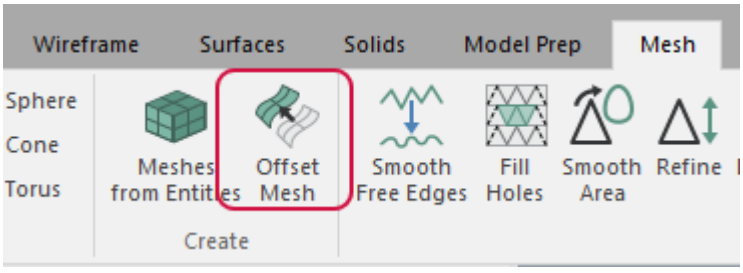
Listed below are enhancements for the functions located on the **Mesh** tab.

New Offset Mesh Function

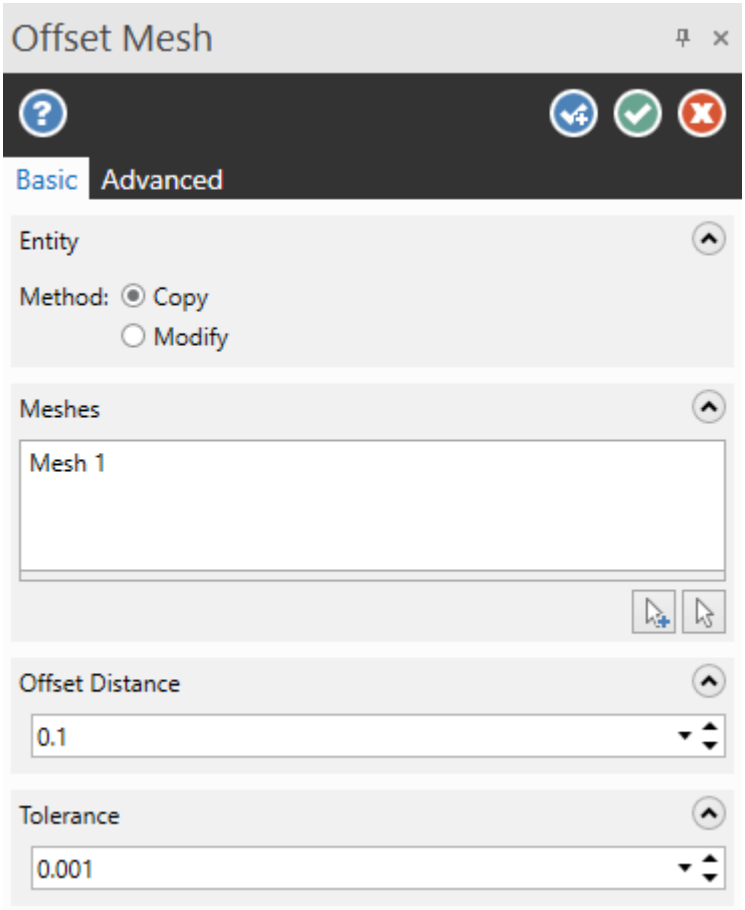
To offset a mesh entity in previous versions of Mastercam, you had to create a stock model operation and then convert the stock model to a mesh. Mastercam 2023's new **Offset Mesh** function offers you a quick and easy way to achieve the same outcome.



To start the function, select **Offset Mesh** from the **Mesh** tab.



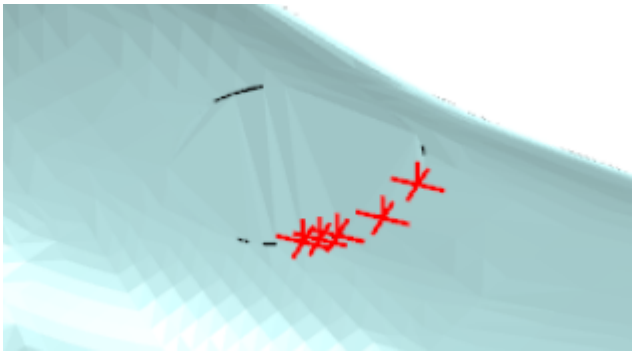
Then, select the mesh or meshes you wish to offset, and enter a positive **Offset distance**. Mastercam uses a 3D spherical offset to create new meshes or modify existing ones.



Automatically Check and Fix Mesh Results

The **Smooth Free Edges**, **Fill Holes**, **Smooth Area**, **Refine**, and **Decimation** functions now warn you during **Preview** or creation if you are creating a problematic facet.

If you click **OK** or **OK and Create New Operation**, a dialog box displays. You can delete the problematic facets at that time or keep them. Mastercam also marks these facets using a red point in the graphics window, however these indicators are removed after selecting **Yes** or **No** in the dialog box.

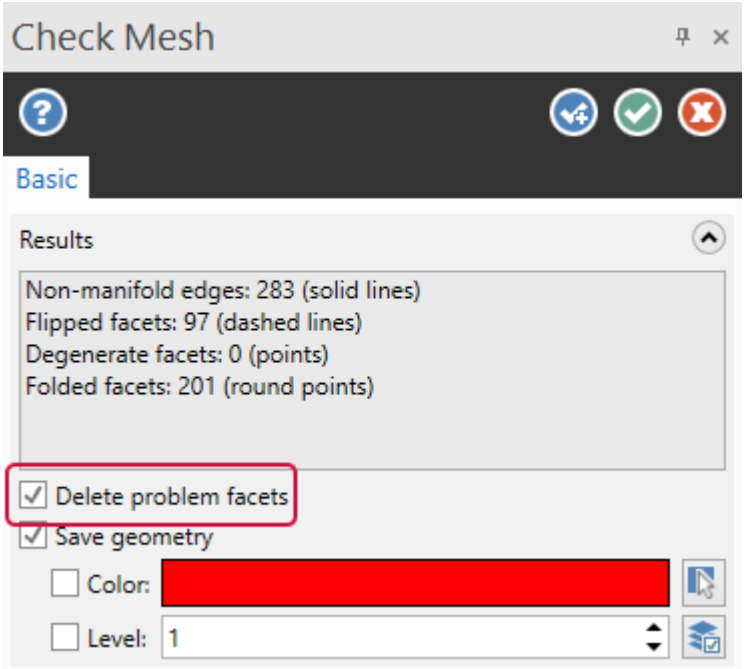


Check Mesh Enhancements

Listed below are enhancements made to the **Check Mesh** function.

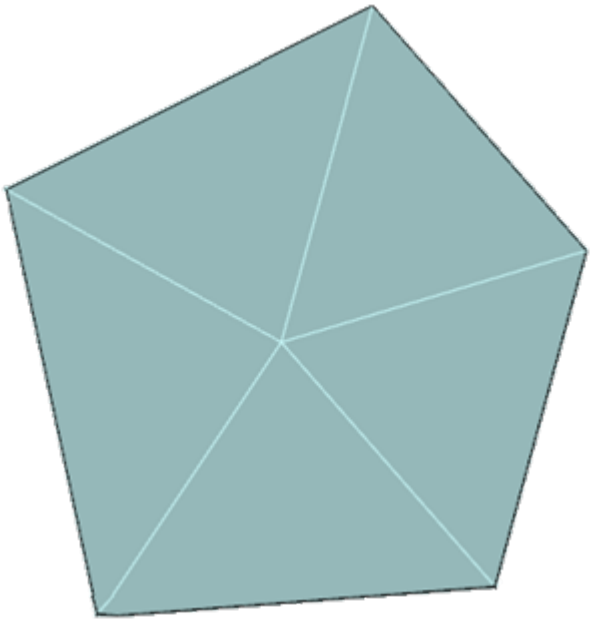
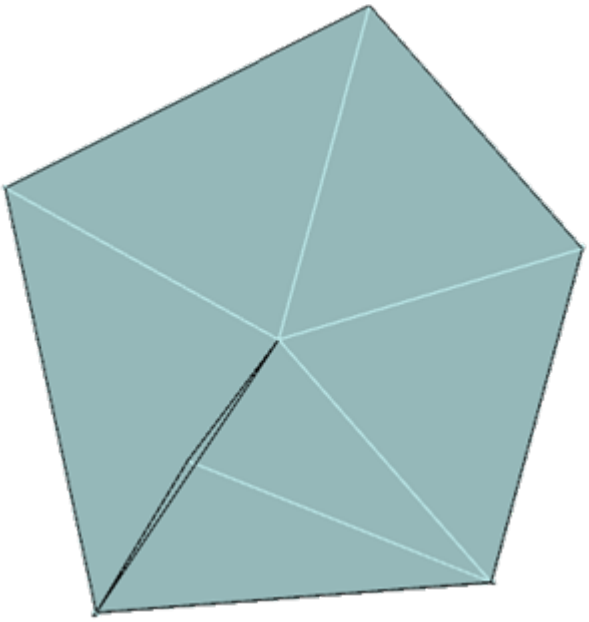
Deleting Problematic Facets

Check Mesh now includes a new option to delete problem facets. By selecting this option, Mastercam deletes all facets identified with errors.

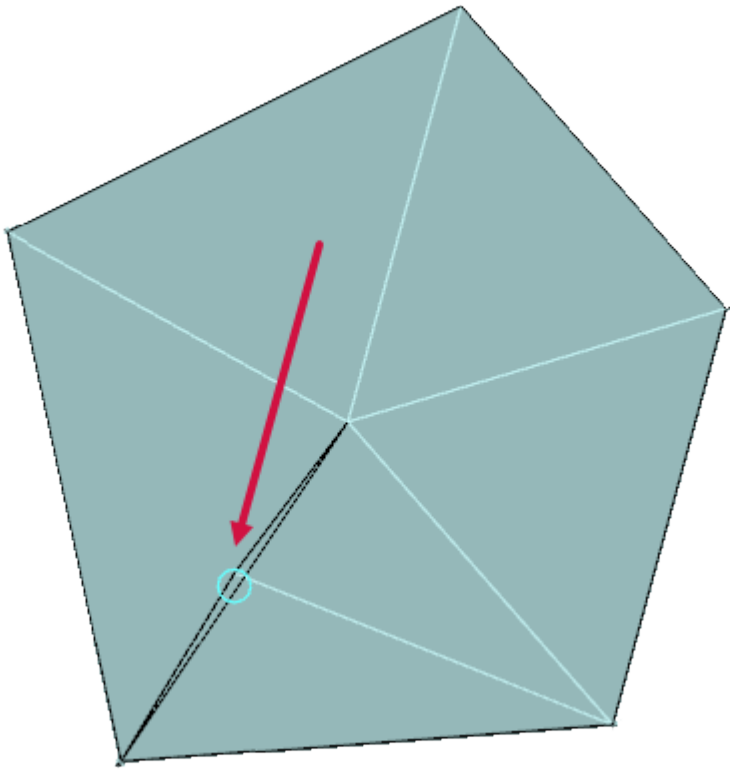


Detecting Folded Facets

Check Mesh now detects folded facets, where a neighboring node is pulled over an existing facet, creating a crease which has no geometric meaning.

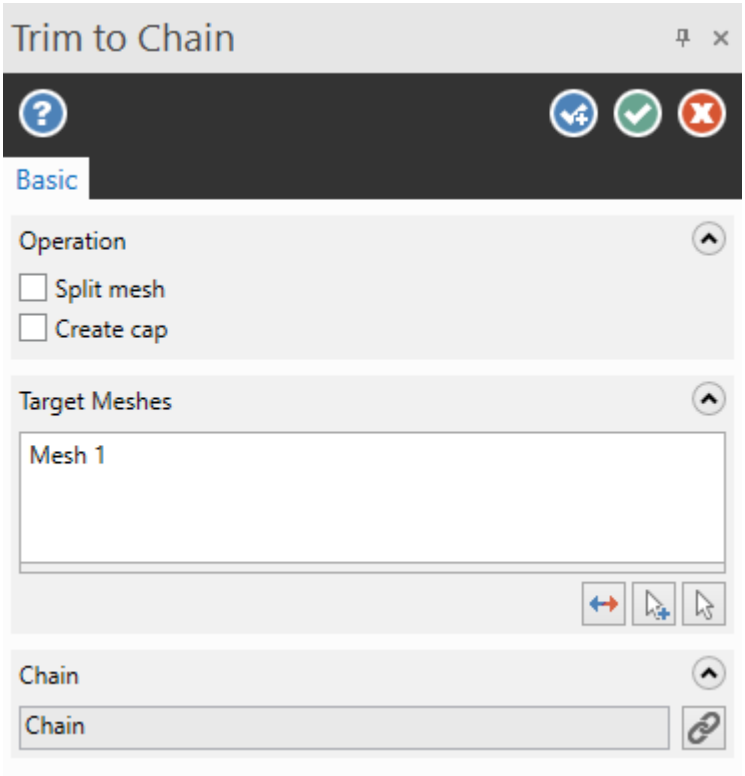
Correct	Incorrect
	

If **Check Mesh** discovers a folded facet, Mastercam marks the bad facet with a round-style point indicator.

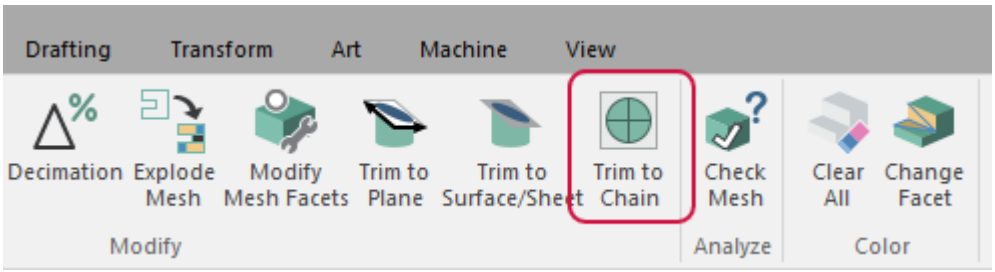


New Trim to Chain Function

The new **Trim to Chain** function allows you to trim selected mesh entities by a single chain, using the current Cplane for the direction. Additionally, you can split the mesh where it intersects with the chain and create caps by closing free edges of the mesh created by the trim.

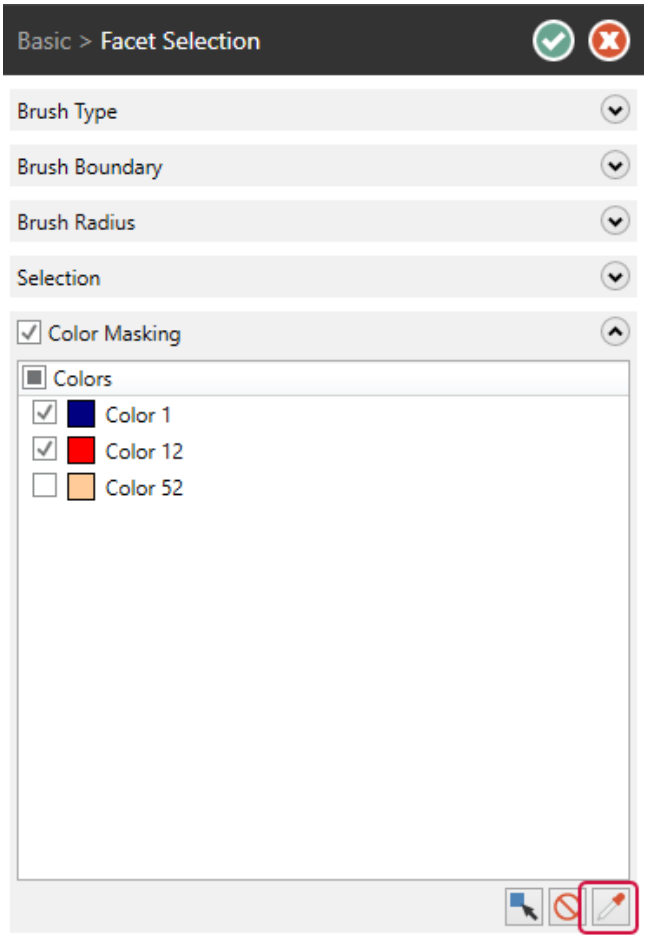


Trimming a mesh is helpful when you need to remove features or create multiple meshes from an individual mesh. To start the function, select **Trim to Chain** from the **Mesh** tab.



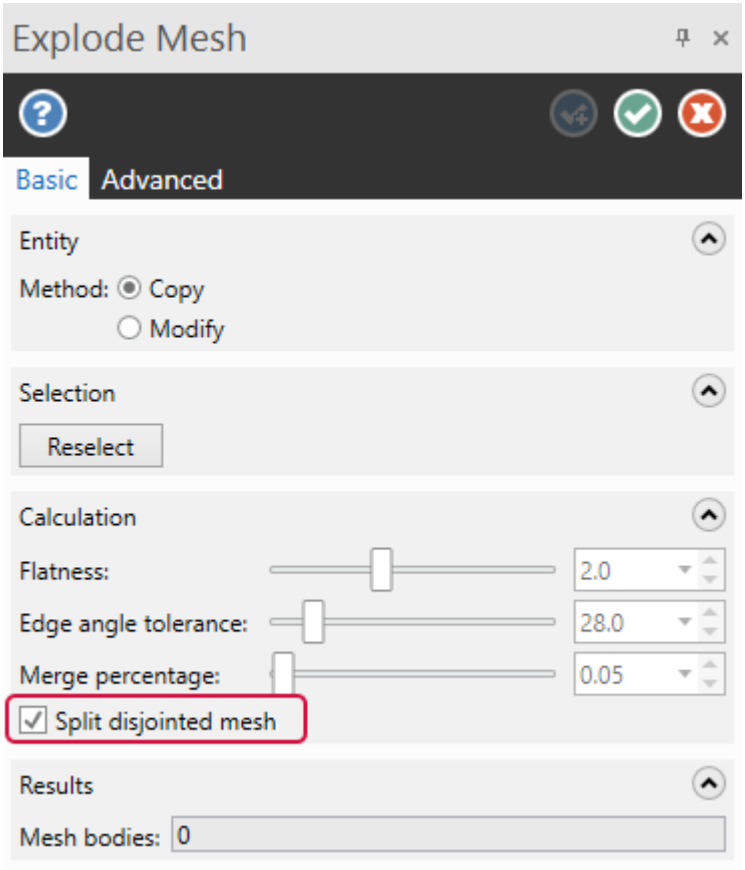
Improving Your Facet Selection

Facet selection for **Decimation**, **Refine**, and **Smooth Area** has been improved. You can now select facets by color using the new **Color Masking** options in the **Facet Selection** fly-out. Color masking allows you to select facets that match the colors you select, saving time by having Mastercam automatically select the facets for you. The **Sample facets** button allows you to precisely add or remove a facet color from the masking list if your mesh contains entities with subtle color differences.

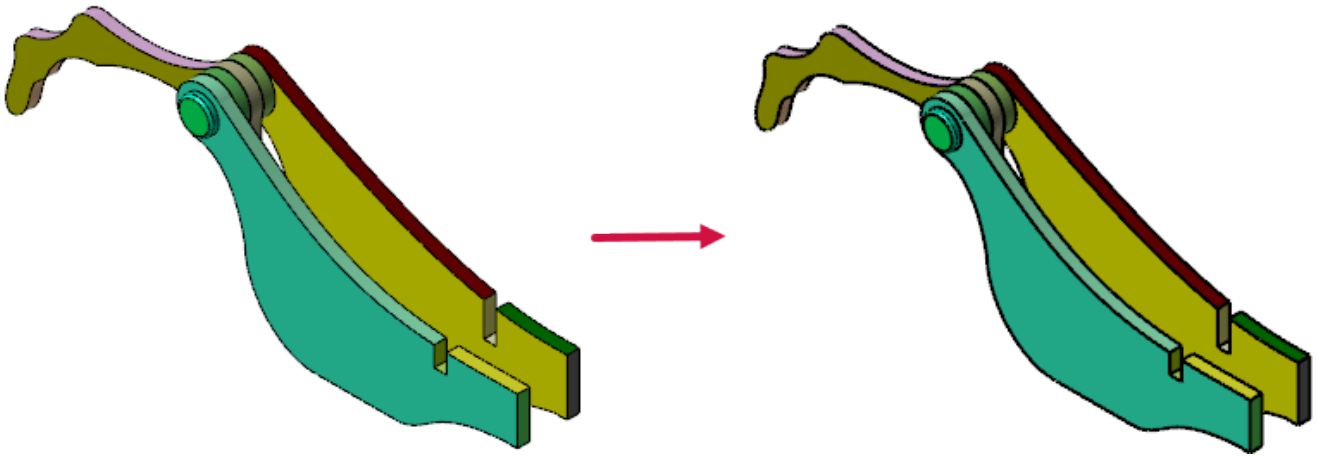


Automatically Split Meshes

Explode Mesh now includes the new **Split disjointed mesh** option. This new option automatically separates the mesh model along existing disjointed areas, instead of using the calculation fields. If you select this option, the **Flatness**, **Edge angle tolerance**, and **Merge percentage** parameters are disabled.

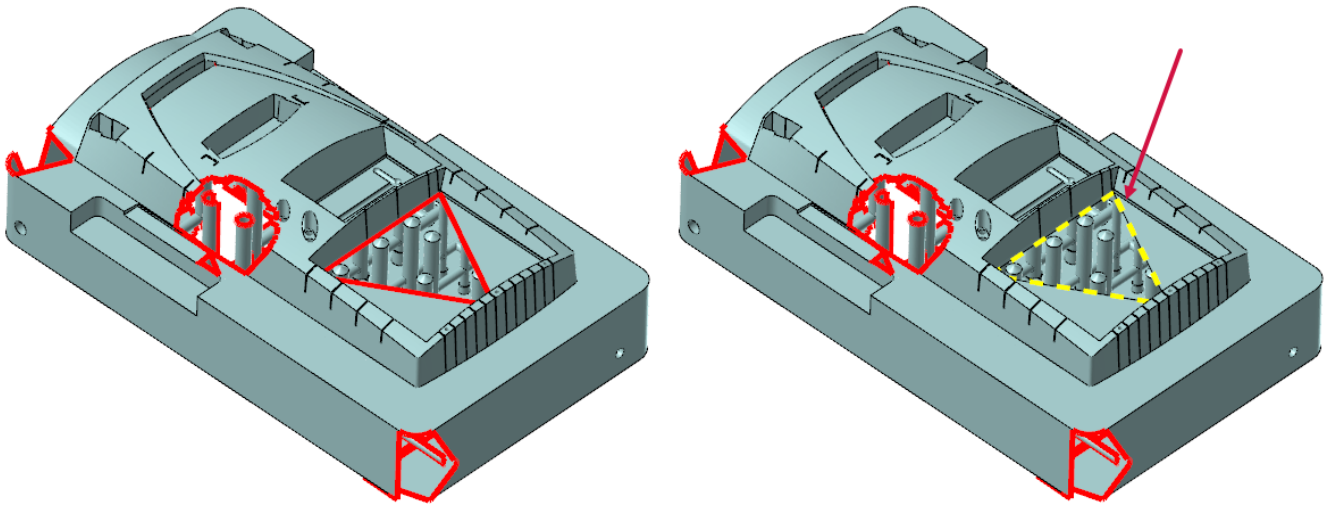


Additionally, the boundaries between the resulting meshes are more apparent.



Selecting Edges to Fill Holes

Fill Holes now allows you to select edges when selecting holes to fill. Additionally, **Fill Holes** includes a **Preview** button on the **Basic** tab, allowing you to see your modifications without leaving the function panel.



Wireframe Enhancements

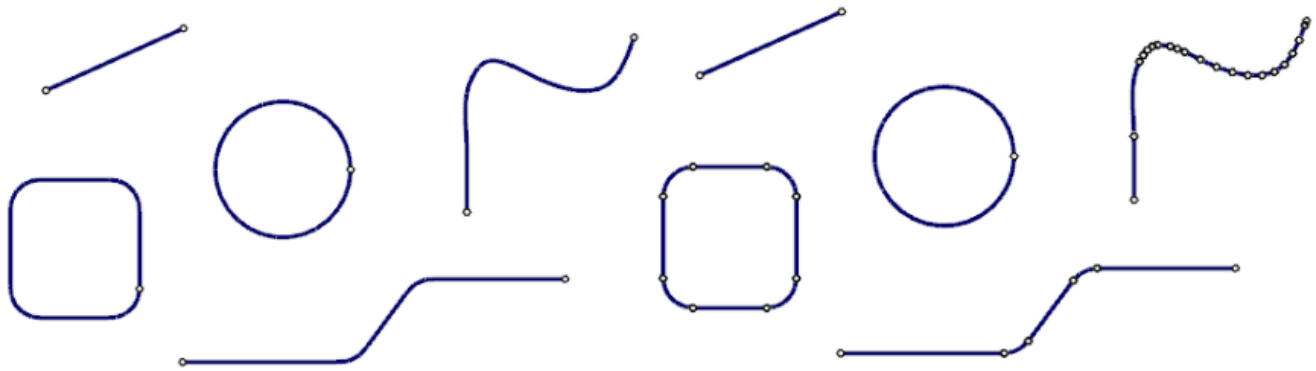
Listed below are enhancements for the functions located on the **Wireframe** tab.

Spline Enhancements

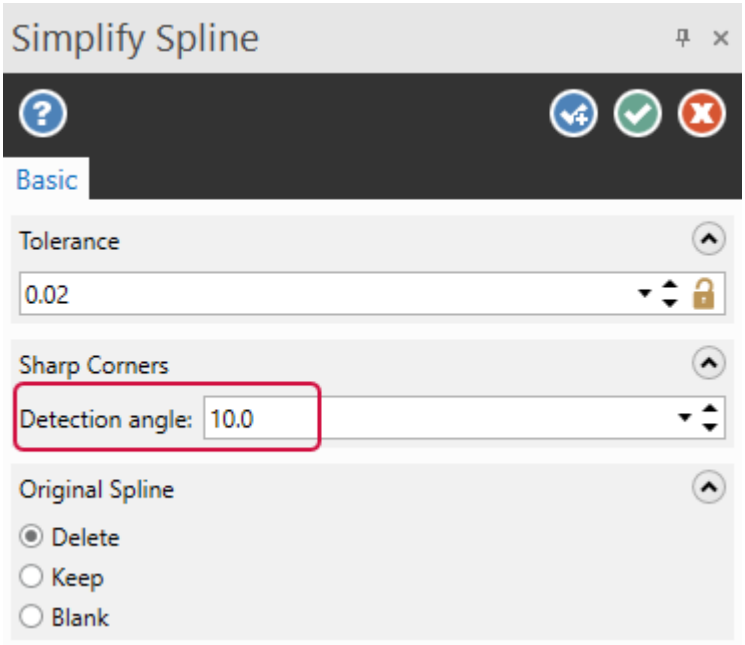
Listed below are enhancements made to **Spline** functions.

Converting Complex Spline Shapes

In previous releases, **Simplify Spline** had difficulty recognizing more complex spline shapes to convert into arcs and lines. In Mastercam 2023, **Simplify Spline** is aware of these complex shapes, and creates arcs and lines as necessary.

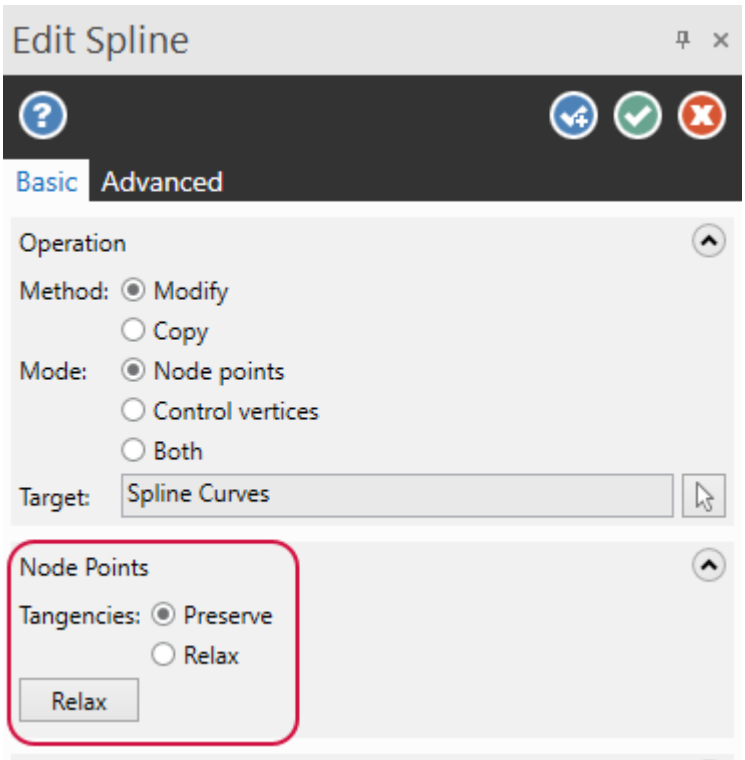


Simplify Spline also includes a **Detection angle**, which defines the angle at which the node is considered a sharp corner.

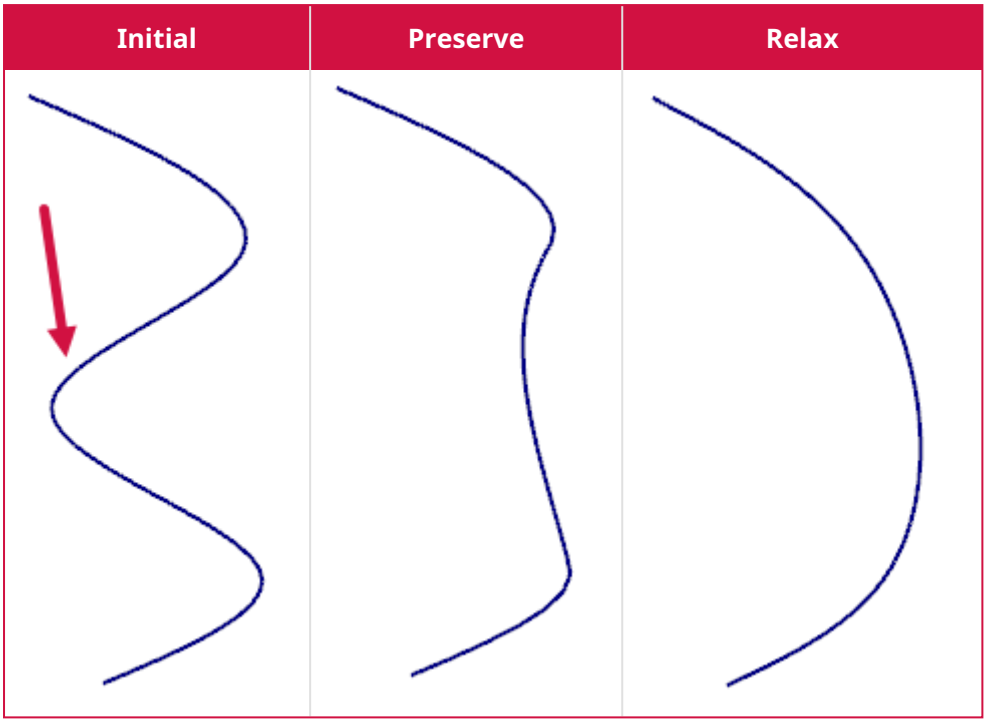


Modifying Spline Tangencies

Edit Spline includes a new **Relax** option. When you edit or delete a node, this new option modifies the spline as if the node was never part of it. In previous releases, when you modified a node point, the pre-existing tangencies of the spline did not change.

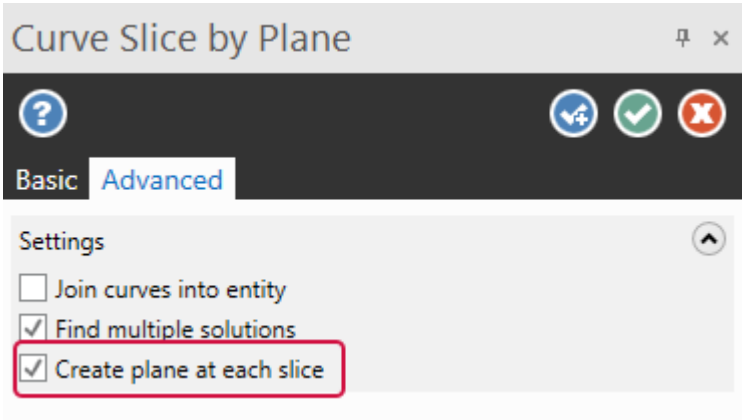


The examples below show how the options affect the spline after deleting the node indicated by the red arrow.

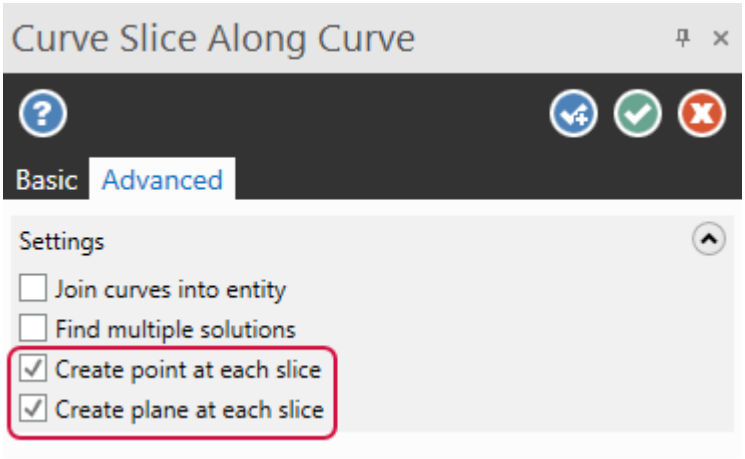


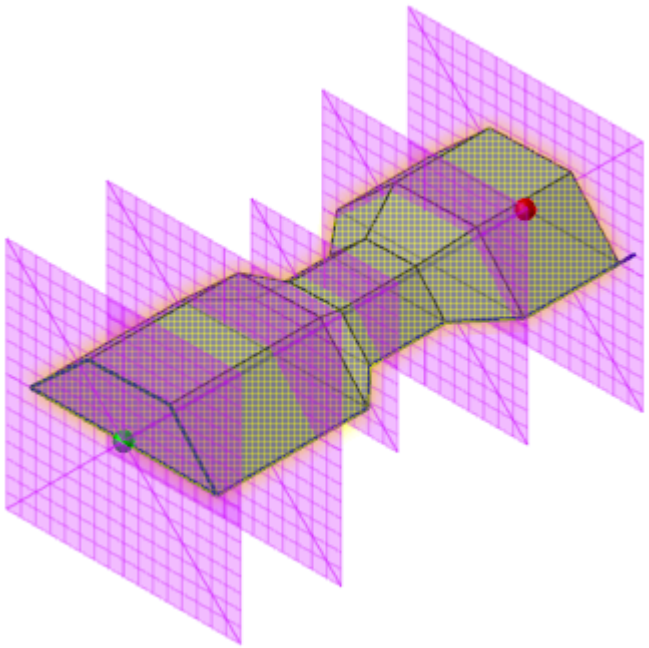
Creating Planes and Points for Each Curve Slice

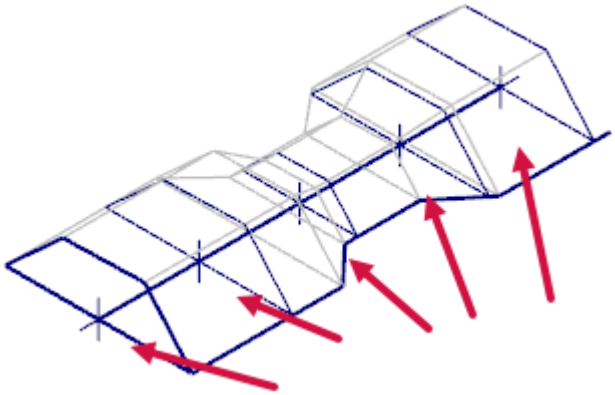
Curve Slice by Plane and **Curve Slice Along Curve** now include the **Create plane at each slice** option in the **Advanced** tab. When selected, Mastercam creates a plane at each curve slice along the selected chain or plane. This allows you to create planes automatically to use for other functions, such as creating a toolpath.



Curve Slice Along Curve also includes the **Create point at each slice** option, which creates a point at each curve slice on the **Along curve** that can be reused later to determine where slices will occur.

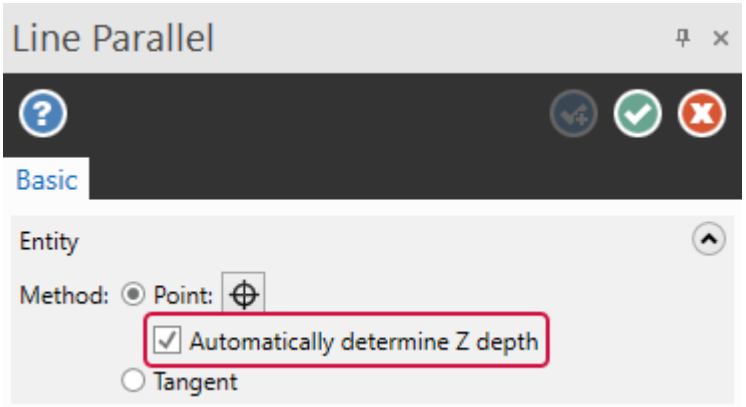






Automatically Determine Z Depth

Line Parallel now includes the **Automatically determine Z depth** option when creating a line using the **Point** method. This option keeps new, non-AutoCursor points at your first endpoint's AutoCursor Z depth. This option only applies in **3D** mode.



Linear Extension to Splines

Mastercam has always been able to trim and extend most wireframe entities to one another. However, these capabilities did not include splines, which were limited to only trimming. In previous versions, to extend a spline you had to use top-level editing (introduced in Mastercam 2022) or **Modify Length** to extend the spline prior to trimming. In Mastercam 2023, the **Trim to Entities** function now adds a linear extension to splines that intersect with other selected wireframe entities.

This is especially useful when you need to clean up wireframe in imported files that contain splines.

2D/3D Construction Mode Support

Mastercam now includes improved support for switching between **2D/3D** construction modes. You can now change the **2D/3D** mode while some entities are live. This applies to the following functions:

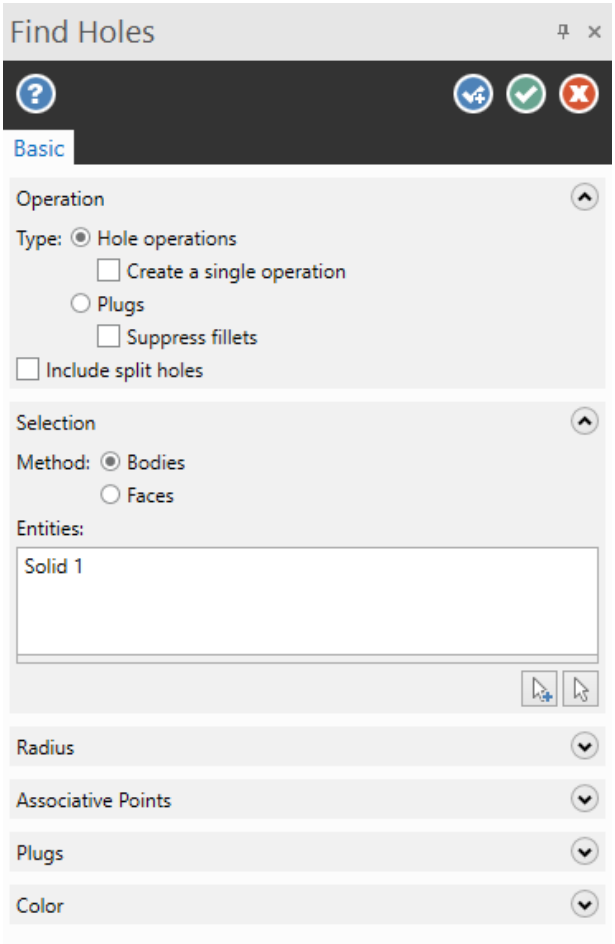
- Spline Blended
- Circle Edge Point
- Arc 3 Points
- Circle Center Edge Point (also supports changing the Cplane while the circle is live)

Model Prep Enhancements

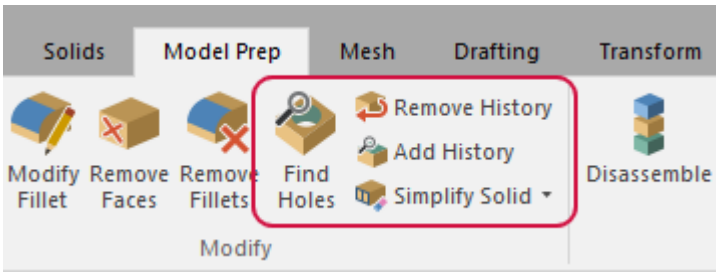
Listed below are enhancements for the functions located on the **Model Prep** tab.

Detecting Holes in Solid Bodies

The hole detection capabilities of **Add History** have been broken out into their own function called **Find Holes**. **Find Holes** detects holes in one or more solid bodies with no history. When Mastercam detects a hole, use **Find Holes** to detect holes or plugs, and create new operations in the Solids Manager.

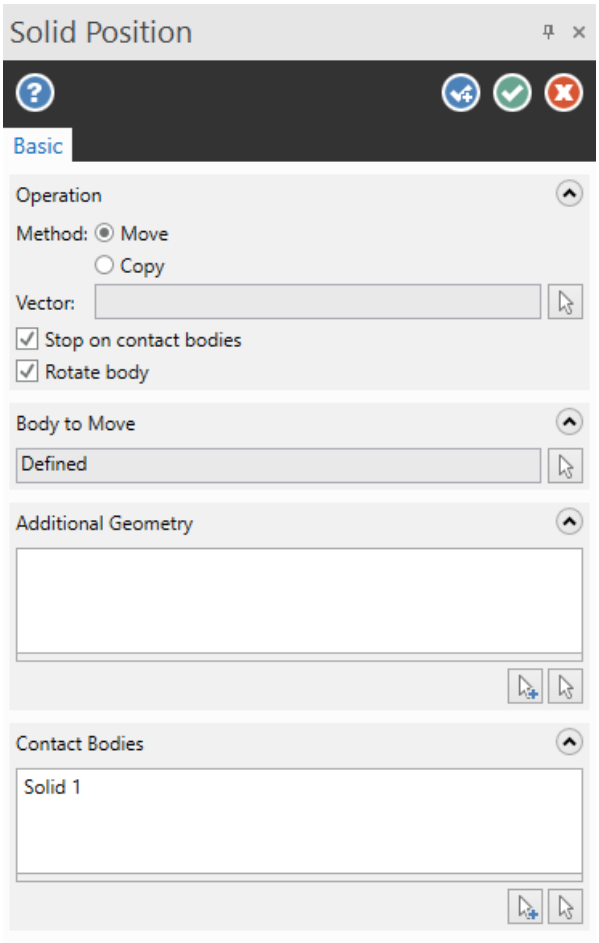


Add History continues to detect fillets and create extrusion operations from round holes on solid bodies. Both functions are located on the **Model Prep** tab.



Positioning Solid Bodies

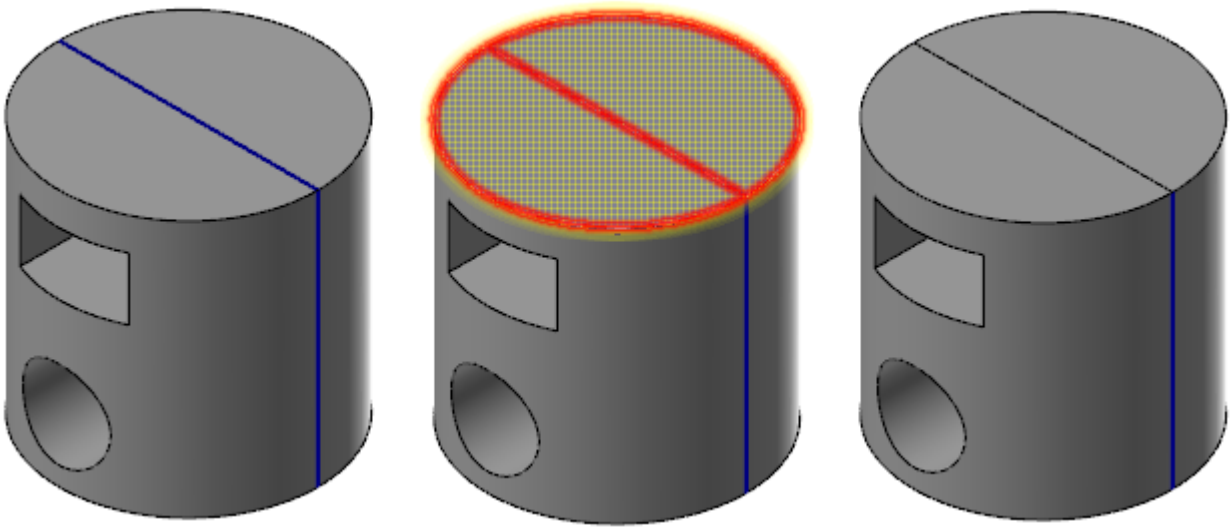
The new **Solid Position** function, located on the **Model Prep** tab, allows you to place a solid body against multiple solid bodies such as vices or clamps. This eliminates the need for multi-step processes using other functions to achieve your goal. This new function uses Mastercam's collision checking technology to accurately place the solid against the selected contact bodies.



To use this function, select a face on the solid body to move and adjust the gnomon to orient the movement along its axes. You can rotate the part or use the **Vector** option to quickly align the part's X axis along a vector that you define. Then, select the contact bodies. **Contact bodies** set the boundaries for the movement of your solid body. Use the gnomon to move the body into position.

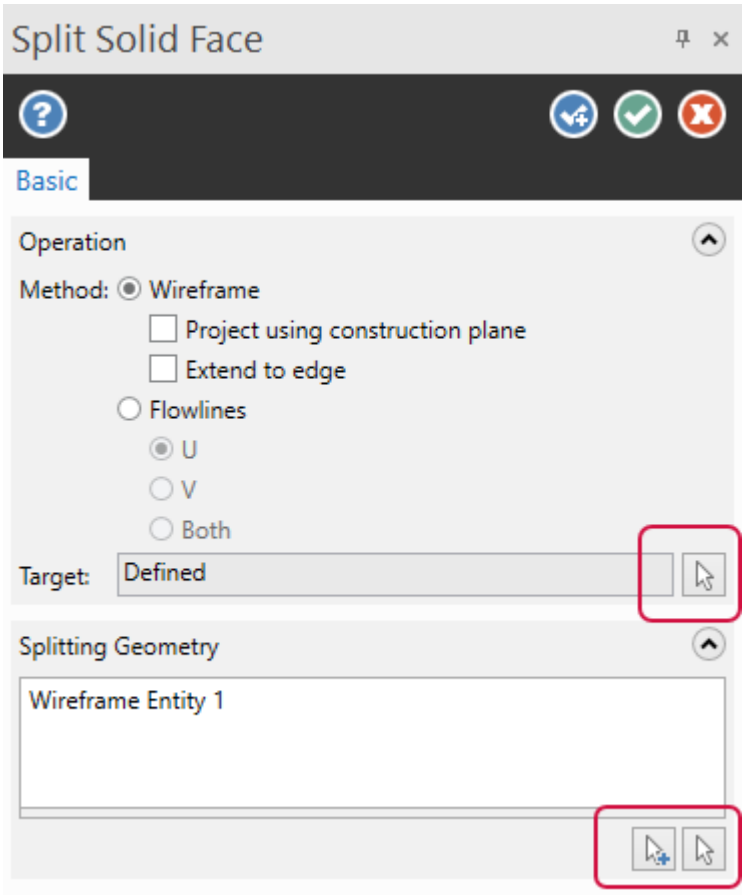
Improving the Workflow for Split Solid Face

Mastercam 2023 introduces a new, clearly defined selection process for splitting faces with wireframe geometry. In previous versions, when you used the **Wireframe** method to split the face, Mastercam prompted you to select both the face and the splitting geometry. This was problematic if the wireframe was on top of the face. In Mastercam 2023, this process is divided into two distinct steps. Only after you select a target can you select the splitting geometry.



Providing Consistent Controls and Greater Flexibility

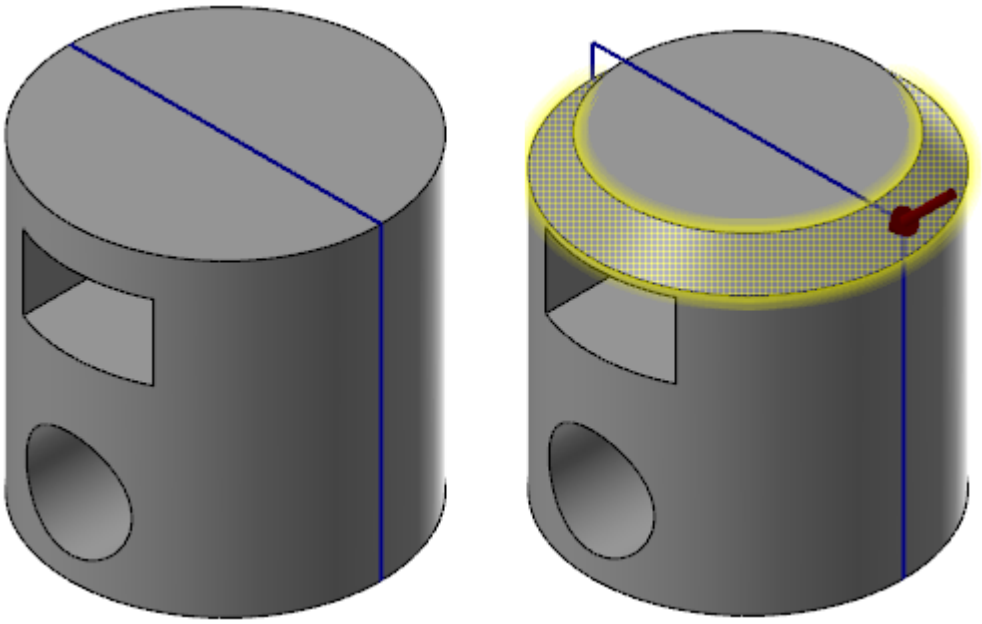
Split Solid Face includes standard Mastercam controls to reselect the face and trimming geometry. This provides more flexible selection and is consistent with other Mastercam functions.

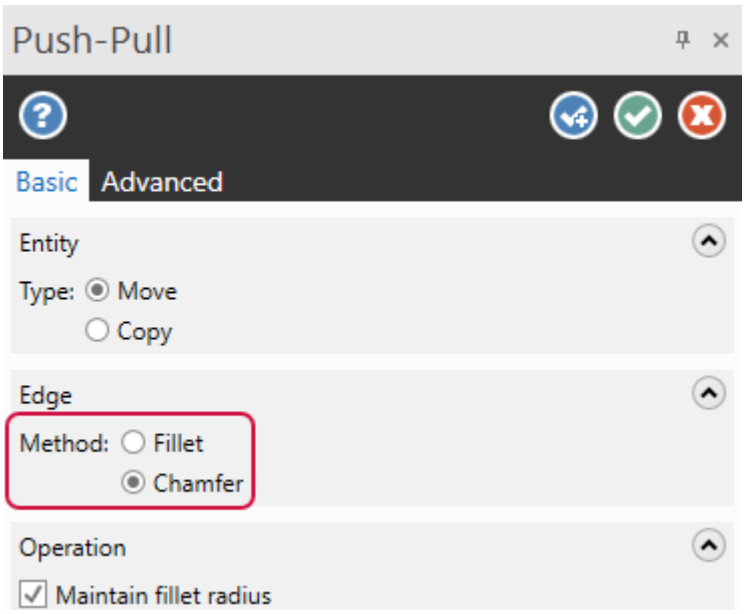


When you select a face to trim, the **Target** parameter changes to **Defined**, and the **Reselect all** button becomes active allowing you to choose a different face at any point in the process. Then either the **Splitting Geometry** or **Position** group is enabled depending on your selected **Method**.

Creating Chamfers with Push-Pull

Push-Pull now includes the new **Chamfer** option, which allows you to push the selected edges to create a one-distance chamfer. Push or pull the selected edges to modify existing chamfers. This new option makes it easy to create a chamfer on a solid edge.



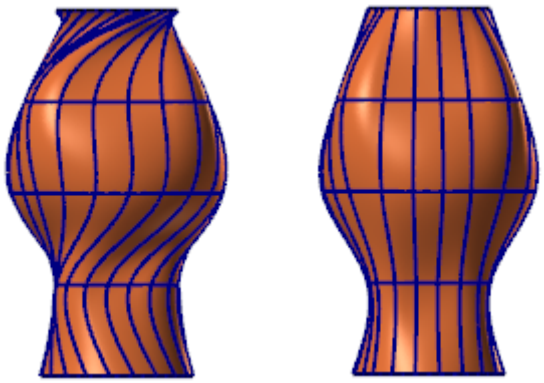


Surfaces Enhancements

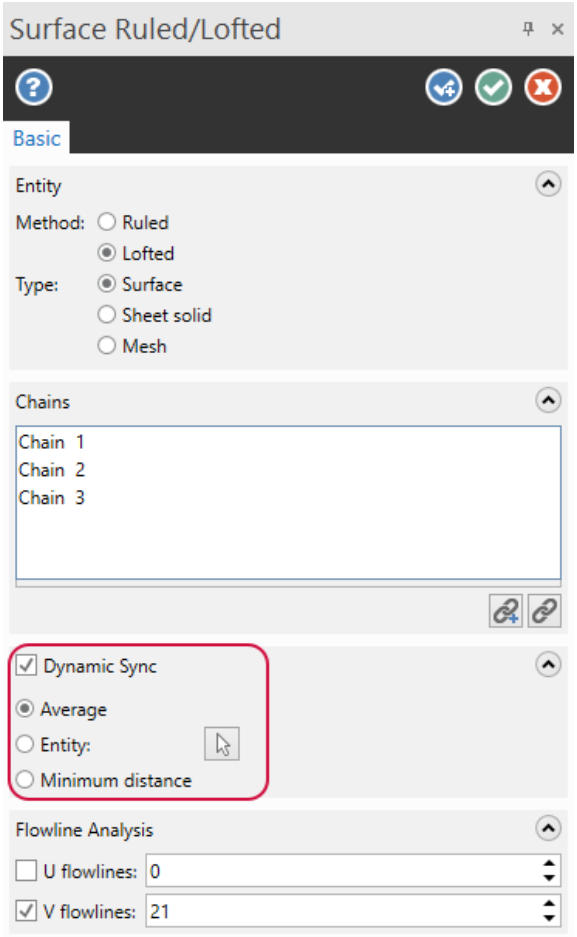
Listed below are enhancements for the functions located on the **Surfaces** tab.

Creating Consistent Flowlines with Surface Ruled/Lofted

Irregular flowlines on a surface can sometimes occur when the surface is created from independent curves with inconsistent point spacing. Connecting these points of constant parameter values, as the **Surface Ruled/Lofted** function does, may produce undesirable flowlines, which may result in an undesirable surface shape. The new **Dynamic Sync** option redefines the way existing points are arranged on the selected chains to produce smoother, more consistent flowlines.

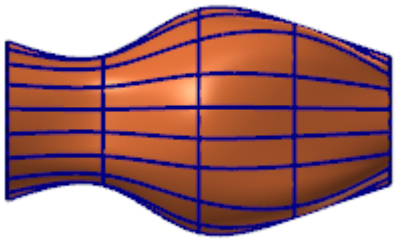


By generating an imaginary spine curve from your selection, **Dynamic Sync** provides an invisible "backbone" to the surface flow. This backbone provides a new set of aligned points to the wireframe chains.

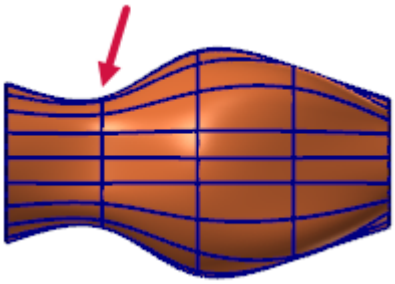


There are three ways to generate **Dynamic Sync**'s spine curve.

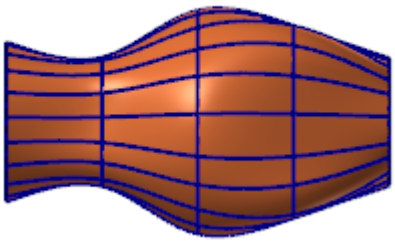
- From an **Average** of all the selected chains. The spine curve creates points on the wireframe that are perpendicular to it.



- From an **Entity** you select. The generated surface uses the selected entity's point spacing to create points that are perpendicular to it on the wireframe.



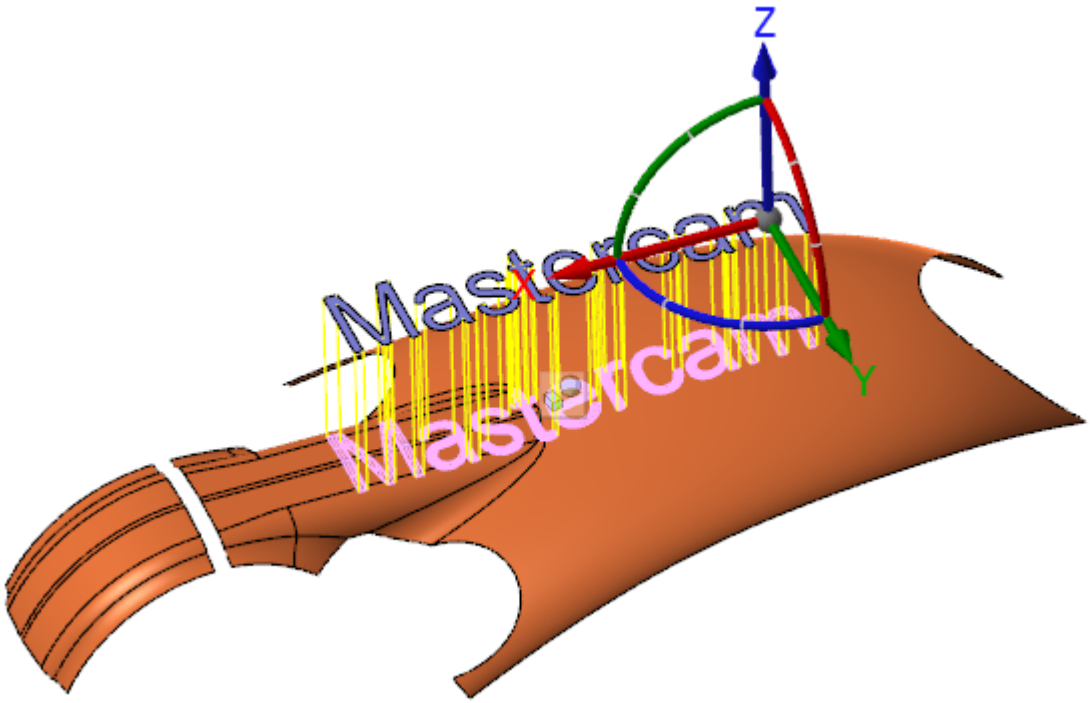
- By projecting a set of points on each curve that are perpendicular at the **Minimum distance** to the one before it. Use this option when a spine curve from **Average** and **Entity** options cannot be generated or produces an undesirable result.



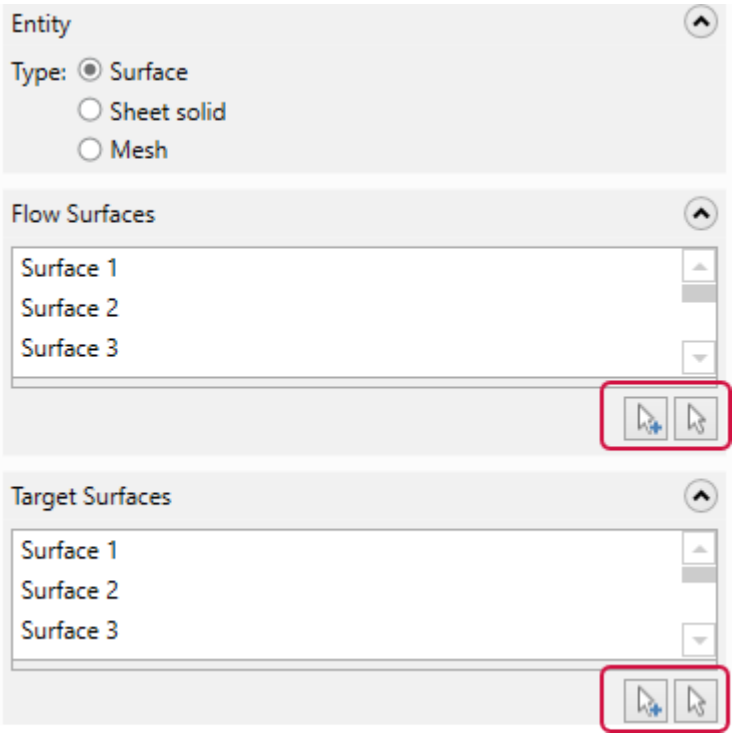
In conjunction with Dynamic syncing capabilities, Mastercam brings additional enhancements to **Surface Ruled/Lofted** that give you more control of your chain selections and your surface creation. **Surface Ruled/Lofted** now includes a selection list which allows you to view the chains you have selected. You can add, remove, reverse, or reorder your chains using the right-click menu or dragging and dropping a chain. You can also use the **Select chains** and **Reselect chains** buttons located below the list box.

Supporting Multiple Flow Surfaces in Overflow UV

The original implementation of the **Overflow UV** function in Mastercam 2022 supported flowline projections from a single flow surface. In Mastercam 2023, **Overflow UV** supports flowline projections from multiple flow surfaces. This means you no longer need to re-run the function for each surface you wish to project to a set of target surfaces.



The panel and workflow have been modified to use standard Mastercam selection techniques to add, remove, and reselect flow and target surfaces.



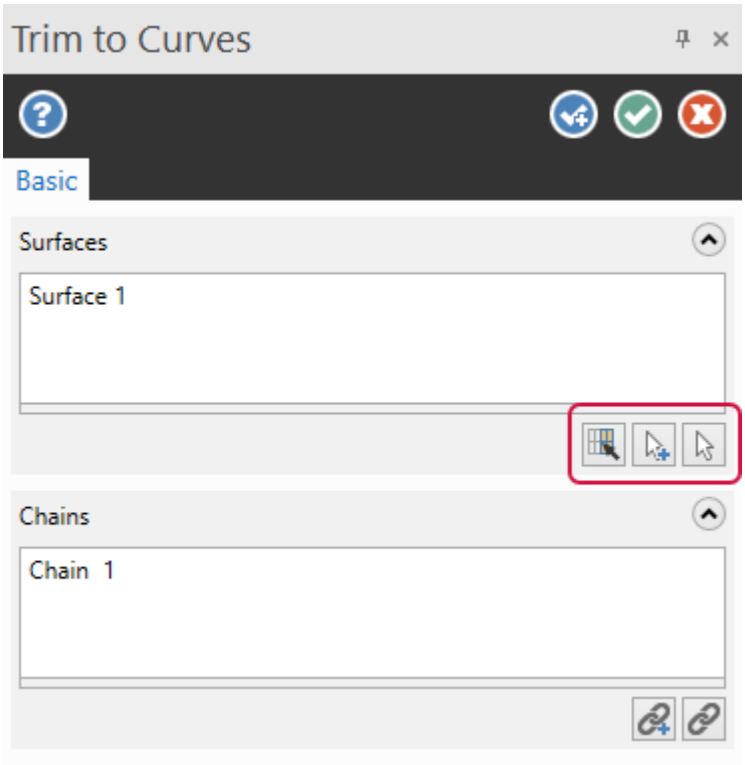
Additionally, **Overflow UV** now allows live entity conversion so you can quickly create a single mesh or sheet solid from multiple target surfaces.

Improving Selection for Trim to Surface and Trim to Curves

The selection for the **Trim to Surfaces** and **Trim to Curves** functions have improved for Mastercam 2023. The improvements allow for greater flexibility when selecting trimming geometry.

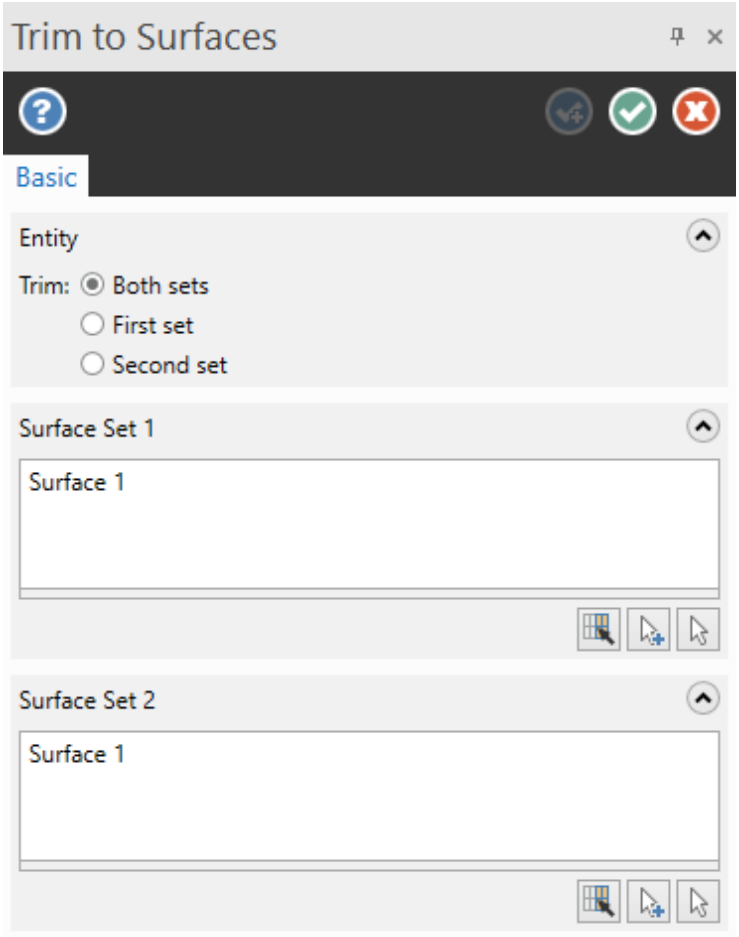
There are three new buttons: **Area to keep**, **Add selection**, and **Reselect all** for both functions. The **Add selection** and **Reselect all** buttons can be found throughout Mastercam and they work exactly the same in these two functions. The new **Area to keep** button allows you to change which area should remain if you do not get the results you were expecting from your first trimming attempt.

When trimming with **Trim to Curves**, occasionally surfaces would be left behind in the graphics window. Mastercam 2023 now properly removes these surfaces from the part.



Additionally, the **Keep multiple regions** option from both functions and the **Interference check boundaries** option in **Trim to Curves** have been removed. Both functions now create multiple regions by default, and improvements to performance and processing have rendered the **Interference check boundaries** unnecessary.

Trim to Surfaces now supports the selection of multiple surfaces for both surface sets. In previous releases, you were limited to a single surface in one of the sets. When using **Split model**, Mastercam now organizes the surfaces by color. The original surfaces retain their color, but the newly split model inherits the current surface color.



Drafting Enhancements

Listed below are enhancements for the functions located on the **Drafting** tab.

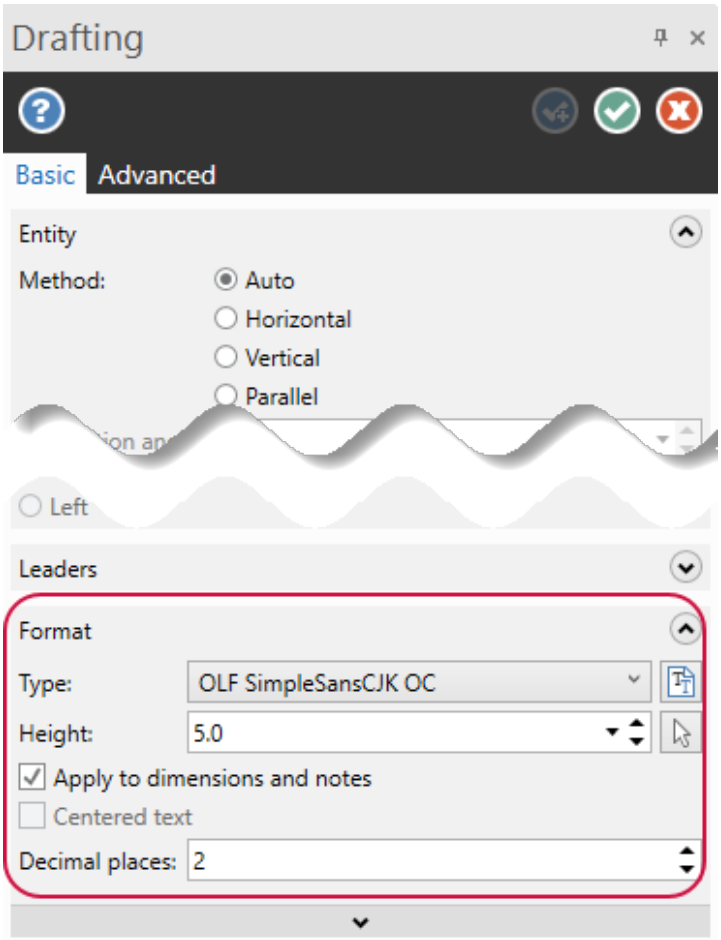
Top-Level Editing of Drafting, Label, and Note Entities

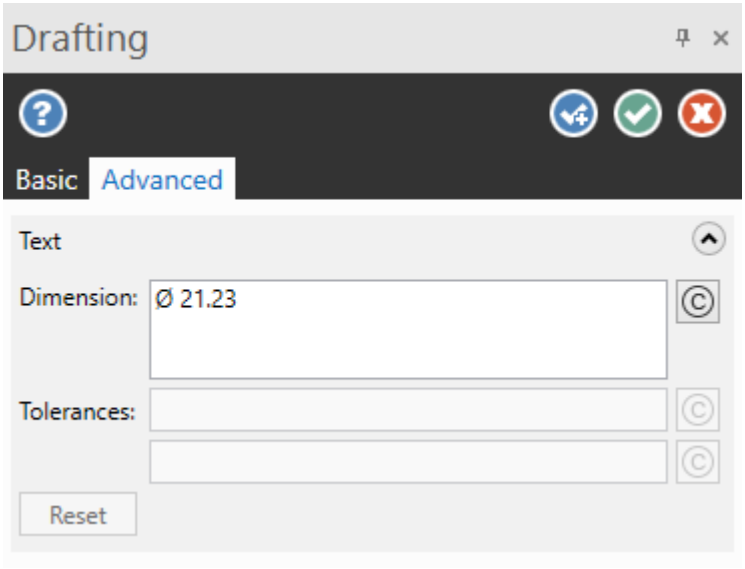
Mastercam 2023 makes it easier to access and edit notes, labels, and dimensions using top-level editing. In previous releases, you could use the **Edit text** or **Quick edit** buttons in the **Analyze** function to modify the text or re-open the **Note**/**Drafting** functions to edit the text and position.

Now, if no other function is active, you can double-click existing notes, labels, and dimensions in the graphics window to open the **Note** and **Drafting** function panels to make edits.

Modifying Your Drafting Font Directly

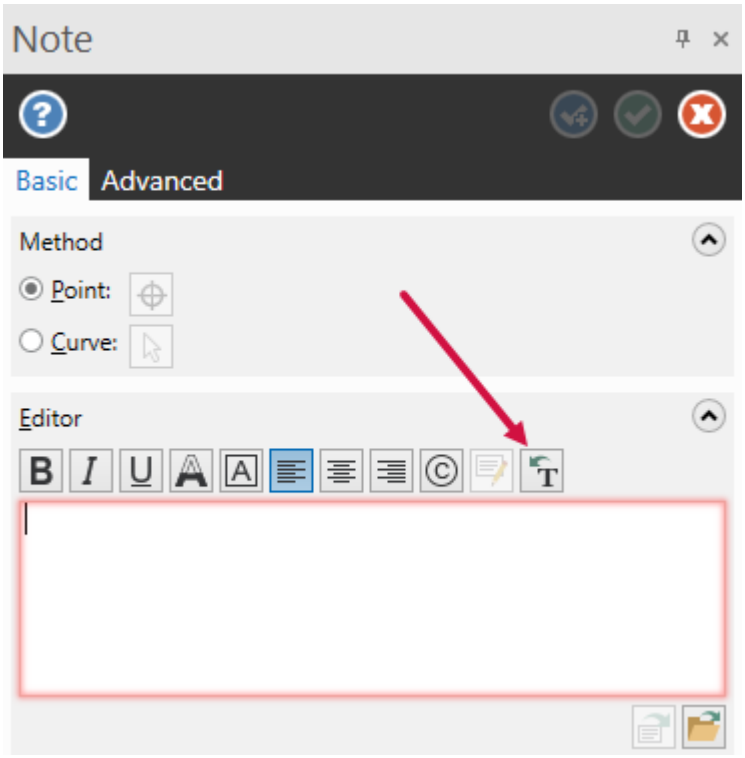
The **Drafting** function panel now includes easier, in-panel methods to modify your font format and placement of the drafting entity. In previous releases, you had to select a button to open a separate dialog box to make your modifications. This new workflow removes extra steps you took to make modifications.





Remembering Previous Note Text

The **Note** function now includes the new **Restore text** option, which populates the text box with the most recently entered text. This new option only restores content entered during the current Mastercam session.



Improving Visibility for Drafting Dimensions

Mastercam 2023 includes a new single-stroke font with a more pronounced period. Since the introduction of the now default **OLF SimpleSansCJK OC** TrueType font, the period in dimensions has been hard to distinguish and potentially causing confusion. An font with a more pronounced period is now included with Mastercam 2023. This new font is called **OLF SimpleSans CJKP OC**. The **P** in the name indicates the enhanced period.

OLF SimpleSansCJK OC	OLF SimpleSans CJKP OC
38.19	38.19

Including Dimensions When Aligning

The **Align Note** function now includes the alignment of dimensions. Previously, **Align Note** only affected notes.

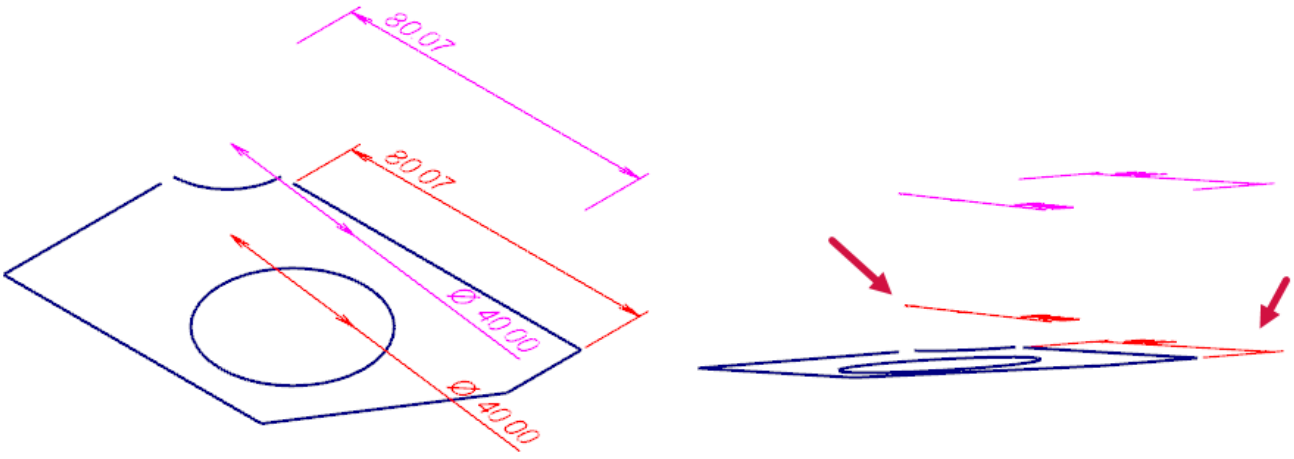
Transform Enhancements

Listed below are enhancements for the functions located on the **Transform** tab.

Projecting Dimension Entities

You can now project dimension entities using the **Project** function to a new Z depth. In previous releases, you could use **Translate** or **Dynamic** to move the dimensions. However, if your file contained multiple dimensions at different depths, you needed to repeat the process multiple times. **Project** allows you to select all dimensions and project them to a single depth with one operation.

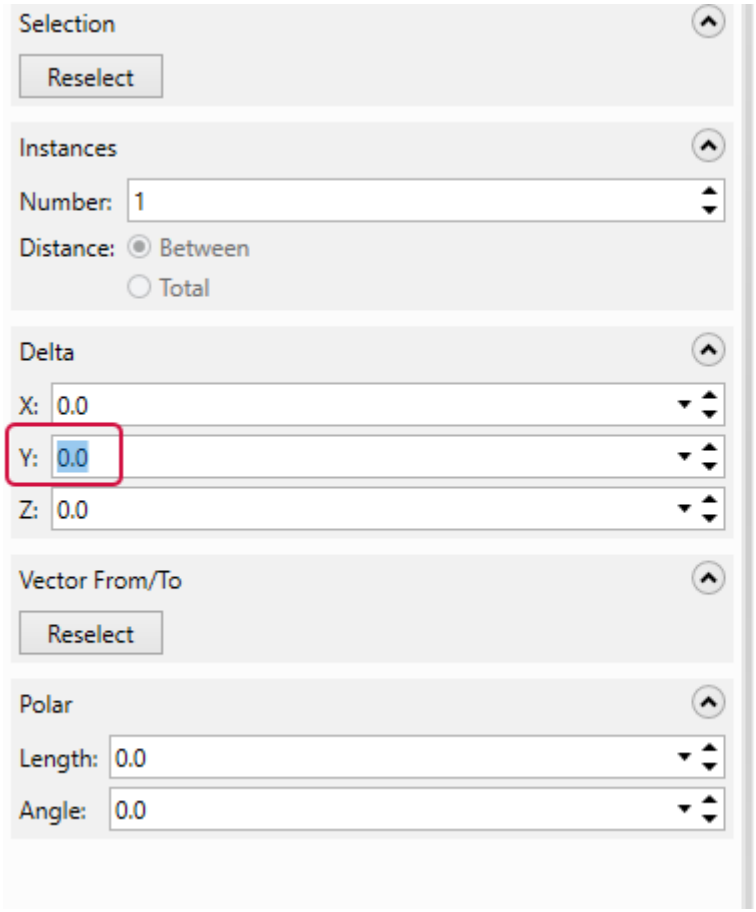
The image below depicts two dimension entities, which were located at different depths, projected to the same **Depth**.

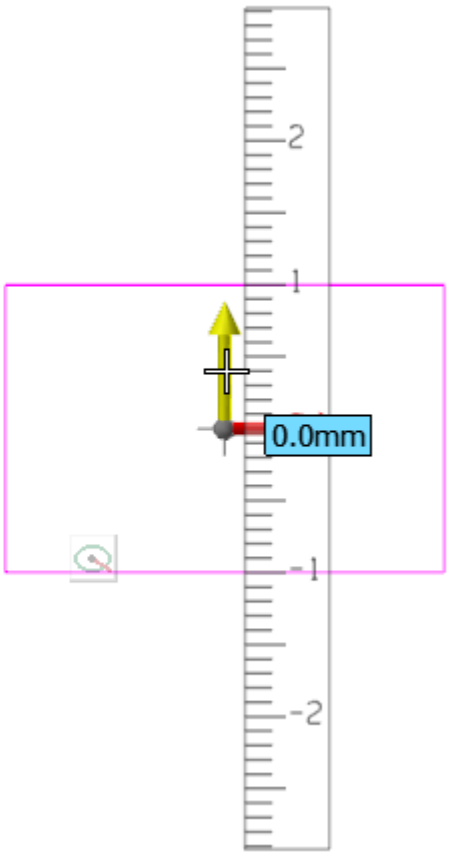


When **Project To** is set to **Depth**, the dimensions are projected to the entered value. If set to **Plane** or **Surface/Solid**, the dimensions do not move from their current depth.

Easily Enter X, Y, and Z Values While Manipulating Geometry

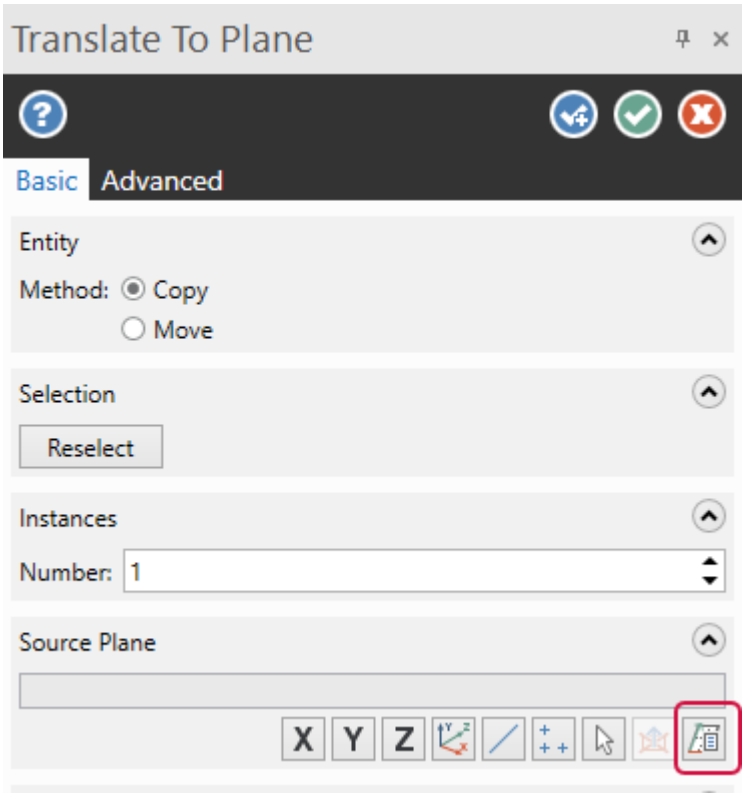
You can now easily enter values when using **Translate** or **Stretch** on geometry. When you hover over an axis of the Dynamic Gnomon, Mastercam automatically activates that field in the function panel. Once the field is active, you can then enter the X, Y, or Z value without having to click in the graphics window or in the function panel. This reduces the amount of clicks and mouse travel needed to use these functions.

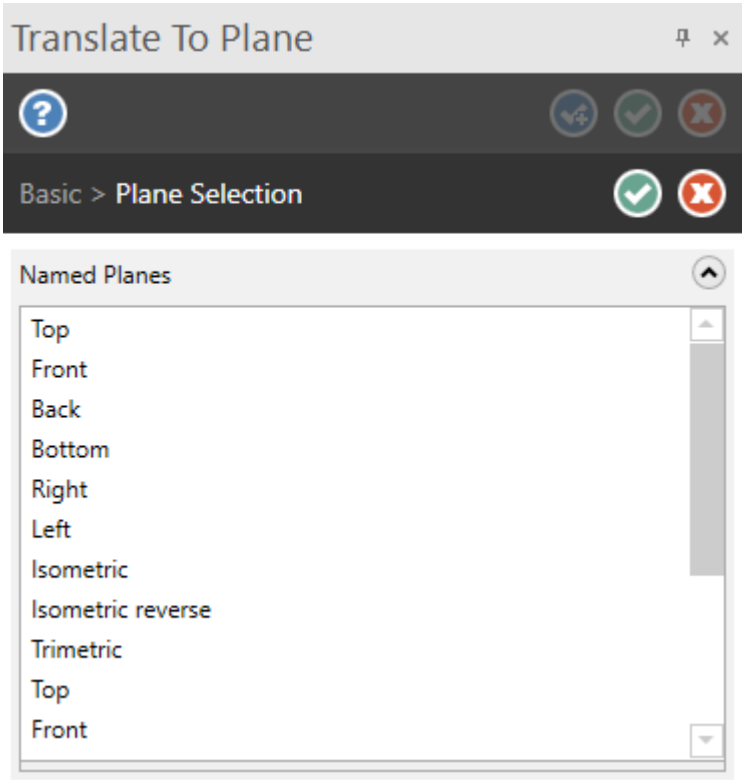




Improving the Plane Selection for Translating to Plane

Translate to Plane now uses the new **Plane Selection** fly-out panel when selecting a plane to move or copy entities. You no longer need to navigate through a separate dialog box.





Remembering the Last Dynamic Transform Operation

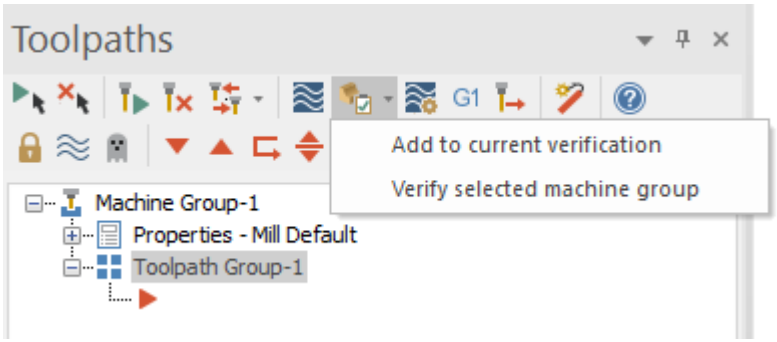
The **Only apply last transform** option is now located on the **Advanced** page of the **Dynamic** function panel. This option applies only the last transform operation when you make multiple transform edits in a session.

SIMULATION ENHANCEMENTS

Listed below are major enhancements to Mastercam Simulator.

Appending Operations to a Current Mastercam Simulator Session

Mastercam 2023 now includes the ability to add operations to the current verify session. When verifying, you can add more operations using the **Verify selected operations** drop-down in the Toolpaths Manager. From this drop-down, you can choose to **Add to the current verification** or **Verify the selected machine group**. The **Verify selected operations** button behavior is the same as previous releases.

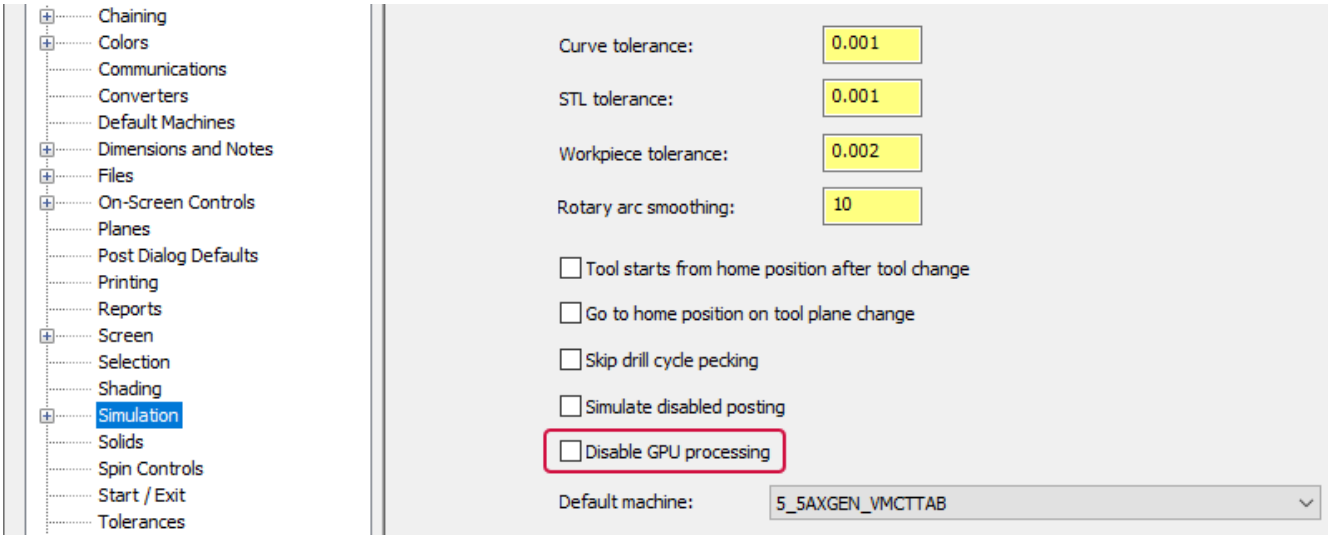


There are restrictions when adding an operation to a current session:

- Only operations positioned below the selected operations in the Toolpaths Manager can be added to the current session.
- Operations that depend on a previous operation cannot be added unless the parent operation is also selected.

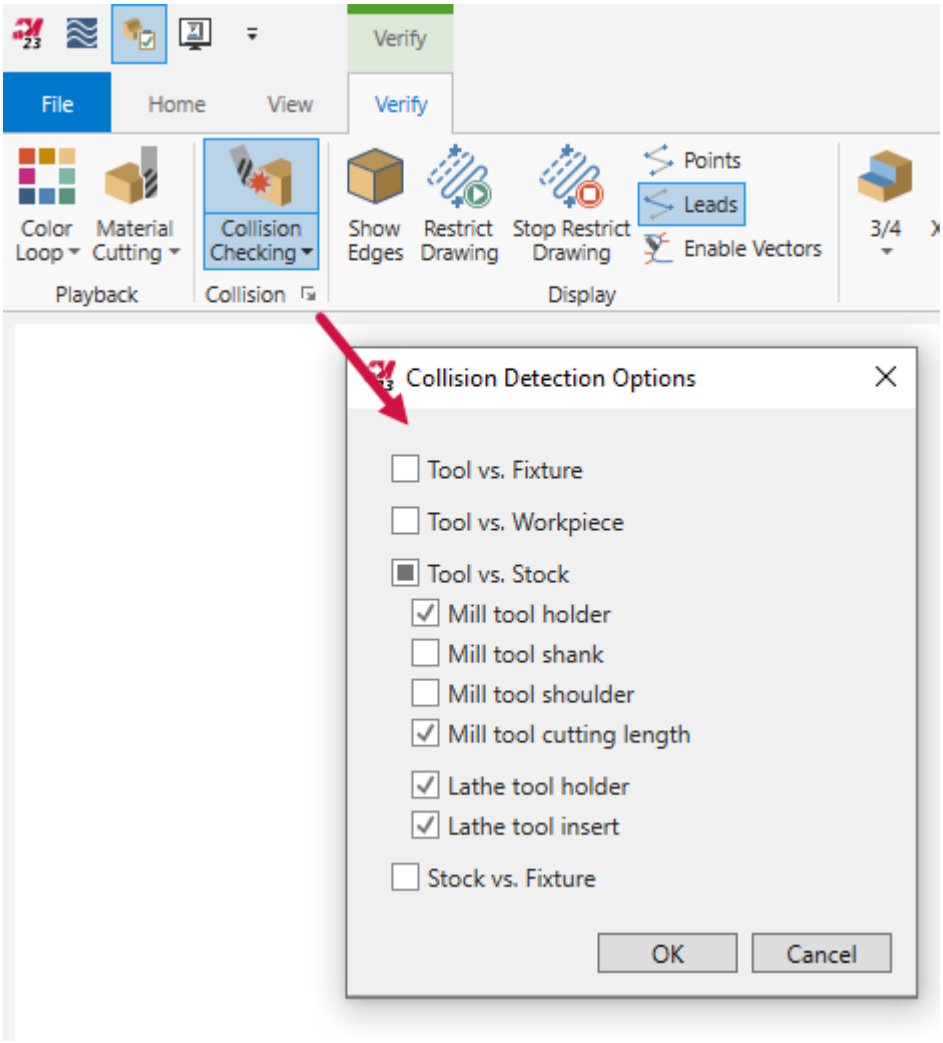
Improving the Overall Processing Speed

Mastercam 2023 includes overall processing speed and time to load data into Mastercam Simulator. These improvements only apply to milling toolpaths (2D, 3D, and multiaxis) and are particularly helpful for files that contain large toolpaths. If you are experiencing performance issues, you may wish to disable the processing using the **Disable GPU processing** option in the **Simulation** page of the **System Configuration** dialog box.

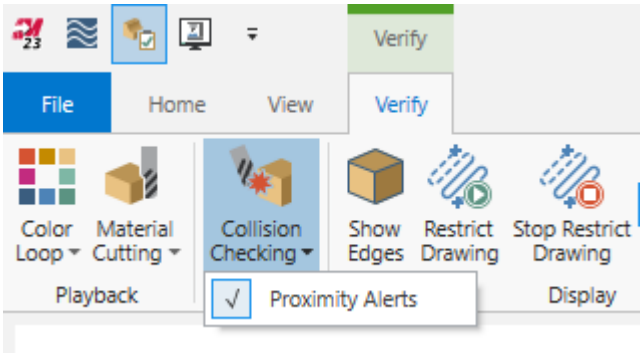


Setting Your Collision Checking Options

Mastercam Simulator includes improvements to collision checking while in **Verify** mode and **Simulation** mode. In previous releases, collision checking options were set in the **Options** dialog box. You could then toggle collision checking in the **Verify** or **Simulation** tab. In Mastercam 2023 the collision checking options have been removed from the **Options** dialog box and can now be accessed directly using the dialog box launcher on the tab.

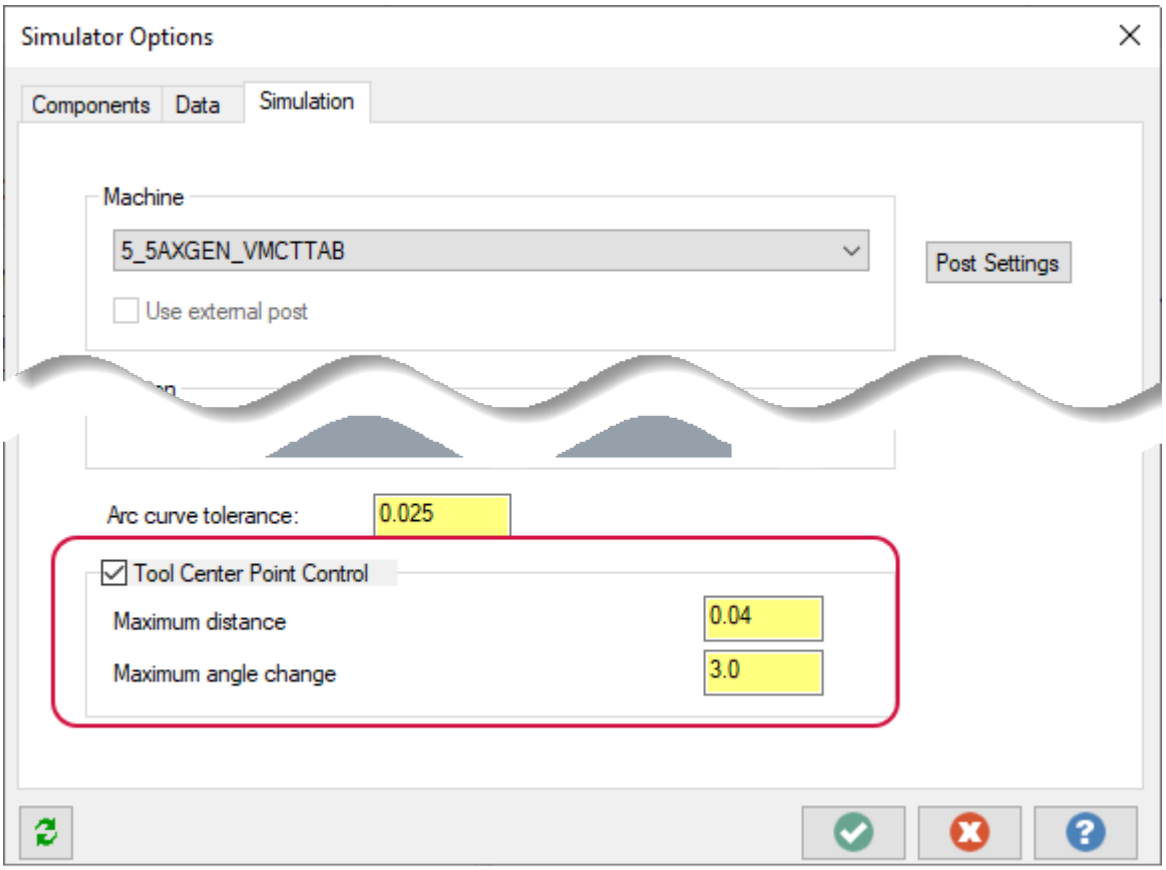


The **Collision Checking** drop-down lets you select the types of collisions to detect. Depending on your selected machine and if you are in **Verify** or **Simulation** mode, you may see different options.



Viewing Rotational Axis Jumps During Simulation

If you have large jumps in the rotational axes, you can select the new **Tool Center Point Control** options in the **Simulator Options** dialog box to define a linear interpolation around the tool tip or spindle head.



Select **Tool Center Point Control** to enable the options. Then set the following:

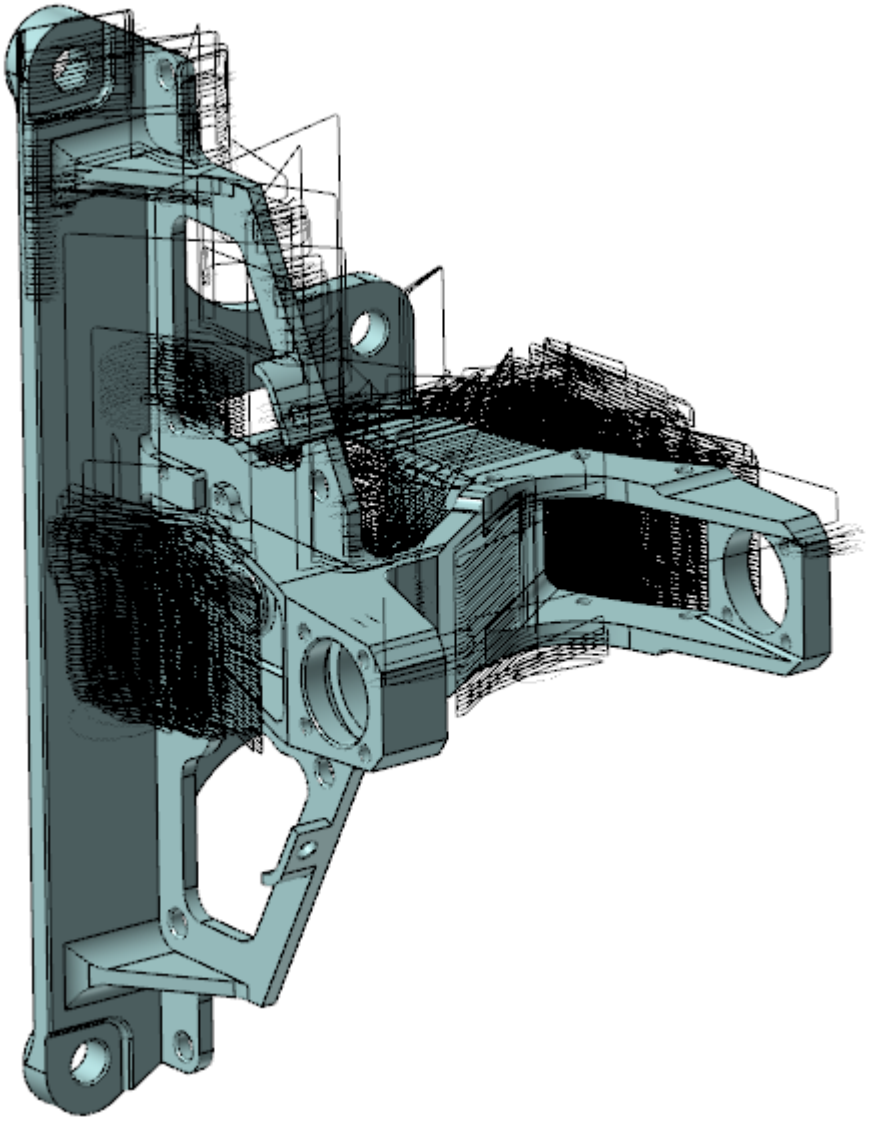
- **Maximum distance**: Sets the maximum length segment to create between interpolated points.
- **Maximum angle change**: Sets the maximum angle difference between subsequent positions.

TOOLPATH UTILITY ENHANCEMENTS

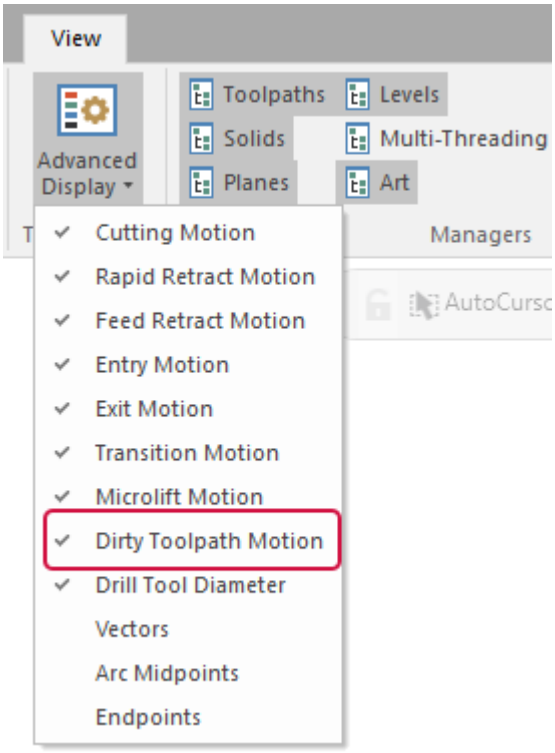
Listed below are major enhancements to toolpath utilities, including the Toolpaths Manager and stock model.

Displaying Dirty Toolpath Motion in the Graphics Window

Mastercam 2023 includes the new **Dirty Toolpath Motion** option in the **Advanced Display**. This option allows you to view the original toolpath motion even though the operation is dirty. In previous versions of Mastercam, once a toolpath went dirty, the toolpath motion would disappear from the graphics window.



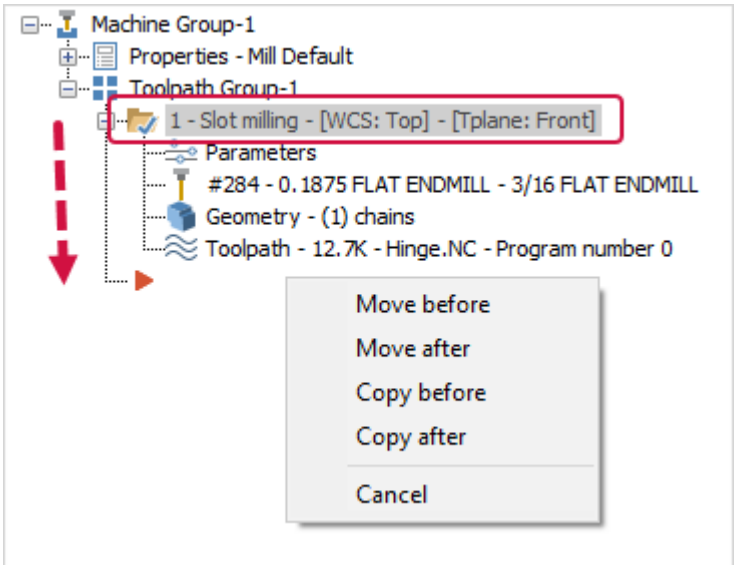
To view dirty toolpath motion, select **Dirty Toolpath Motion** in the **Advanced Display** drop-down.



As with all options in **Advanced Display**, you can edit the color and attributes of dirty toolpath motion. Use the **Advanced Toolpath Display** page of the **System Configuration** dialog box or select the dialog box launcher under **Advanced Display** in the **View** tab.

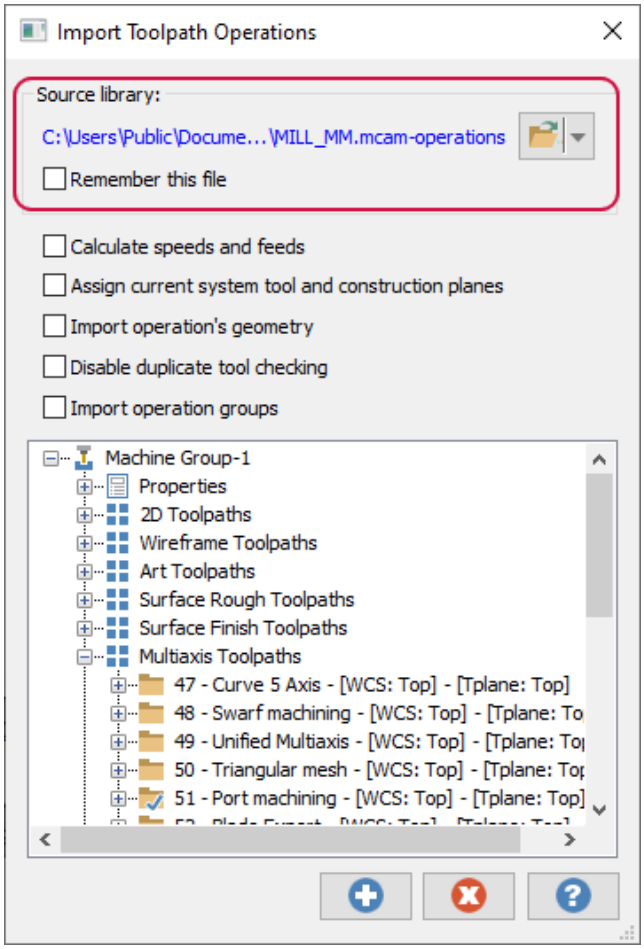
Copying and Moving Operations with the Left Mouse Button

Mastercam 2023 displays options during left-click drag-and-drop of operations in the Toolpaths Manager. Previously, left-clicking on an operation and dragging it to a location in the Toolpaths Manager moved the operation, but did not result in a prompt or alert you of the move. This enhancement gives you more control when moving or copying operations and prevents unintentional movement of operations within the manager.



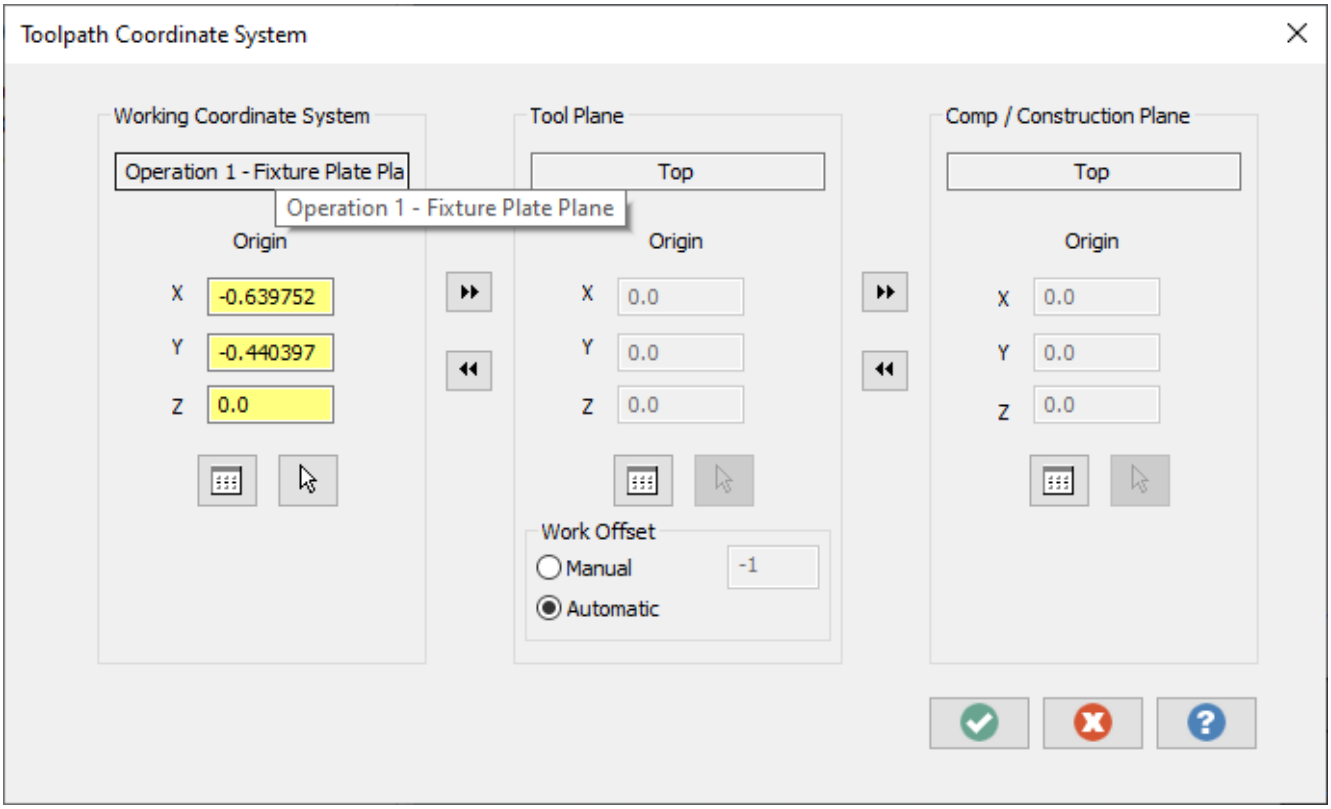
Importing Toolpath Operations from Recent Files

When importing toolpath operations into Mastercam 2023, the **Select Operations File** button includes a drop-down menu listing recently imported operations libraries. Improve your workflow by selecting an operations library from the list, instead of navigating with the file explorer.

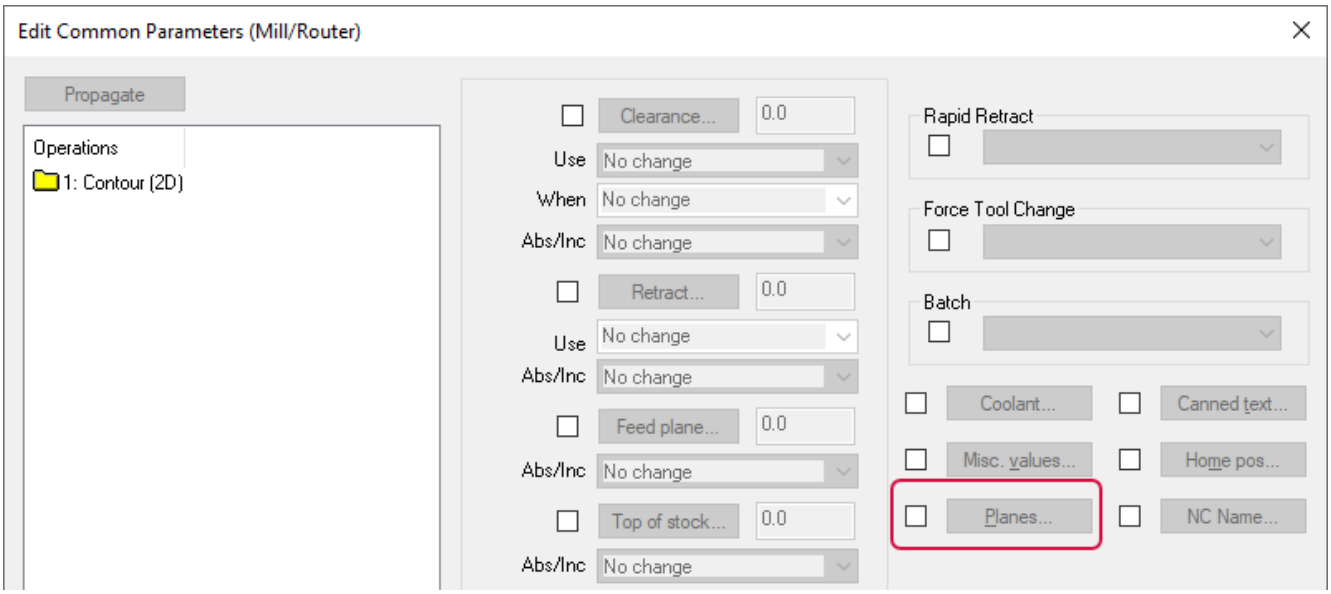


Viewing Plane Name Tooltips

Mastercam 2023 displays plane names as tooltips in the **Toolpath Coordinate System** dialog box, making it easier to select the correct plane when editing an operation. This change is consistent with the behavior of plane names in other areas of Mastercam.



You can access the **Toolpath Coordinate System** dialog box from the **Edit Common Parameters** right-click option in the Toolpaths Manager, and then select **Planes**.



SYSTEM ENHANCEMENTS

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

Dynamic Plane Enhancements

Listed below are enhancements when creating a dynamic plane.

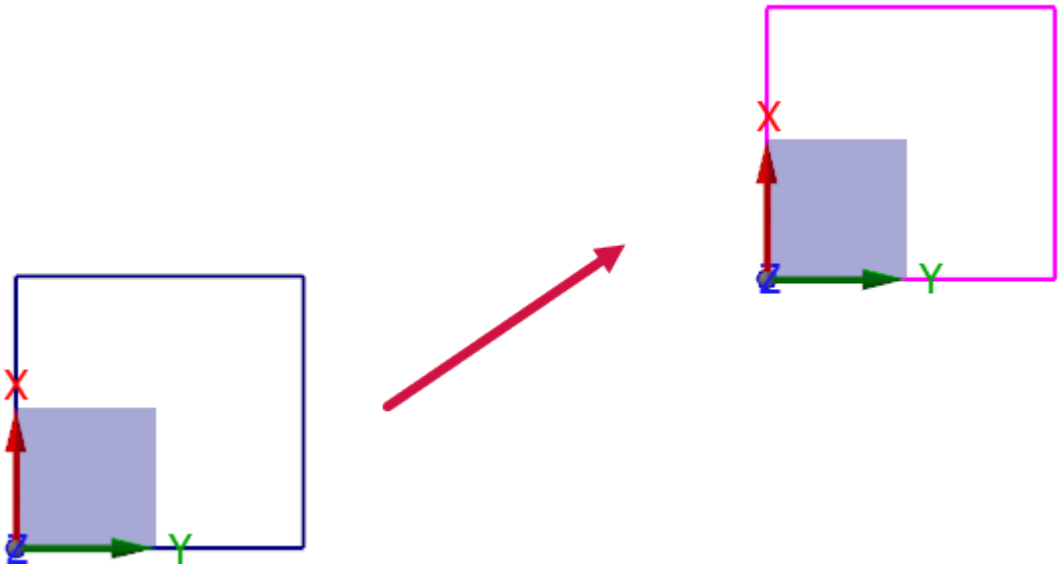
Creating Associative Dynamic Planes

Mastercam includes the ability to associate dynamic planes to geometry (including wireframe, solids, meshes, and surfaces) and to set the plane origin to any point on the geometry. Dynamic plane creation now consolidates the many ways of creating planes within Mastercam, making planes easier and faster to use. Previous versions of Mastercam only allowed planes created from entities to associate to geometry, and the plane origin was in the center.

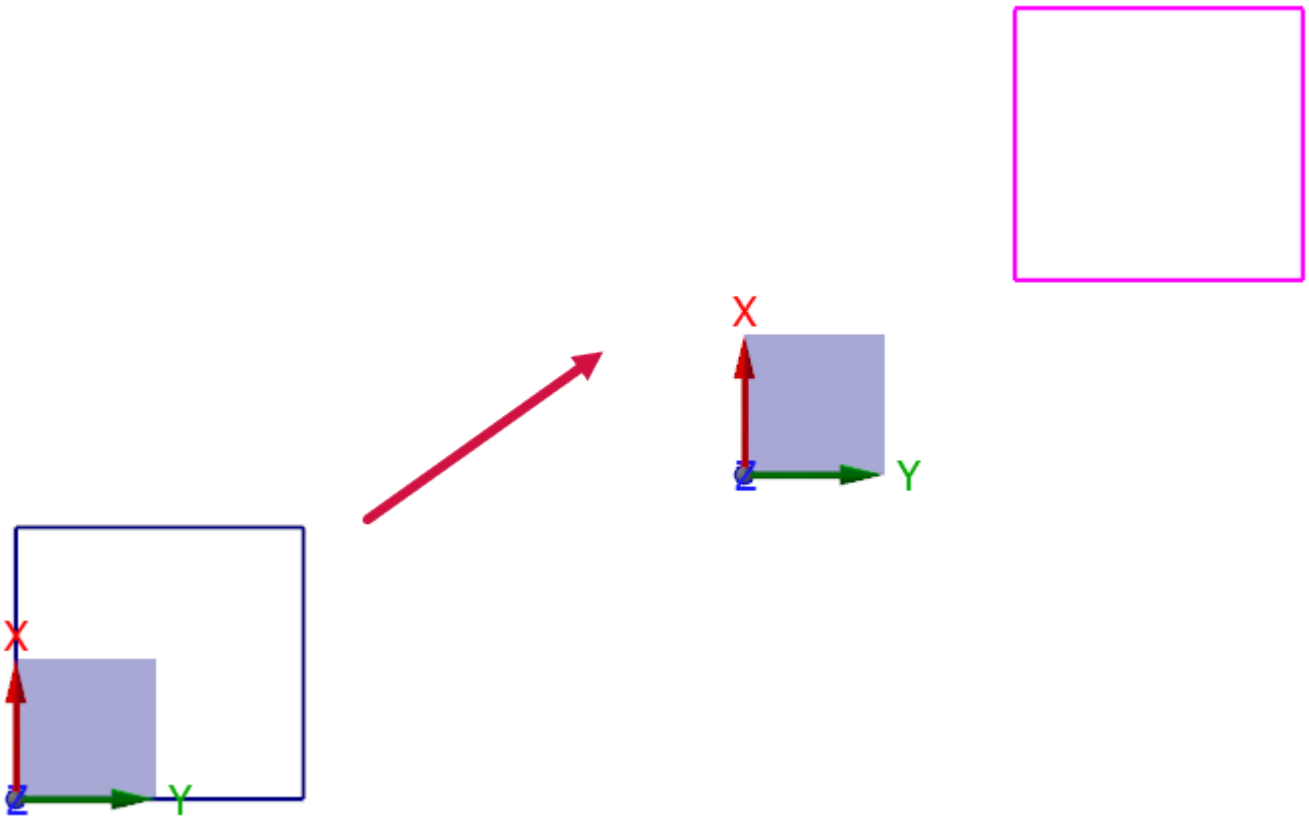
You can now set the dynamic plane origin to any point on the selected geometry and can associate a dynamic plane to geometry:

- Associate the plane **Orientation** to solids, meshes, and surfaces.
- Associate the plane **Origin** to wireframe, solids, meshes, and surfaces.

Transforming geometry with an associated plane:



Transforming geometry with an unassociated plane:

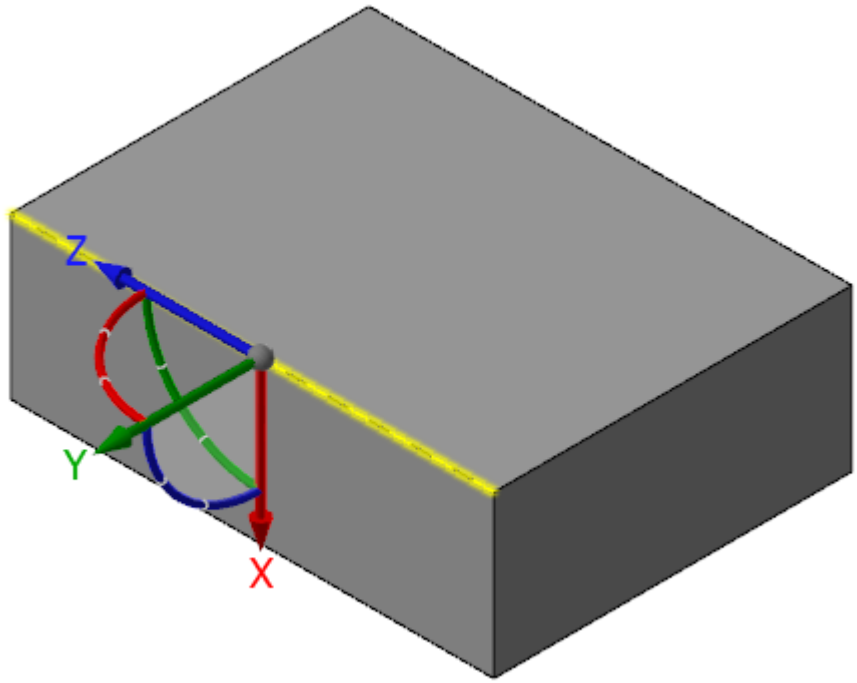


To create a dynamic plane, select **Dynamic** from the **Create a new plane** drop-down in the Planes Manager or click the Dynamic Gnomon in the lower left corner of the graphics window. By default, new planes automatically associate to the selected geometry. When editing an existing plane, Mastercam asks if you wish to associate the plane origin to geometry. You can change the default settings for planes associativity in the new **Planes** page of the **System Configuration** dialog box.

NOTE
At any point in your Mastercam session, you can remove the plane associativity by deselecting the checkboxes in the **Settings** group of the **Advanced** tab in **Edit/New Plane**.

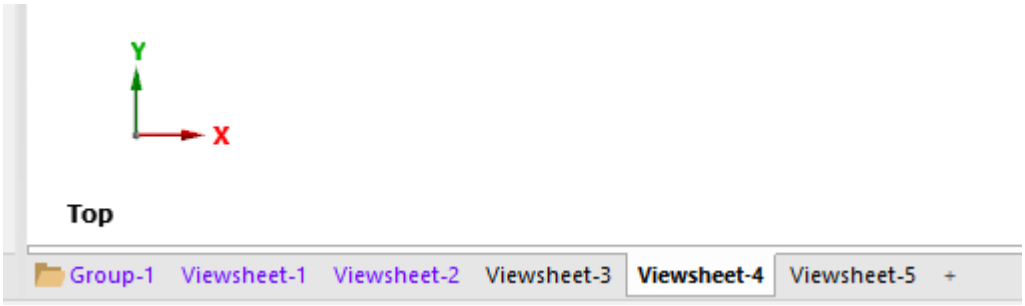
Creating Dynamic Planes with Normal Orientation

Mastercam 2023 includes normal support for the Dynamic Gnomon. During dynamic plane creation, moving your cursor along wireframe geometry results in the Z axis of the gnomon orienting to the normal of the geometry. When possible, Mastercam also orients the X axis of the gnomon along the tangent to the geometry. This feature helps you to orient your planes faster, resulting in less mouse movement and increased productivity.

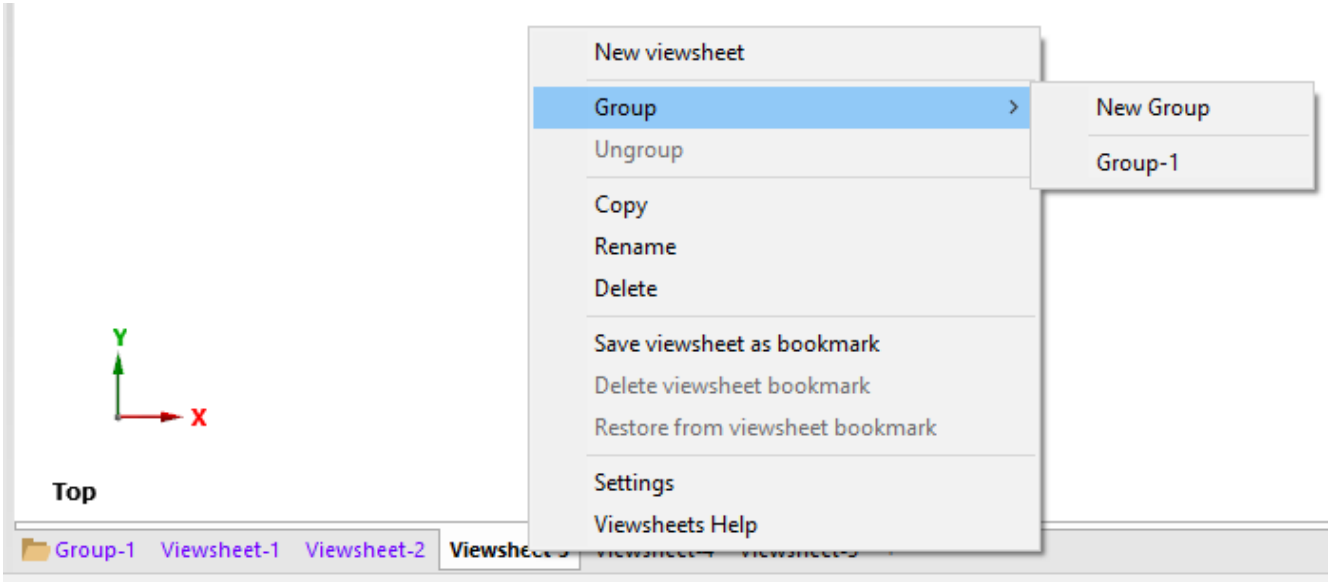


Grouping Viewsheets in Mastercam

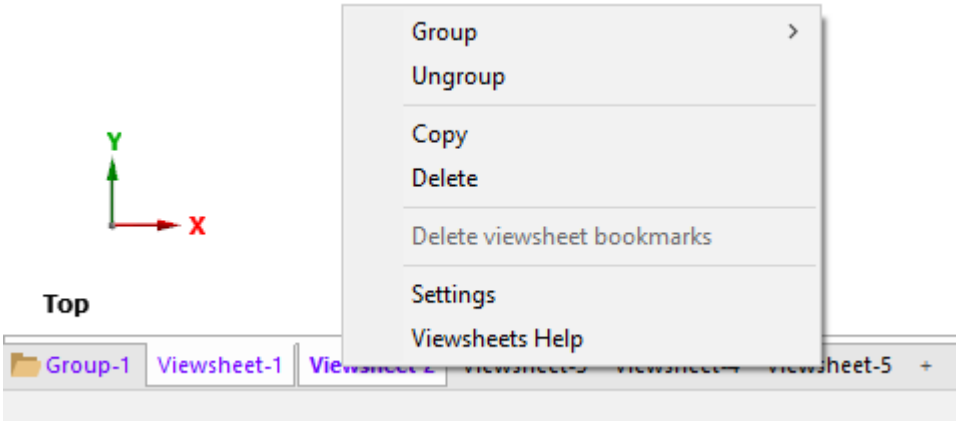
Mastercam 2023 gives you the ability to create viewsheet groups. Use viewsheet groups for better viewsheet organization, more efficient workflow, and reduced on-screen clutter.



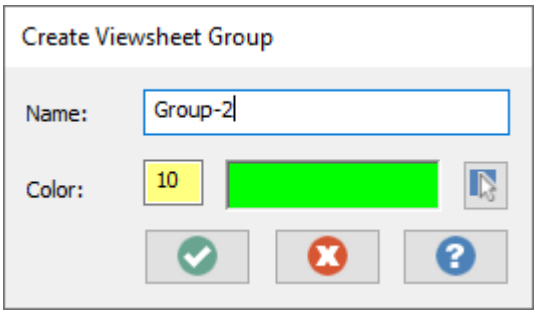
To add a viewsheet to a group, right-click on the viewsheet. Select **Group**, **New group** to create a new group or choose an existing group. To remove a viewsheet from a group, right-click the viewsheet and click **Ungroup**.



You can also select multiple viewsheets to group or ungroup by pressing [**Shift**] and then selecting the viewsheets, or using the right-click menu.



When you create a new group, Mastercam prompts you to customize the group name and color. You can edit these at any time by right-clicking on the group tab and selecting **Rename** or **Color**.



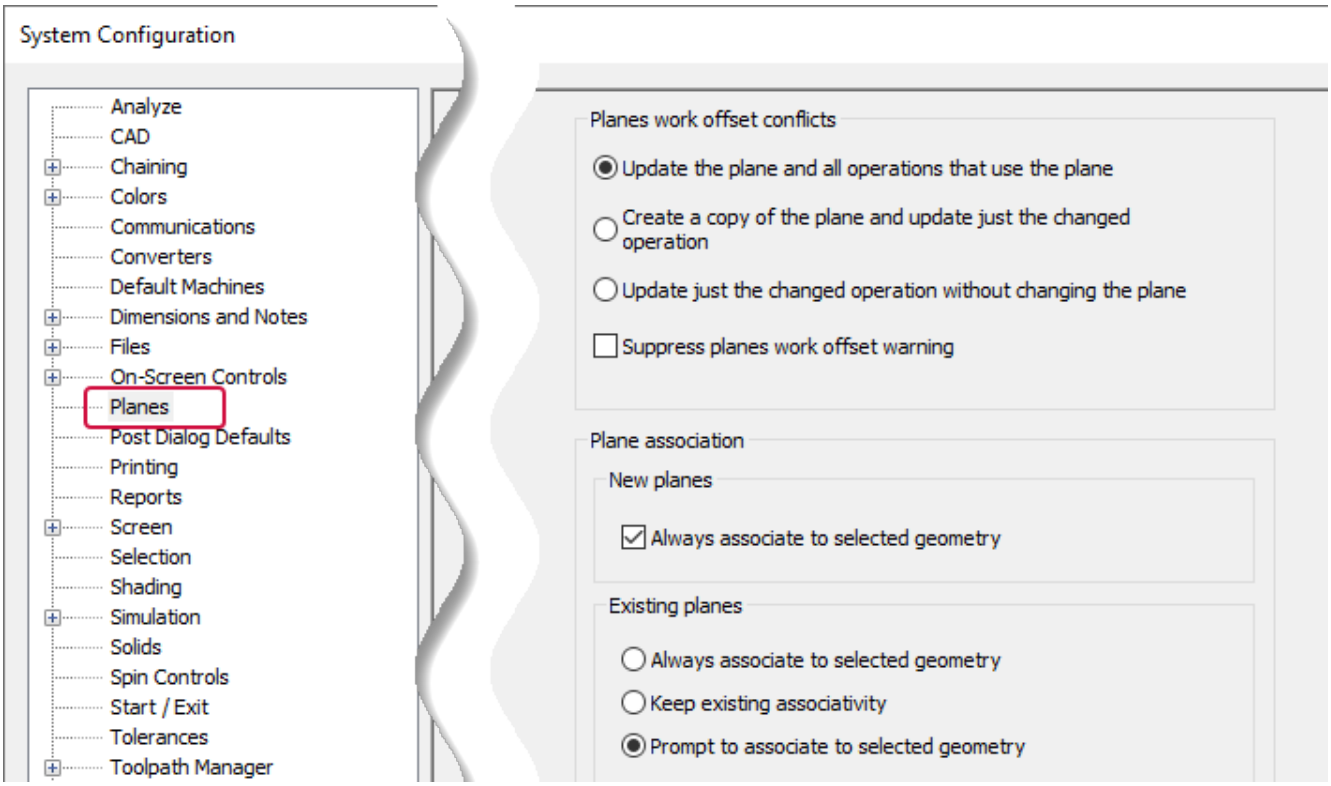
Another way to add a viewsheet to an existing group is by dragging-and-dropping it between the group tab and any group member. Similarly, you can remove a tab from a group by dragging it outside of the grouped viewsheets. To expand or collapse a group, click on the group tab. The folder icon displays as open or closed depending on the group status.

Configuration Enhancements

Listed below are enhancements made to the **System Configuration** dialog box.

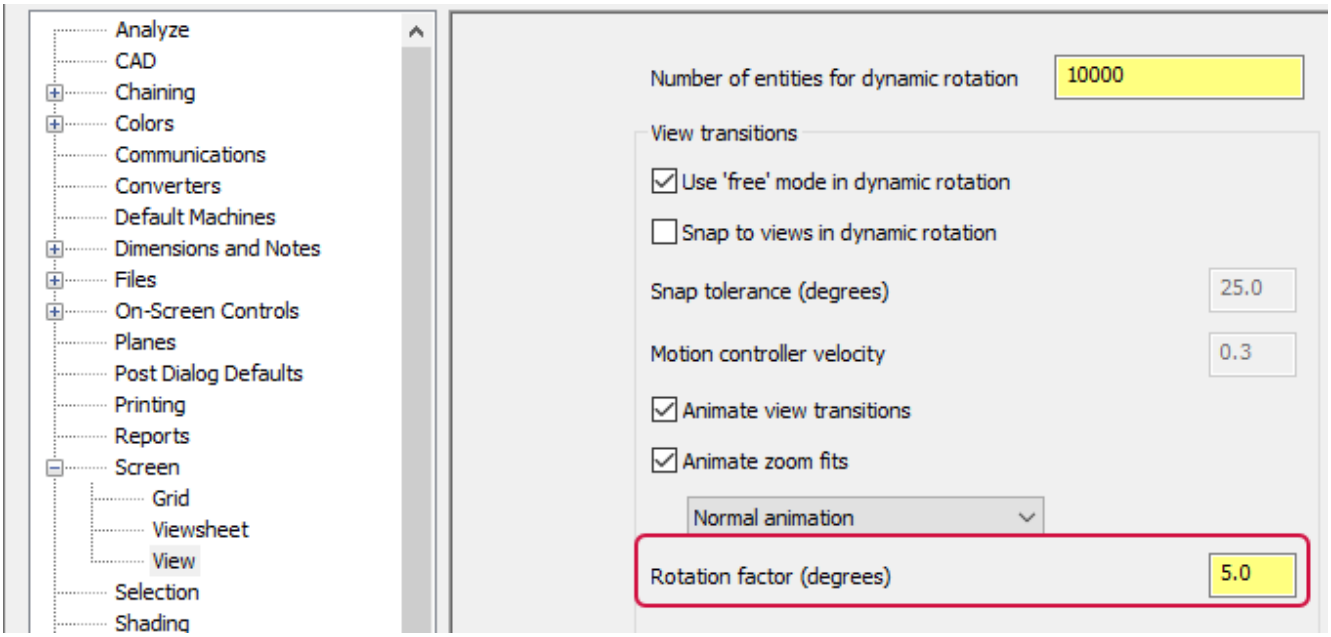
Configuring Your Planes Settings

The new **Planes** page allows you to customize the association between new planes and selected geometry using the new **Plane association** options. Use these new options to set the default associativity behavior for planes and selected geometry. This new page also includes the **Planes work offset conflicts** options previously located on the **Toolpaths** page.



Adjusting the Degree of Rotation for the Graphics View

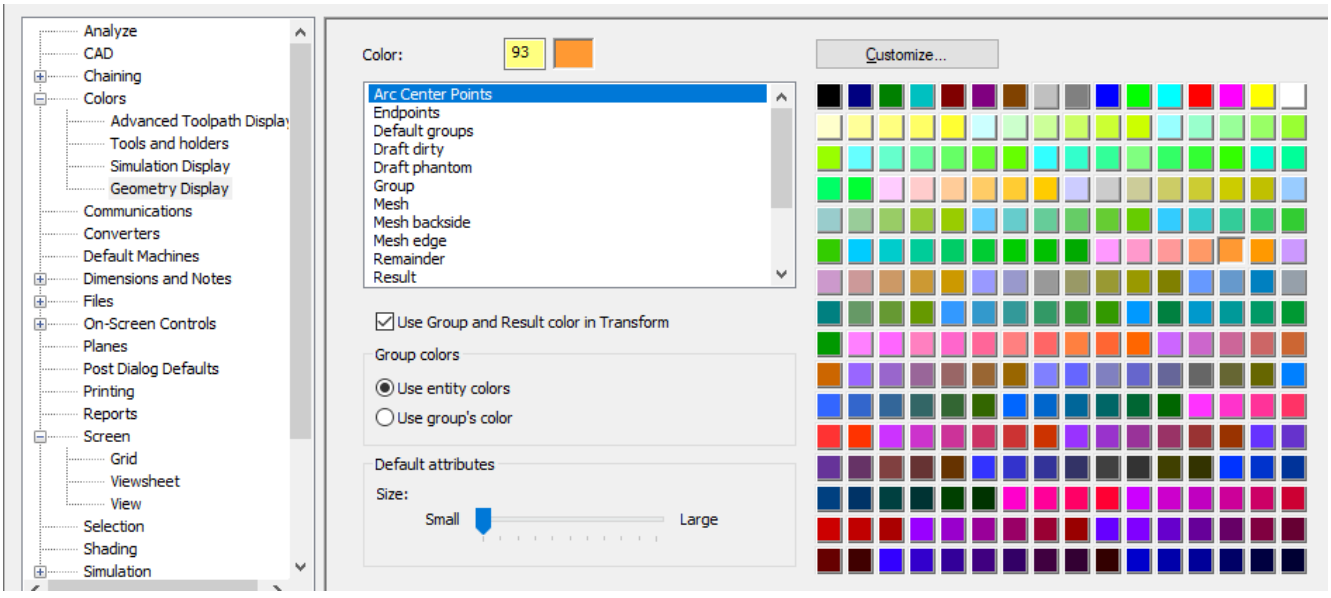
Mastercam 2023 allows you to set the number of degrees Mastercam rotates the graphics view when you hold [**Alt**] and press an arrow key. This feature provides more control over how you view your part.



Set the degree of rotation by entering a value between **0** and **360** into the **Rotation factor (degrees)** option in the **View** page. You can use this value for a single Mastercam session, or save your changes to the configuration file for future use.

Customizing the Geometry Display

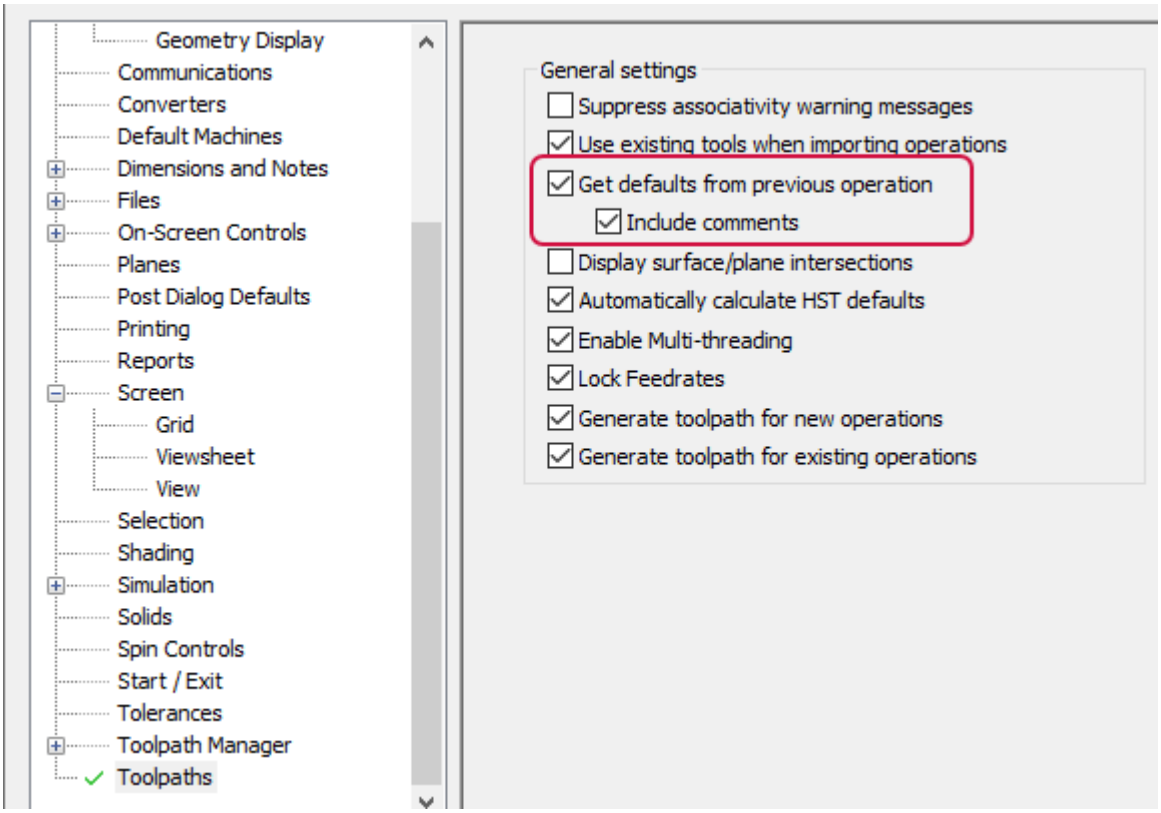
Exercise greater control over geometry appearance in the graphics window using the new **Geometry Display** page. This page condenses geometry-related options from the **Colors** and **Advanced Toolpath Display** pages, making it easier for you to customize your geometry from a single location. You can also edit the default size of **Arc Center Points** and **Endpoints**, making it easier to view them in the graphics window.



You can also access this page from the **Display** group of the **Home** tab.

Including Comments with Defaults from a Previous Operation

Save time and create more consistent operation comments by using the new **Include comments** option. This option, in the **Toolpaths** page, allows you to carry over comments in the **Tool** page of an operation's parameters to subsequent new operations of the same type. **Include comments** is available when you select **Get defaults from previous operations**.



Selecting a Language for the Mastercam Installation

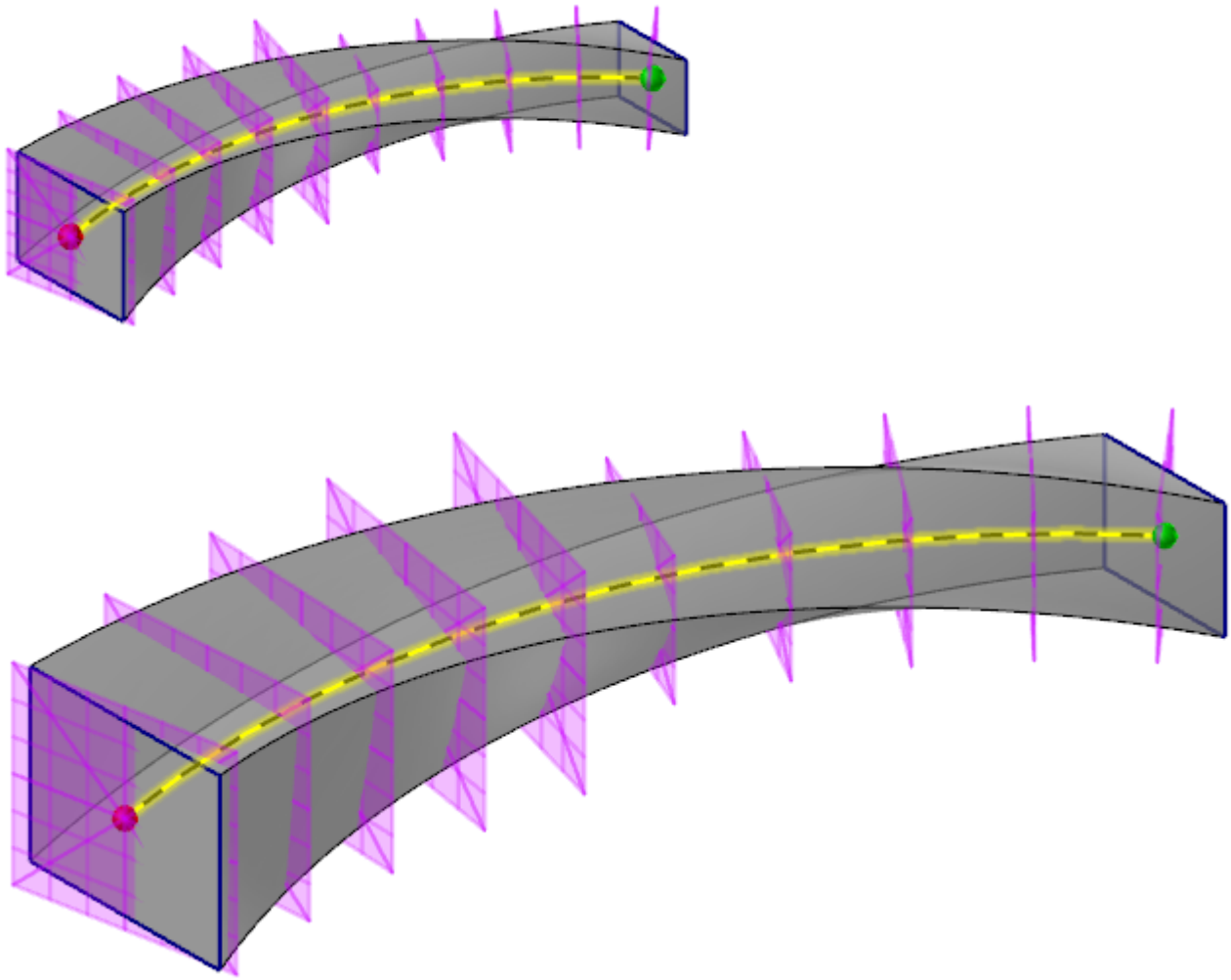
The Mastercam 2023 installation process features language packs for installing Mastercam in a non-English language. Language packs are files that you download separately from the Mastercam installer and select during the installation process. If you wish to install Mastercam 2023 in English, you do not need to download an additional language pack. Language packs enhance the language selection process while minimizing the download size.

Before installing Mastercam, download the desired language pack and place it in a secure location on your computer. Contact your Reseller if you need assistance acquiring a language pack. Follow the instructions in the Mastercam installer to complete the installation.

Mastercam can only be installed in one language at a time. If you want to run Mastercam in a different language, run the installer again and select **Modify**. The installer will take you through the installation process again and you can select a different language pack.

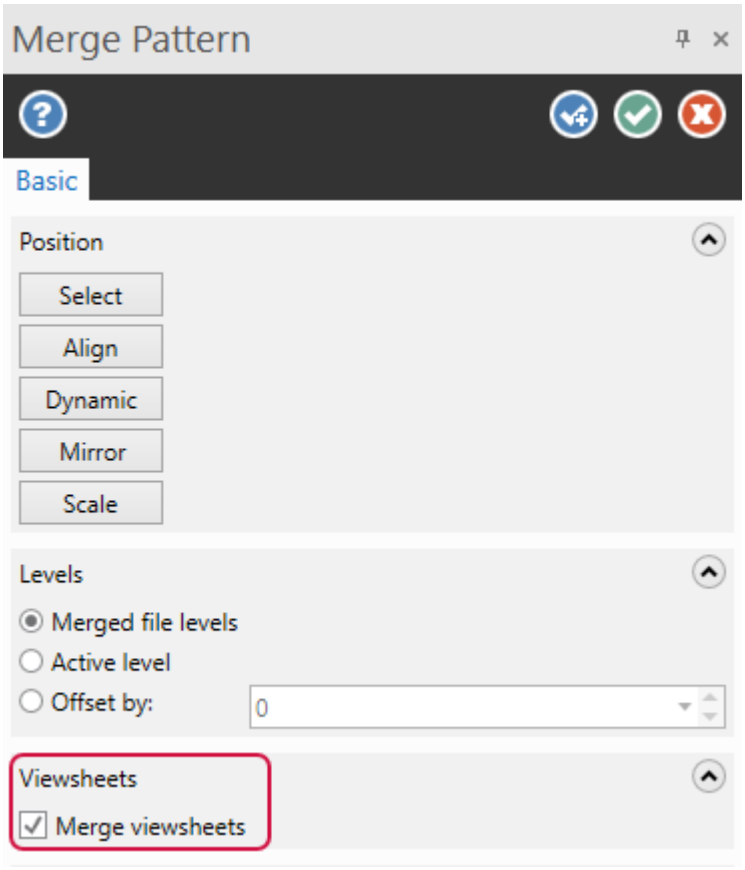
Auto Scaling of Temporary Planes During Functions

Mastercam 2023 automatically scales the size of temporary planes during functions, such as **Curve Slice Along Curve**. Zoom in or out in the graphics window and the planes scale based on the size of the part geometry. This feature makes it easier to view temporary planes while you are working within functions that use them.



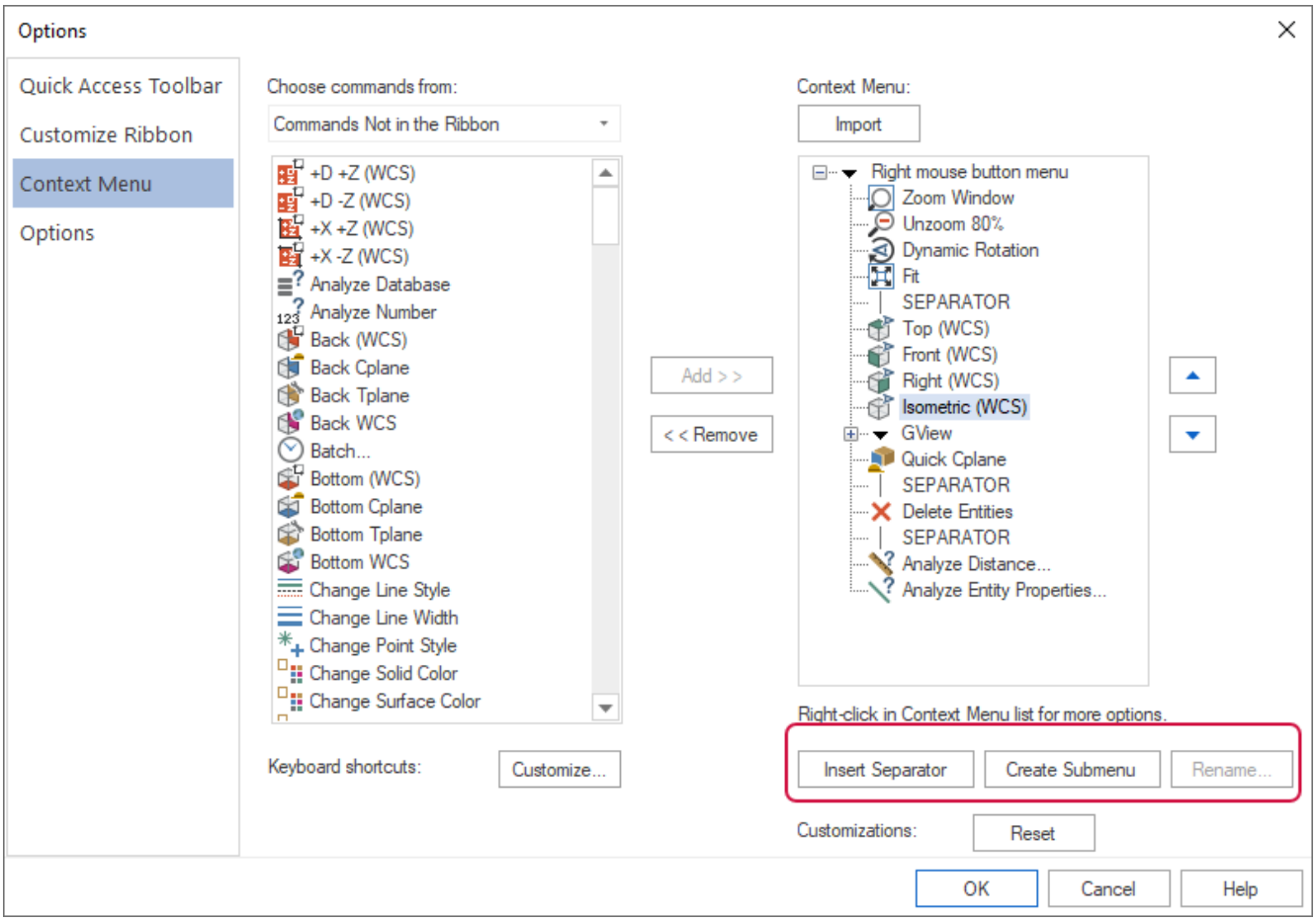
Importing Viewsheets for Merged Files

When merging part files using the **Merge Pattern** function, you can merge incoming viewsheets into the target part file using the **Merge viewsheets** option. If there are duplicate viewsheet names, Mastercam appends a number to the viewsheet name.



Customizing the Right-Click Menu

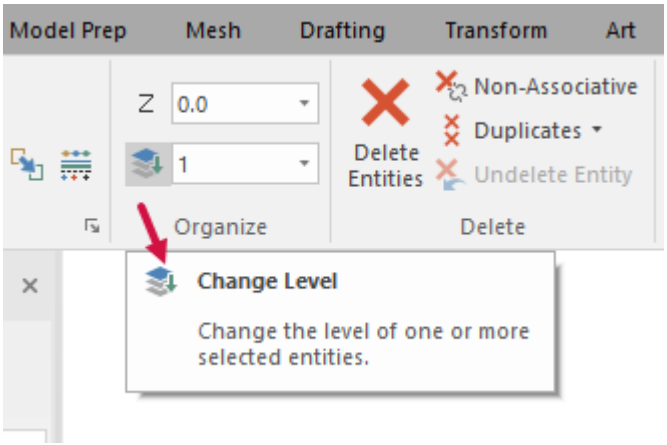
Mastercam 2023 has improved the customization of the context (right-click) menu. The options now align with the options available for customizing the Mastercam ribbon including the ability to reset a customized right-click menu to factory defaults. Common options in the right-click menu of **Context Menu** list box can now be accessed from the dialog box. These enhancements lead to a more consistent Mastercam experience, and make it easier to customize the interface to meet your needs.



To customize the context menu, select **File**, **Options** and click the **Context Menu** page.

Displaying Tooltips Longer

Ribbon tooltips in Mastercam 2023 display five times longer than in previous versions of Mastercam. This is particularly useful for reading long tooltips and is consistent with the display time of tooltips in other areas of Mastercam.

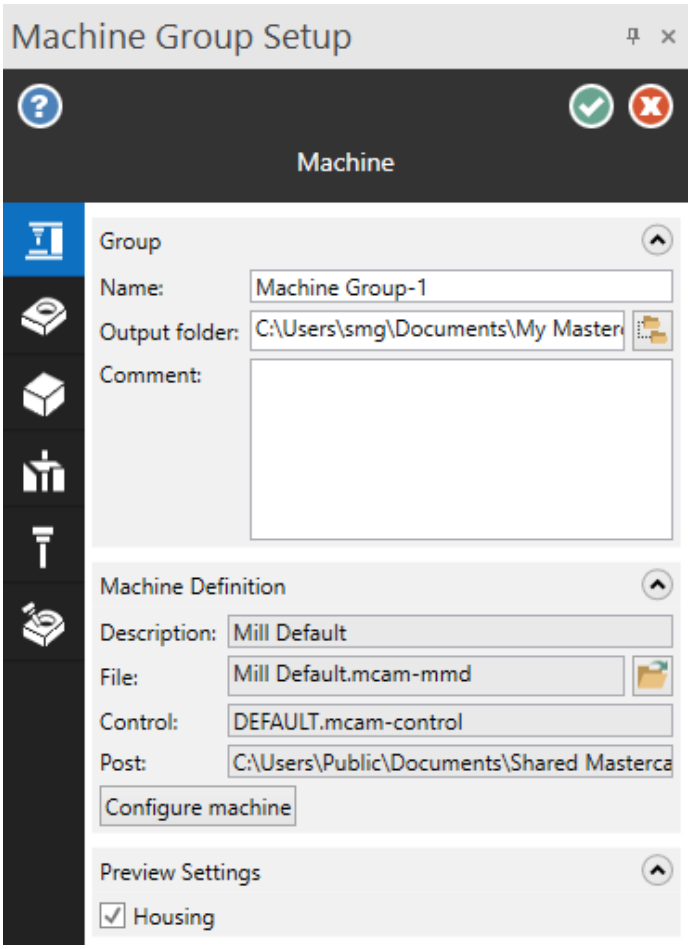


MACHINE SYSTEMS ENHANCEMENTS

Listed below are enhancements made to Mastercam's machine systems.

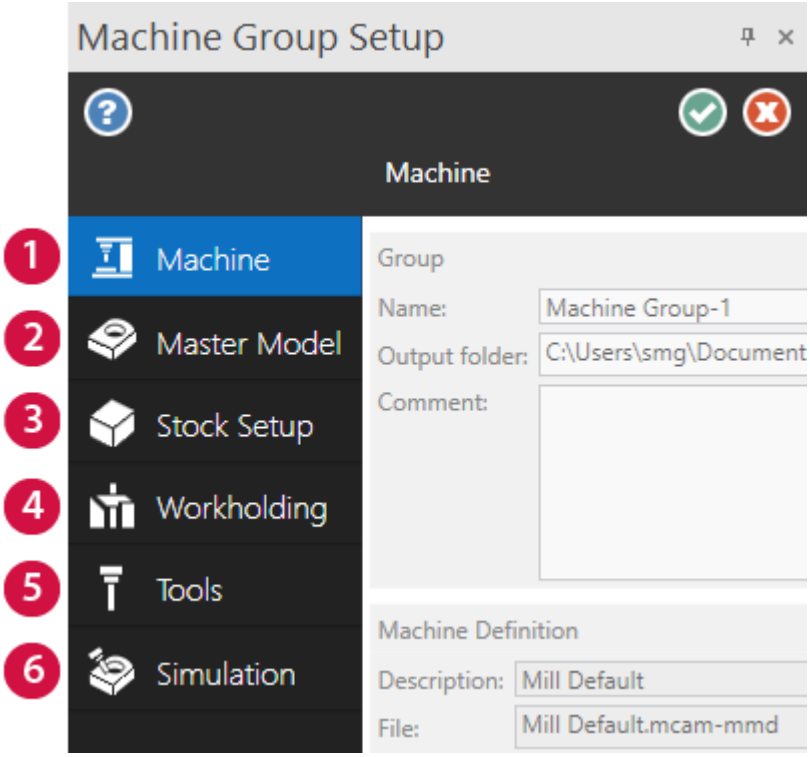
New Workflow and Interface for Managing Your Machine Groups

Mastercam 2023 introduces the new **Machine Group Setup** function panel for Mill and Router machine groups. This new function panel encompasses features and functionality from the **Machine Group Properties** dialog box, including new features. These changes continue Mastercam's effort towards kinematic awareness.



The previous locations where geometry and job setup information, such as stock, fixtures, and machine definitions were set have been consolidated into one location, making it easier for you to work with your part. The **Machine Group Setup** function panel opens when you select **Files**, **Tool settings**, or **Stock setup** under **Properties** for a Mill or Router machine in the Toolpaths Manager.

This new interface includes everything you need to set up your part, including parameters for Mastercam Simulator. The function panel has seven pages, providing a top-down structured workflow from setting your machine definition to choosing the machine that is displayed during simulation.



1. **Machine**—Specifies the names and data paths of files needed by the operations in the machine group. This includes the machine definition and operation defaults.
2. **Master Model**—Defines the final part (called the master model) after all machining is complete. This page also includes engineering information, such as the material.

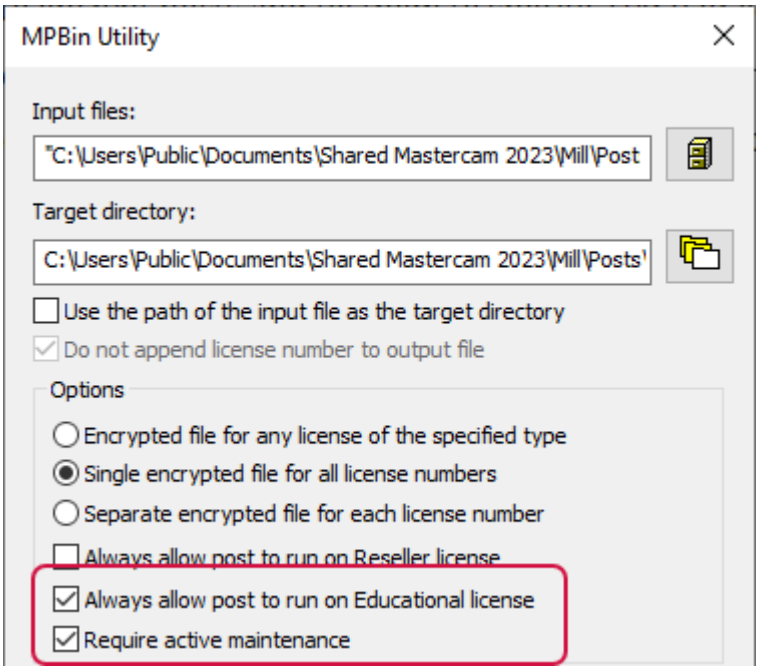
   NOTE
   You do not need to define a master model. However, defining a master model provides workpiece data for Mastercam Simulator, ensuring that you always have a workpiece defined.

3. **Stock Setup**—Defines the stock model. Stock models help you visualize your toolpaths more realistically in the graphics window. The defined stock model can also be displayed with the part geometry when viewing the file or toolpaths, or during Mastercam Simulator.
4. **Workholding**—Allows you to select fixtures and other workholding components with your master model and stock. These components then translate into Mastercam Simulator as **Fixtures**, which you can use for collision checking.
5. **Tools**—Controls how Mastercam assigns tool numbers, tool offset numbers, and default values for feeds, speeds, coolant, and other toolpath parameters.
6. **Simulation**—Allows you to modify the machine that is saved with your Mastercam machine definition (*.mcam-mmd) for use with Mastercam Simulator.

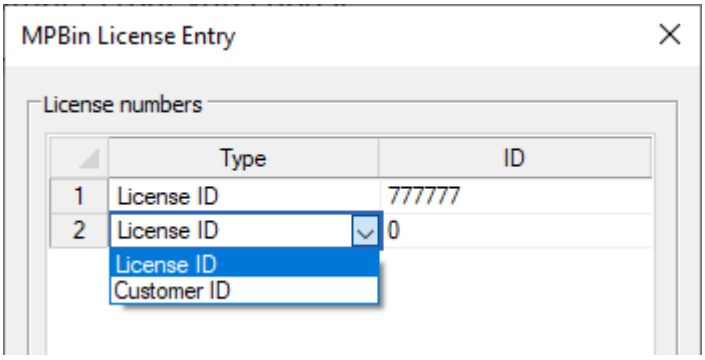
Encrypting Your Posts

The **MPBin Utility**, used to encrypt posts and link them to specific HASP or license numbers, has been expanded for Mastercam 2023.

- You can encrypt a post so that it will run on any Educational license, in addition to specific license numbers that you choose.
- You can now require that users have active Maintenance to run a post. You can add this requirement to any other encryption option.



The **License Entry** dialog box has also been enhanced so you can enter either license numbers (HASP, NetHASP, or software license numbers) or customer IDs (unique to each customer). Using customer IDs lets you support users whose license might change; in previous versions of Mastercam, you would have had to use MPBin for each individual post for each different license.



As in past versions, you can store lists of IDs in text files so that you can reuse them. The format of the text file has changed slightly to accommodate the different license types. Each line in the file should have two comma-delimited values, where the first value is **0** for a license number or **1** for a customer ID. For example:

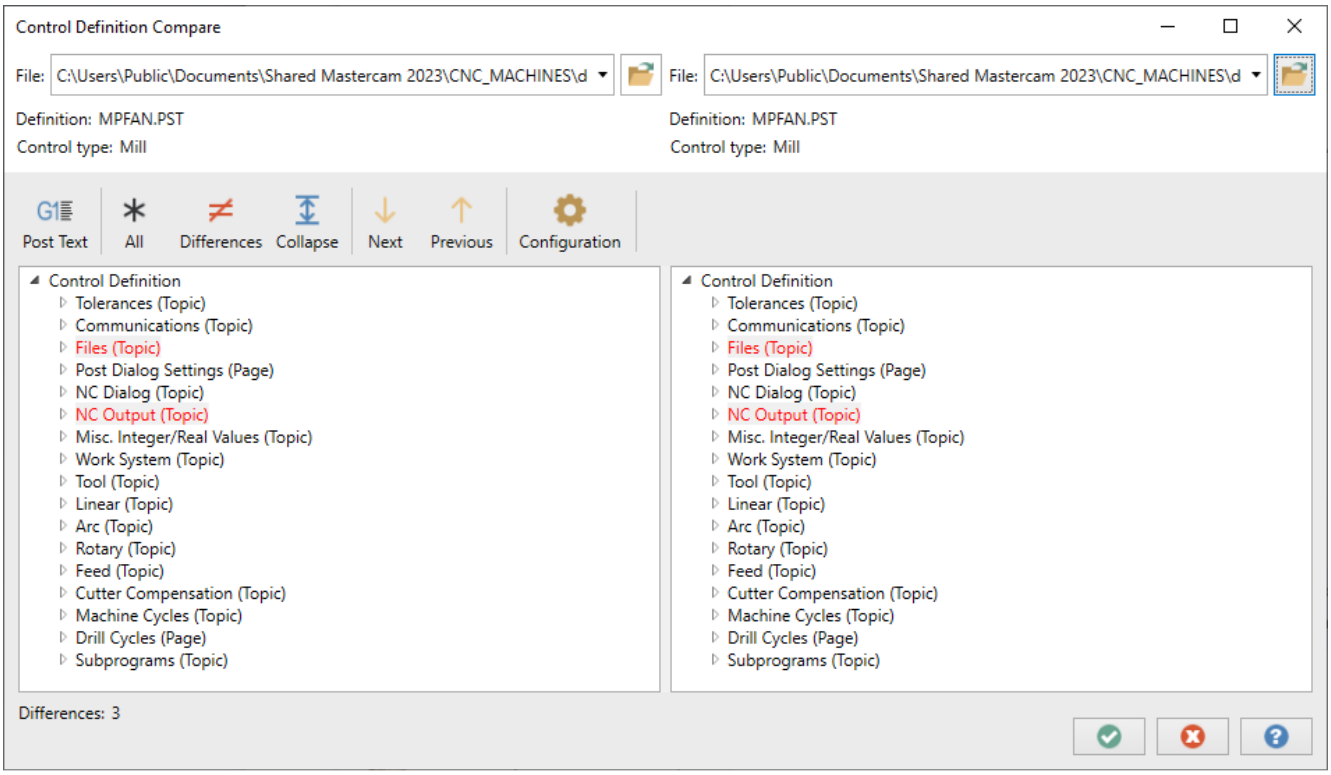
- 0, 12345
- 1, 67890

Existing text files can be used without modification. If the line contains a single value, it will be processed as a license number.

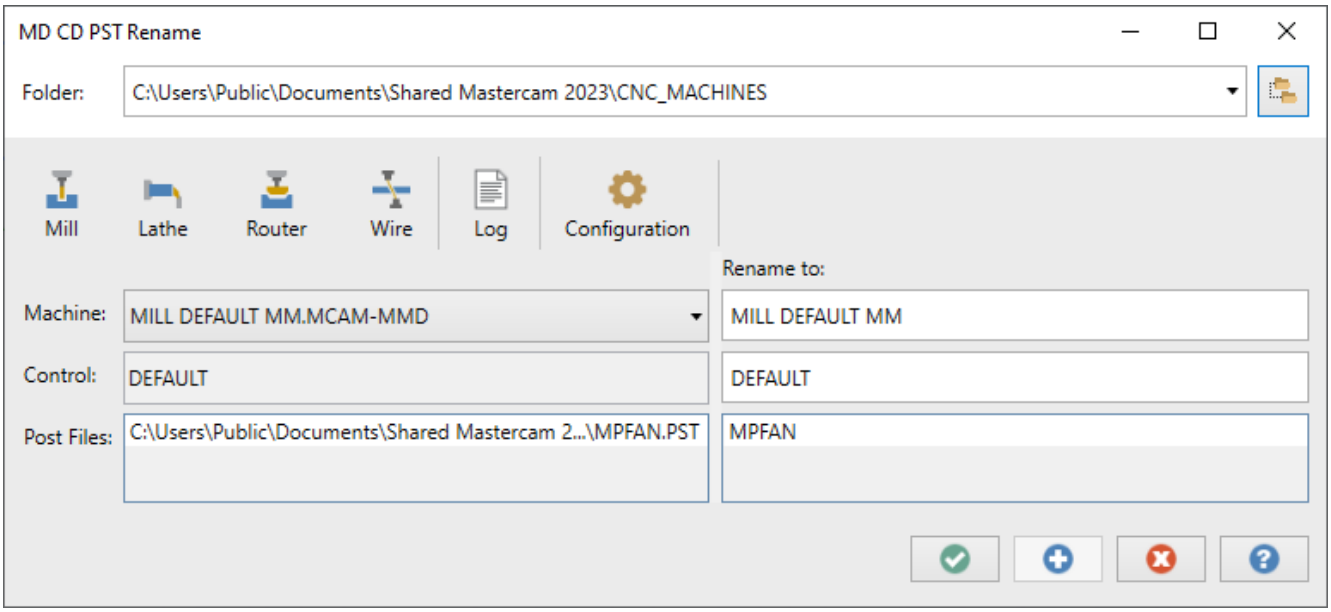
Improving the Experience of CD Compare and MD CD PST Rename

Mastercam's **CD Compare** and **MD CD PST Rename** add-ins have been redesigned for Mastercam 2023 with fresh, streamlined interfaces.

The **CD Compare** add-in lets users quickly compare control definition and post text settings, even when the original files are from different versions of Mastercam:



The **MD CD PST Rename** add-in lets users rename the machine definition, control definition, or post (with either a .PST or .MCPOST extension) while preserving the links between all the files. In addition, for Mastercam 2023, the add-in has been enhanced so that it can back up your original files to a .mcam-content file.



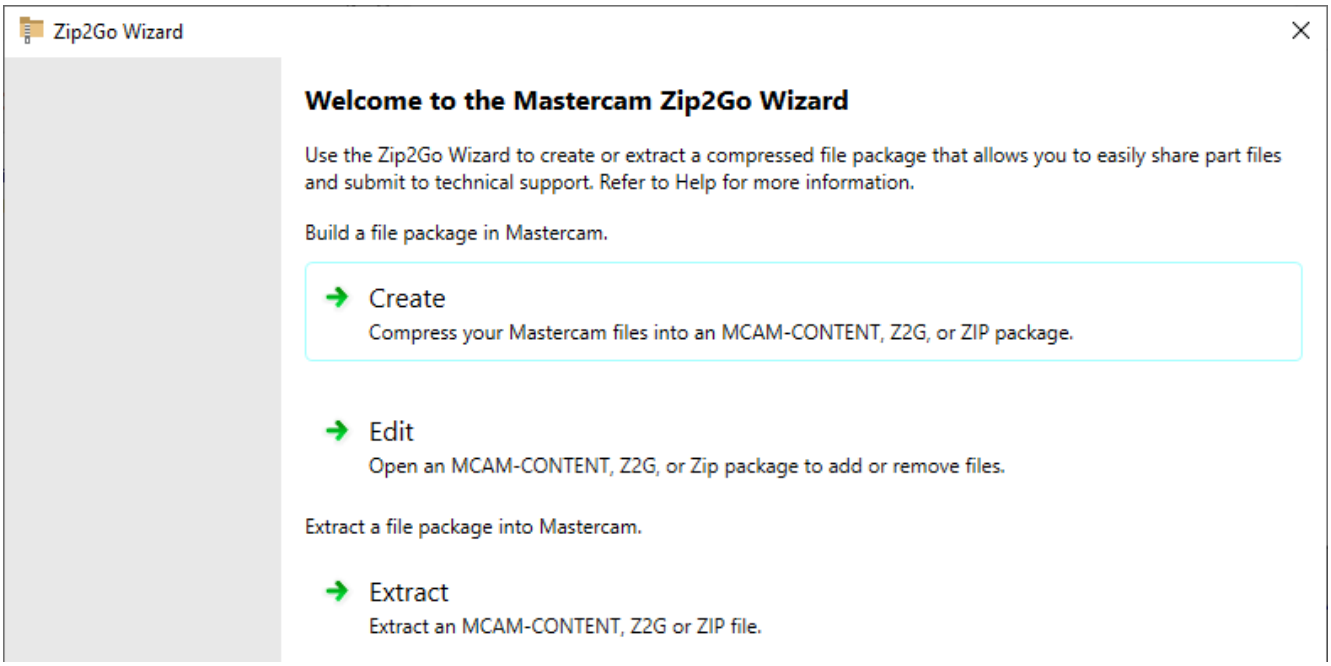
FILE MANAGEMENT ENHANCEMENTS

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

Compressing and Extracting Mastercam Files

Mastercam's new **Zip2Go Wizard** leads you through the process of compressing and extracting Mastercam files, and allows you to customize the file types and machine groups in the package. Use the **Zip2Go Wizard** to send small file packages to technical support and other Mastercam users.

The **Zip2Go Wizard** replaces the Zip2Go utility. To access the new wizard, select **File**, **Zip2Go**.



The **Zip2Go Wizard** lets you create, edit, or extract Mastercam file packages that are in .mcam-content, .z2g, or .zip format.

- **Create** compresses your Mastercam files with the settings you choose. You can compress the entire set of files or individually select components to exclude or include.
- **Edit** displays the contents of a file package, as well as their current location within the file structure and the location to which Mastercam will extract them. Previously, you could only view the contents of an .mcam-content file by changing the file extension to .zip.
- **Extract** opens a file package in Mastercam. By default, Mastercam extracts all files in the selected package. To extract specific files, select an individual file or use [**Shift+click**] to select multiple files.

Better Quality of Imported and Exported CAD Files

Mastercam now improves the importing and exporting of several CAD file formats resulting in increased data quality, an improved user experience, and a lower support burden. This affects the following file formats:

- STEP
- SOLIDWORKS
- Solid Edge
- Autodesk Inventor LT
- ACIS-kernel-based CAD formats (SAT, SpaceClaim, Alibre Design, and KeyCreator)
- JT and 3DXML (new for Mastercam 2023)

Mastercam is now able to import wireframe geometry and 3D annotations from SOLIDWORKS, Inventor, and Solid Edge files.

Files from SOLIDWORKS and Autodesk Inventor no longer need software to import functional data. This enhancement fixes an issue in previous releases where conflicting versions of the **SWDocumentMgr.DLL** affected the proper import of SOLIDWORKS files.

In previous versions of Mastercam, there were often issues importing data (sketch data for SOLIDWORKS, colors, annotations, and assemblies for Autodesk) if you did not have the product installed or the proper file versions.

Mastercam 2023 can also import all solids saved in a SpaceClaim assembly file. Previous versions of Mastercam would only import these assemblies as a single solid.

Improved Quality for Importing and Exporting of 3D Annotations

Mastercam is now able to import wireframe geometry and 3D annotations from SOLIDWORKS, Inventor, and Solid Edge files and export 3D annotations to IGES and AutoCAD DWG/DXF files. 3D annotations in Mastercam are written to IGES files as copious data (type 106) entities and to AutoCAD DWG/DXF files as line entities within AutoCAD Blocks.

Additionally, semantic data from **Surface Finish** 3D annotation types is now imported. Once imported, you can view the annotation attributes using the **Analyze** functions.

Exporting Drafting Entities to STEP Files

You can now export dimensions and notes to the STEP file format. The dimensions and notes will be written to the STEP file as 3D annotations. Previous versions of Mastercam could only export these entities to IGES and AutoCAD DWG/DXF.

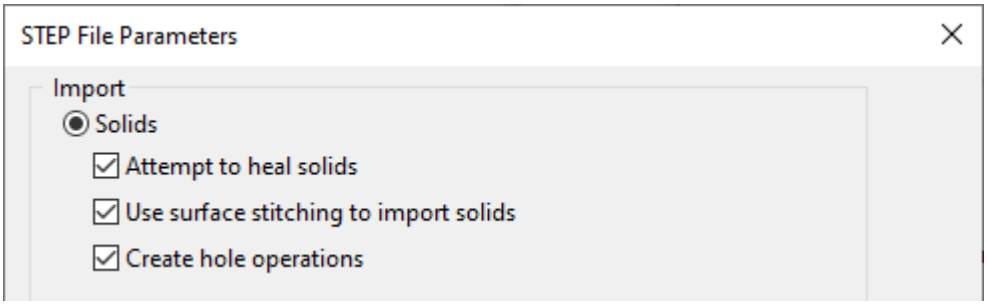
Quickly Find a 3D Annotation Associated with an Entity

Mastercam 2023 includes a new add-in, **3DAnnotationFinder**, that allows you locate all associated 3D annotations by selecting the geometry that is related to them. In previous versions of Mastercam, the only way to display this relationship was to use the **Analyze Entity** function and select the 3D annotation itself.

To open the add-in, select **Run Mastercam Add-In** from the **Run Add-In** drop-down on the **Home** tab. Then select the **3DAnnotationFinder.dll** to open the function. Select a solid face or edge, a surface, or wireframe entity. Mastercam highlights the 3D annotation associated with the entity and displays it in the **3D Annotations Properties** dialog box. Use the **Previous** and **Next** buttons to cycle through any other associated 3D annotations, including annotations that are not currently visible.

Enhancements When Importing Solid Models

Mastercam 2023 includes three significant enhancements when importing solid models.



Healing Stitched Models

Mastercam's solids importers now heal stitched models. In previous releases, selecting the **Use surface stitching to import solids** option disabled the **Attempt to heal solids during import** option. In Mastercam 2023, selecting either of those options no longer disables the other. This can yield better solid bodies with less effort.

Automatically Detect Hole Features

You now have the option to automatically detect Mastercam and create Solid Hole operations when importing solid models. Selecting this option shortens the time and work necessary to go from importing to drilling.

Improved Management of Assemblies, Sub-assemblies, and Component Parts

Mastercam 2023 creates groups for imported solid models that include assemblies, sub-assemblies, and component parts. These groups and the intuitive naming conventions will help you better manage the relationships between the solids in Mastercam.

Additionally, the **Group** right-click menu option now includes new options. Use **Show**, **Hide**, and **Show Only** to display only components within the currently selected group in the graphics window.

Newly Supported File Formats

Mastercam 2023 now includes support for the following file formats: JT, 3DXML, AMF, OBJ, and PDF.

- **JT** is an open-source 3D file format promoted by Siemens. JT files support solids, meshes, wireframe, and 3D annotations. Mastercam 2023 can import JT files, but cannot export in this format.
- **3DXML** is a proprietary 3D file format developed by Dassault Systèmes. 3DXML files support solids, meshes, wireframe, and 3D annotations. Mastercam 2023 can export in this format, however a CATIA V5 license is required to import 3DXML files into Mastercam.

- **AMF** (Additive Manufacturing Format) and **OBJ** (Wavefront files) are triangular mesh file formats. Importing these files types creates mesh entities in Mastercam. Both formats can export surfaces, solids, meshes, and stock models. Additionally, the OBJ format exports wireframe geometry as polylines.
- You can now import 2D PDF files into Mastercam. PDF (Portable document format) is a proprietary file format developed by Adobe. Since PDF files are one of the most used file types today, the ability to import them into Mastercam provides you with a ready source of design material for engraving and other operations. Each page of the document comes in on a separate level and on a viewsheet assigned to that level. Entities in the file come in as wireframe and text.

  Mastercam cannot imported 3D PDF data and Mastercam cannot export 2D PDFs.

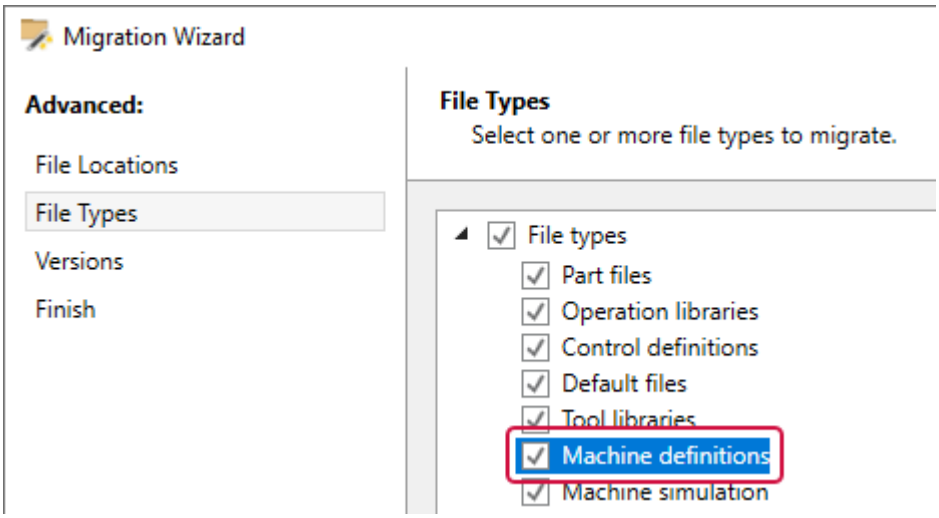
Import and export these files with the **Open**, **Merge**, **Save**, **Save As**, and **Save Some** functions on the **File** tab. This workflow is the same as any other Mastercam-supported CAD file.

The **Converters** page of the **System Configuration** dialog box lets you set default parameters for conversions to and from the .mcam file and these formats. Override these parameters when exporting individual files by choosing **Options** in the **Save As** dialog box when available.

Adding Migrated Machine Definition Files to the Machine List

Mastercam 2023 makes using migrated machine definition files easier by automatically populating the **Machine Type** drop-down on the **Machine** tab with migrated machine definition files. In earlier versions of Mastercam, you clicked **Manage List** to select and add machine definition files to the drop-down options.

To do this, select **Machine definitions** in the **File Types** page of the Migration Wizard. Follow the prompts to complete the file migration.

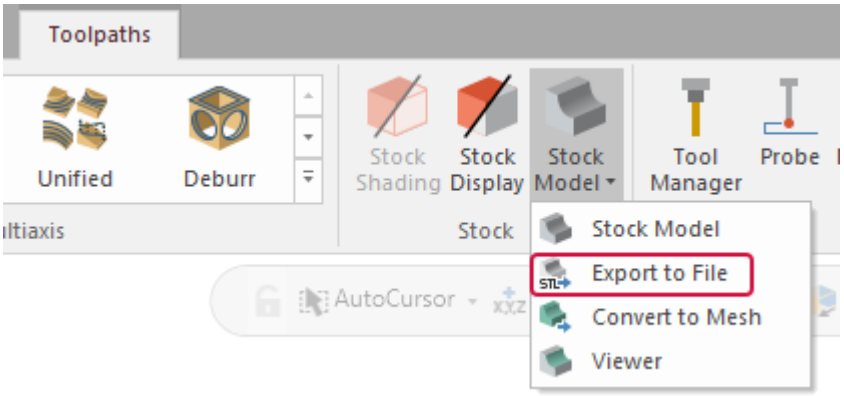


When you create a new **Machine Type**, the drop-down includes the migrated machine definitions.

Exporting a Stock Model Into Different File Formats

Mastercam 2023 expands the exporting of stock models to more file formats. You can now export stock models to the following polygonal mesh formats: 3MF, AMF, and OBJ. In previous releases, you could only export stock models as an STL file.

To reflect this change, the **Stock Model Export to STL** option in the Toolpaths Manager right-click menu is now **Stock Model Export to File**. The **Export to STL** option in the **Stock Model** drop-down on the **Toolpaths** tab is now **Export to File**.



GENERAL ENHANCEMENTS

Listed below are general enhancements to Mastercam.

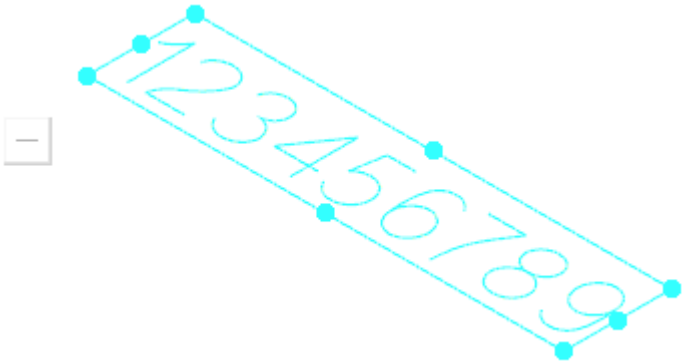
New Mastercam Learning Edition

The Mastercam Demo/Home Learning Edition (HLE) application has been renamed to Mastercam Learning Edition. Additionally:

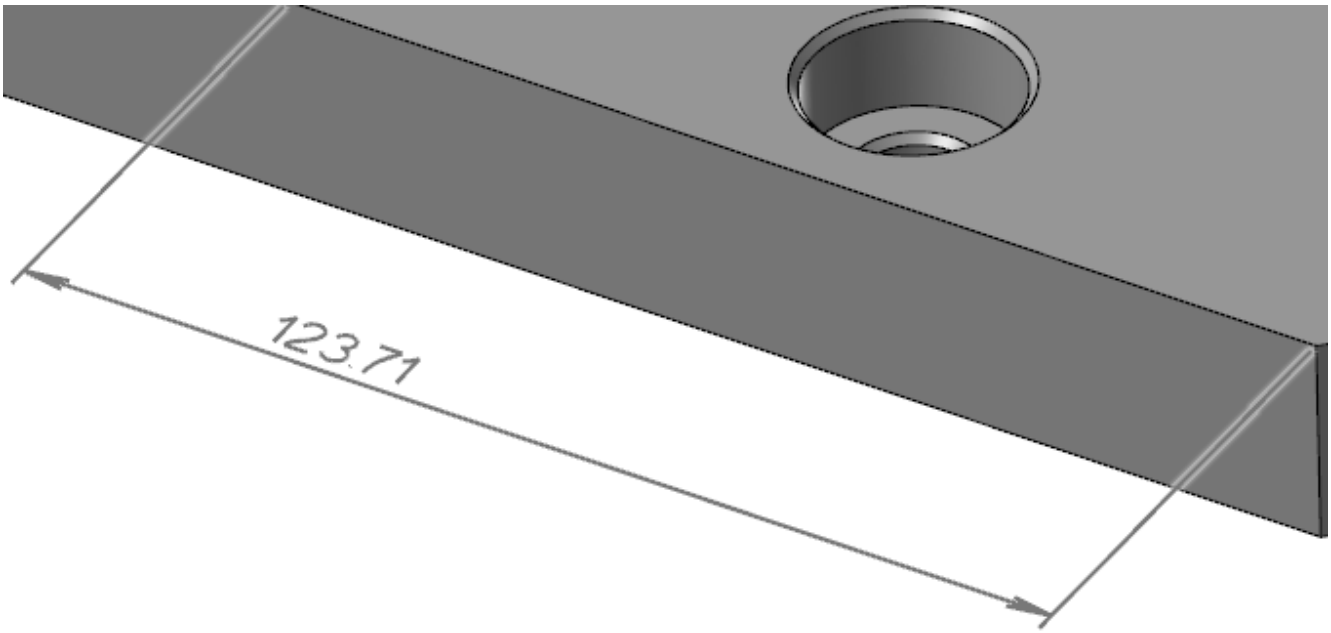
- You can now run the **Check Tool Reach** function without restrictions.
- You can export Wavefront (OBJ) and Additive Manufacturing Format (AMF) file formats.

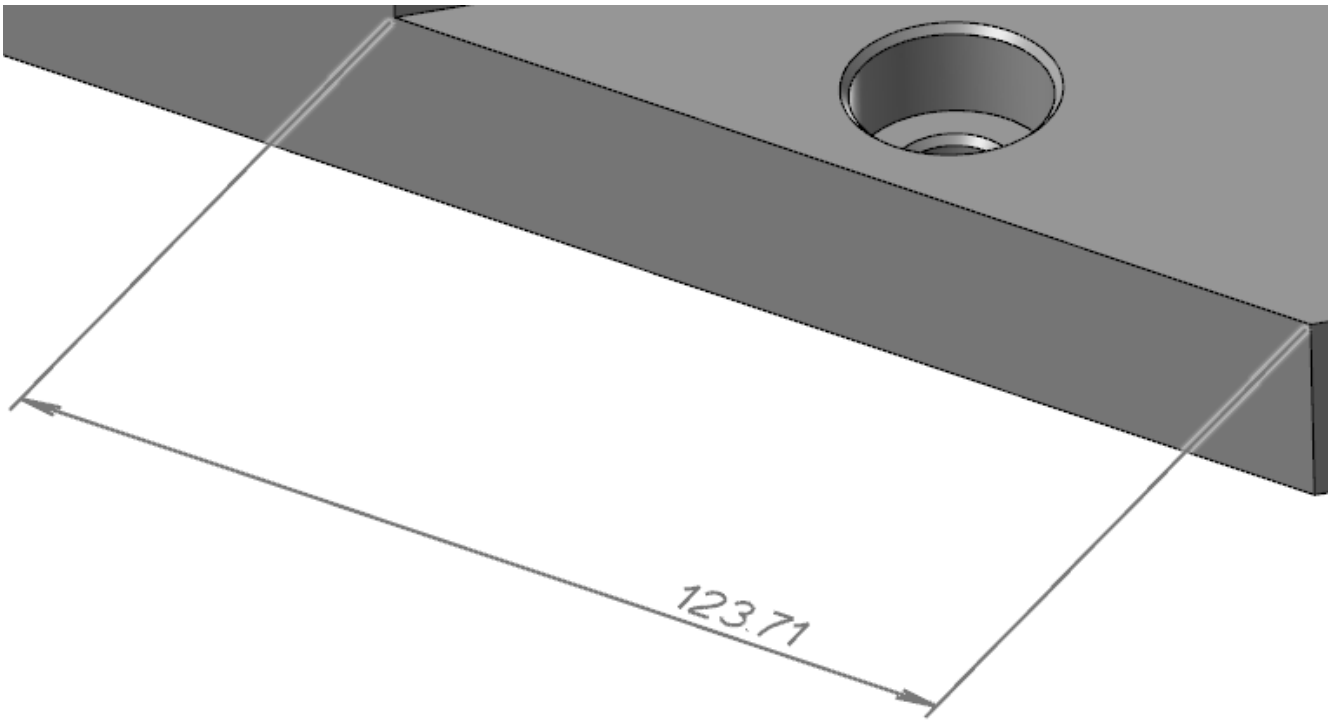
Improving Efficiency When Using Analyze

Mastercam 2023 includes multiple improvements to the **Analyze Entity** function, including overall speed when the **Analyze Entity** dialog box is open. When using **Analyze Entity** on a note, you can copy, move, or re-size the note using the containment box and a new button in the graphics window.



When using **Analyze Entity** on a dimension, you can reposition the text by selecting the dimension again.





When analyzing a mesh Machine Component Object (MCO), the **Analyze Entity** dialog box title changes to reflect that.

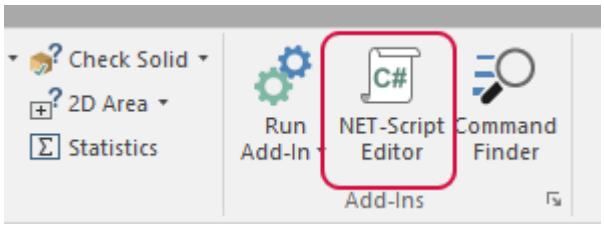
Combining Duplicate Error and Warning Messages

Mastercam lists duplicate errors as a single message followed by the number of occurrences. The message states that there were one or more errors or warnings, and then lists them below. This reduces on-screen clutter and allows for quicker message review.

Working with the NET-Script Editor

The NET-Script Editor features multiple enhancements in Mastercam 2023 that make it easier to create, edit, compile, and run C# scripts directly inside the software. These enhancements include:

- The NET-Script Editor is now available on the **Home** tab.



- Syntax highlights make the code easier to read and navigate. In addition, a yellow bar displays next to the active line of code in the editor to make it easy to see where you left off.
- Available API keywords auto-complete when you begin to type them, and a tooltip displays information about each option. Save time by no longer switching back and forth between the script in Mastercam and the API documentation. Along with auto-complete, auto-import is now available for certain options in the API, leading to less overhead while typing.

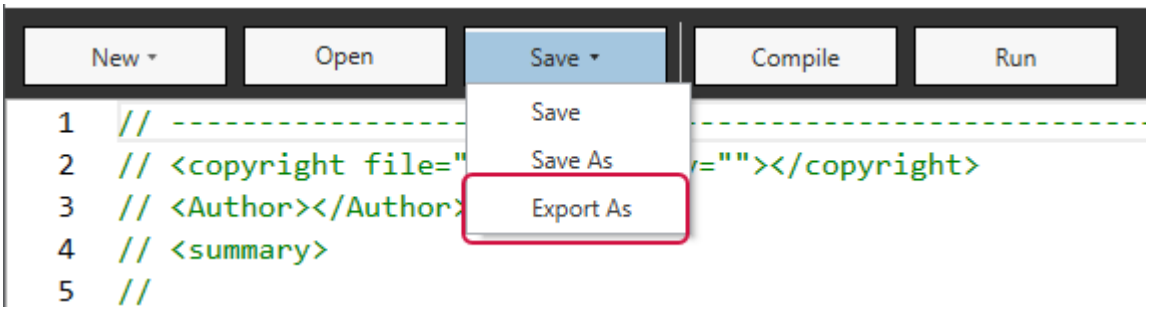
- The script names display in the dialog box header of the editor, with an asterisk next to it when unsaved changes are present. This is useful for identifying a script when working in multiple editors.
- Mastercam auto-formats the script when you press [**Ctrl**+**D**] to make it easier to clean up your code.
- Tooltips display for buttons within the editor and you can now set keyboard shortcuts to the editor functions.

CAUTION
Due to the enhanced functionality of the NET-Script editor, VBScript and the VBScript Manager have been deprecated and will be completely removed from Mastercam 2024. We recommend that you convert your VBScripts to Mastercam NET-Scripts. If you need assistance, contact SDK@mastercam.com.

Turning a NET-Script into a Mastercam Function

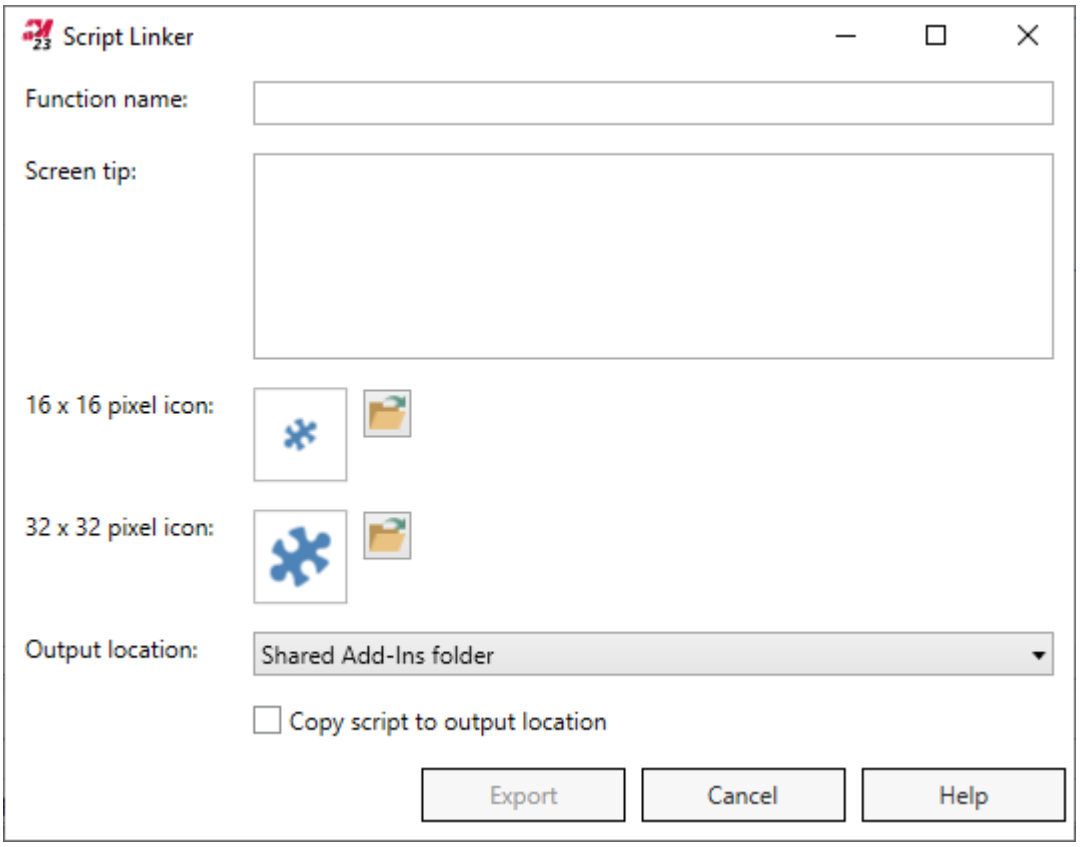
Mastercam 2023 introduces the Script Linker, which lets you create a Mastercam function from a NET-Script. You can add the function to the ribbon, Quick Access Toolbar, or contextual menu. The Script Linker saves you time by allowing you to launch the script with the click of a button instead of opening it from the NET-Script Editor.

To access the Script Linker, open the desired script in the NET-Script Editor and select **Export As** from the **Save** drop-down menu.



Once the Script Linker opens, create the new function by:

- Naming the function
- Entering a tooltip
- Adding icons
- Choosing the output location



Typically, you want to save the function to the shared Add-Ins folder so that all users of the specific computer can access it. However, you also have the option to save the function to the Add-Ins folder in your My Mastercam 2023\Mastercam\ folder, if you only want the function available during your own Mastercam session.

Debugging Mastercam .NET-Scripts in Visual Studio Code

The new Mastercam .NET-Scripting extension, for use in Visual Studio Code, is an advanced development feature that allows you to debug your Mastercam .NET-script by directly connecting to Mastercam 2023. Run your script in Visual Studio Code and watch as it runs line-by-line in Mastercam. Use this extension to find issues and fix defects in the .NET-script. You can make changes as necessary in the VS Code editor, and then re-run the script to view the changes.

The extension will soon be available in the Visual Studio Code extensions marketplace. Currently, you can contact the Mastercam SDK Team by email to request the extension: SDK@mastercam.com.

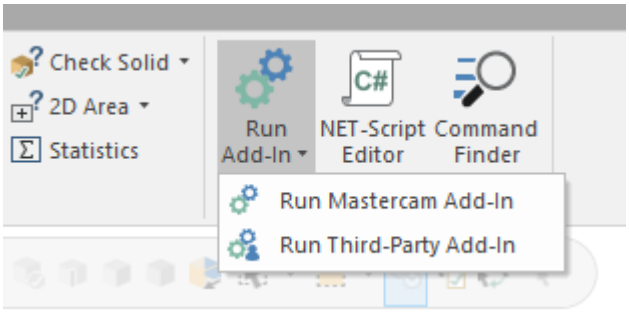
CAUTION
Due to the enhanced functionality of the NET-Script editor, VBScript and the VBScript Manager have been deprecated and will be completely removed from Mastercam 2024. We recommend that you convert your VBScripts to Mastercam NET-Scripts. If you need assistance, contact SDK@mastercam.com.

Storing Third-Party Add-ins Outside of the Mastercam Installation

Mastercam 2023 improves the add-in experience with **Run Third-Party Add-in**, a function that allows you to quickly open third-party add-ins from the ../Documents/My Mastercam 2023/Mastercam/Add-Ins folder.

Store third-party add-ins in this folder when you do not want to require administrator credentials to run the add-in. This is particularly useful when multiple users share a workstation. Additionally, this folder is not affected when you update or reinstall Mastercam, making it a useful place to store add-ins you use across Mastercam releases. Although this folder was included with Mastercam 2022, Mastercam 2023 is the first release to make it accessible from the ribbon.

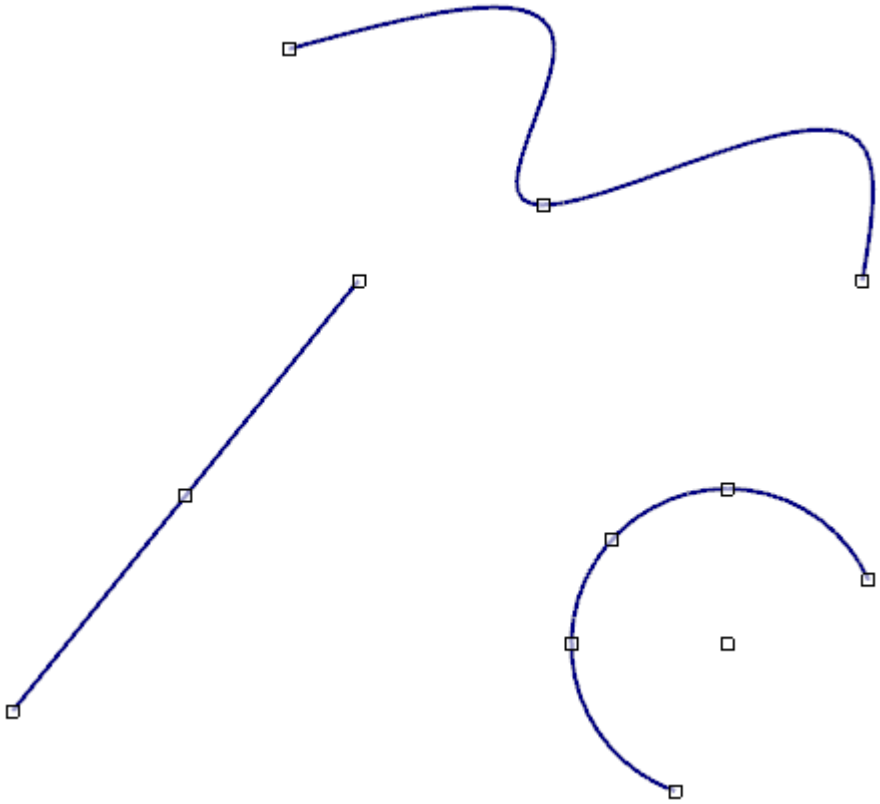
To accommodate this new feature, the **Run Add-In** button on the **Home** tab is now a drop-down menu with two options: **Run Mastercam Add-In** and **Run Third-Party Add-In**. **Run Mastercam Add-In** functions the same as **Run Add-In** functioned in previous versions of Mastercam.



CAUTION
Due to the enhanced functionality of the NET-Script editor, VBScript and the VBScript Manager have been deprecated and will be completely removed from Mastercam 2024. We recommend that you convert your VBScripts to Mastercam NET-Scripts. If you need assistance, contact SDK@mastercam.com.

Previewing AutoCursor Positions

When AutoCursor is active, you can now hover anywhere over a piece of geometry to display multiple AutoCursor positions at once: **Arc Center**, **Endpoint**, **Midpoint**, and **Quadrant**. These positions must be enabled in the **Selection** page of the **System Configuration** dialog box. In previous Mastercam versions, you needed to hover directly over specific parts of the geometry for an AutoCursor position to appear, and you could only see one type at a time.



Improved and Renamed Mastercam Documentation

As part of ongoing efforts to provide documentation that is modern, user-friendly, and easy to search, Mastercam 2023 features changes to the *Administrator Guide*, the *Mastercam Basics Tutorial*, and the *Resolved Issues* (formerly known as the *ReadMe*). These documents now open in a web browser in an HTML5 format, rather than a PDF.

In earlier releases of Mastercam, these resources opened as PDFs. You do not need an active internet connection to view the website. These changes are in line with recent improvements to other Mastercam documentation, such as the *What's New* and the *Help*, and are meant to give you a consistent and informative Mastercam experience.

These new formats include new home pages with a card-based interface for quick and easy access to information about the most common administrator tasks. You can also use the side navigation to explore the entire resource. As with earlier versions of Mastercam, you can access the updated documents from the **Start** menu.

Mastercam Resolved Issues

Mastercam 2023 features a single version of the *Resolved Issues*, with a sub-category for all Mastercam for SOLIDWORKS items. Previously you only had access to the Mastercam or Mastercam for SOLIDWORKS versions of this document, depending on your installed software. The *Resolved Issues* also includes improvements such as sortable tables, easier filtering capability, and links to notable Mastercam resources.

Mastercam Basics Tutorial

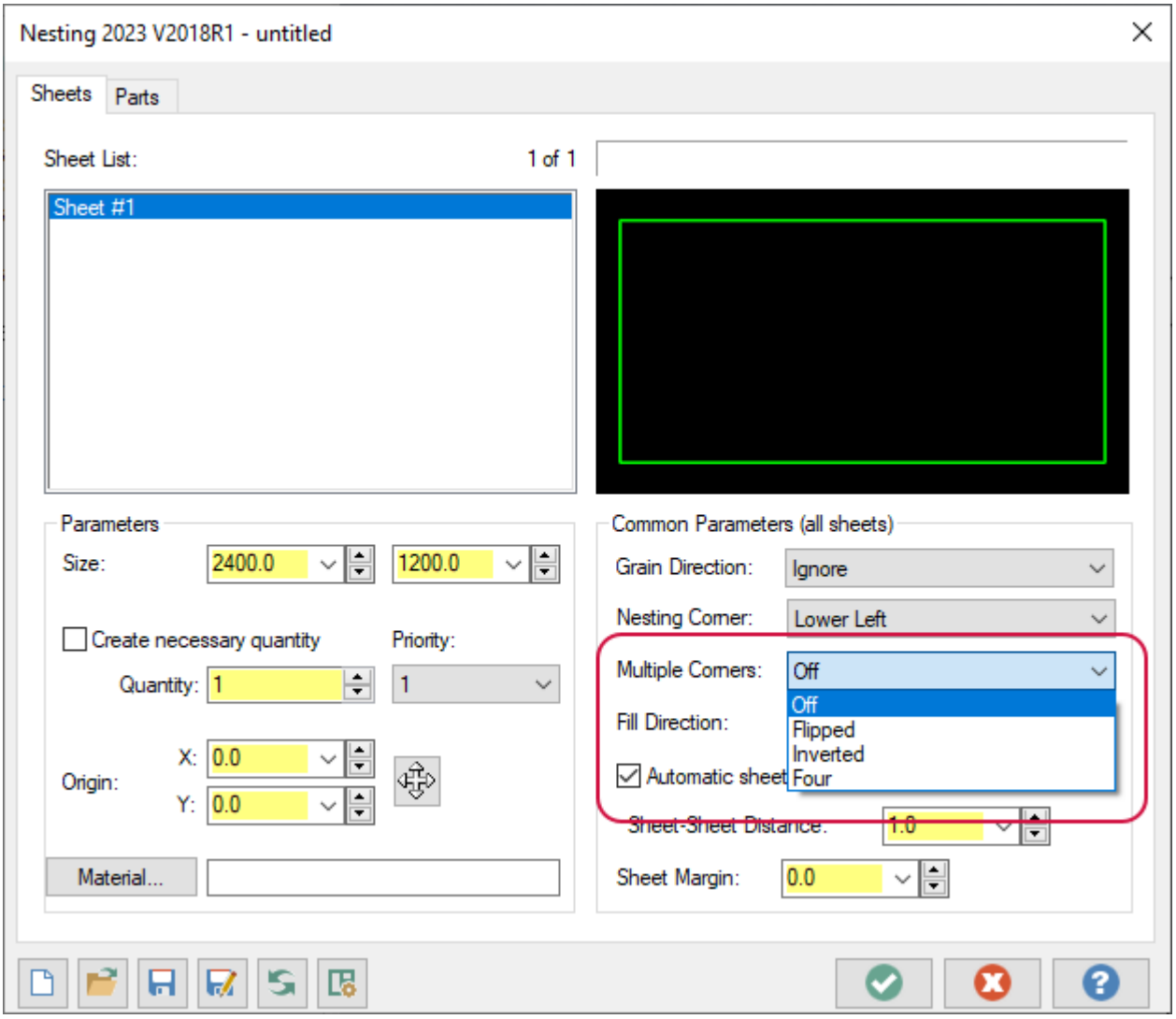
The *Mastercam Basics Tutorial*, in addition to the new format, also includes access to the tutorial parts directly from the website. This removes the need to navigate through Mastercam's installed files to find the tutorial parts. You can also view and download of a copy of the tutorial as a PDF.

Geometry and Toolpath Nesting Enhancements

Listed below are enhancements made to the **Geometry Nesting** function (on the **Transform** tab) and the **Nesting** function (on the **Toolpaths** tab).

Nesting Multiple Corners

The new **Multiple corners** parameter for toolpath and geometry nesting uses more than one corner on the sheet to nest the parts. Mastercam calculates the job from multiple nesting corners and chooses the corner that yields the best outcome. With this new parameter, it is easier to spread the number of required parts evenly throughout the sheet for better vacuumed sheet holding.



The **Multiple corners** parameter is located on the **Sheet** tab and has four options:

- **Off**: Calculates the nesting strategy from the **Nesting corner**.
- **Flipped**: Calculates the best nesting strategy from two corners: the **Nesting corner** and the corner that is perpendicular to the **Nesting direction** and opposite the **Nesting corner**. The result uses the corner that yields the best outcome.

- **Inverted**: Calculates the best nesting strategy from two corners: the **Nesting corner** and the corner that is parallel to the **Nesting direction** and opposite the **Nesting corner**. The result uses the corner that yields the best outcome.
- **Four**: Calculates the nesting strategy using all four corners of the sheet.

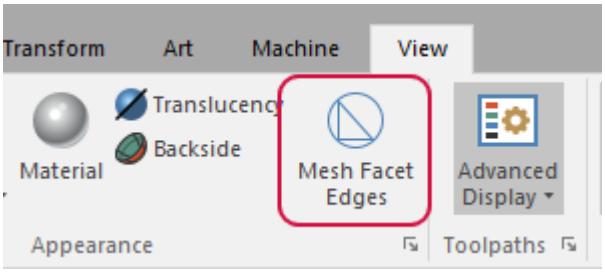
Improved Error Handling and File Support

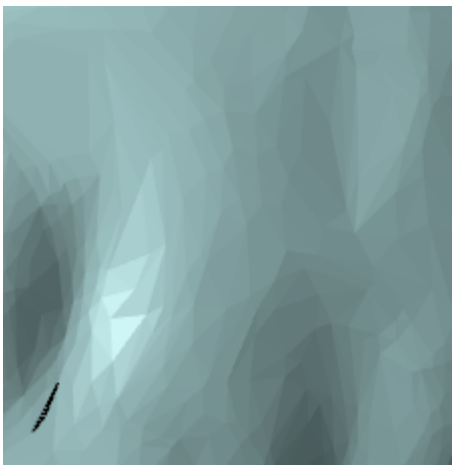
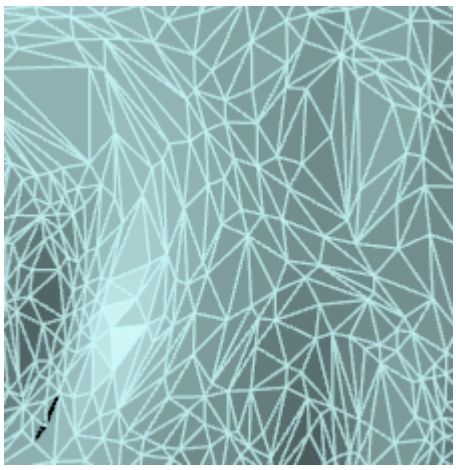
When nesting geometry, all errors and other messages are displayed after completing the operation. Mastercam also supports the loading of multiple default sheets when creating a nested part.

Additionally, your nesting parameter values are remembered for the current Mastercam session. This allows you to fine-turn your results as you work.

Toggling the Display of Mesh Facet Edges

You can toggle the display of mesh facet edges in your part file with the new **Mesh Facet Edges** option located in the **Appearance** group of the **View** tab.

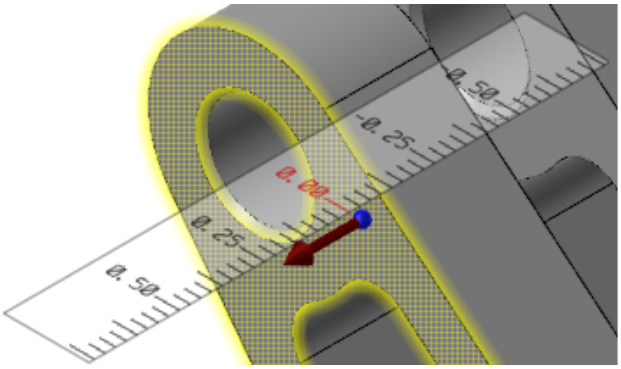


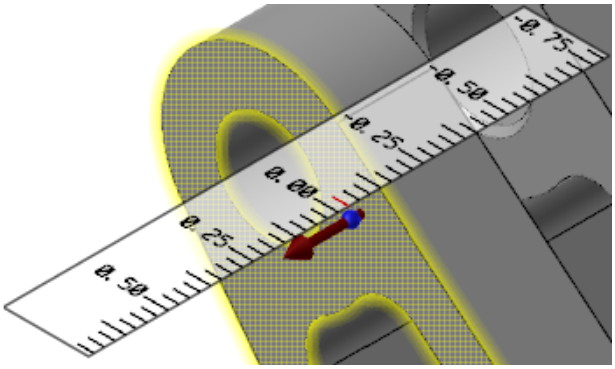
Mesh Facet Edges Off	Mesh Facet Edges On
	

In previous versions of Mastercam, this was an option named **Show triangle edges** in the **Shading** page of the **System Configuration** dialog box.

Automatically Adjust the Opacity of the On-screen Ruler

In Mastercam 2023 the opacity of the on-screen ruler, which displays during functions such as **Dynamic Transform** and **Dynamic Plane**, adjusts depending on the location of your cursor, making it easier to view your part. Hover the cursor off the ruler to display more opacity, or hover the cursor over the ruler to display less opacity.





POSTS AND MACHINE ENVIRONMENTS

Listed below are the new post and machines for Mastercam 2022. These can be downloaded from the Mastercam Tech Exchange.

New Posts and Machines for Mastercam 2022

The following machine environments and posts are now available for Mastercam 2022. These machines can be downloaded from the Mastercam Tech Exchange.

NOTE
Not all machines and posts are available for download. Contact your local Mastercam Reseller for more information about acquiring a machine or post. You can also contact your Reseller for more information about the Mastercam 2023 versions.

MP multiaxis posts

The following posts have been released for Mastercam Mill. These are all multiaxis posts that are supplied with several different machine definitions to support various machine configurations.

Machine	Control	Configurations	Notes
Okuma 5X Mill			
Genos M460V-5AX	OSP	Various	None
Genos M560-V 5X TRT-AC			
Genos M560-V 5X TRT-BC			
MU-4000V			
MU-5000V			
MU-6300V			
Fanuc 5X Mill			
Doosan DVF 5000	Fanuc 30i 31i 32i	Various	A new Fanuc post has been created to include all Fanuc derivatives. This encompasses Doosan and Robodrill machines, and can be modified for any machines.
Doosan DVF 6500			
Doosan DVF 8000			
Doosan VC 630/5AX			
Doosan DNM 200/5AX			
Doosan DNM 350/5AX			
Robodrill			
Generic Fanuc 5X Mill			

Machine	Control	Configurations	Notes
Siemens 5X Mill			
DMG DMU 50 Gen2	Siemens 840D	• Various • Head-head AC • Head-head BC • Table-table AC • Table-table BC	The Siemens post now adds the Doosan machines to its portfolio.
Doosan DVF 5000			
Doosan DVF 6500			
Doosan DVF 8000			
Doosan VC 630/5AX			
Generic Siemens 5X Mill			
Haas 5X Mill			
Haas UMC-1000	Haas NGC	Table-table	New generation Haas 5-axis Mill post, replacing the previous UMC-750. This post supports: • The newer generation of the Haas control • DWO (G254) • TCPC (G234) • TWP (G268) • High speed machining modes • G10 list (tools and work datums)
Haas UMC-1000SS			
Haas UMC-1250			
Haas UMC-1250SS			
Haas UMC-1500-DUO			
Haas UMC-1500-DUO			
Haas UMC-1600-H			
Haas UMC-500			
Haas UMC-500SS			
Haas UMC-750			
Haas UMC-750SS			

Lathe machine environments

The following machine environments have been released for Mastercam Lathe. These let users take advantage of Mastercam Mill-Turn's interface and features (including Simulation) for single-stream machines with no more than one rotary axis. These machines require only a Lathe and Mill license to run.

Machine	Control	Configuration
Doosan		
Lynx 2600Y_BMT55x24	Fanuc i series	Single Spindle / Single Turret / Tailstock
Emco		
Emcoturn E45 SMY	Heidenhain PILOT 640	Dual Spindle / Single Turret

Machine	Control	Configuration
Mazak		
QT-Compact 300MY 500U_12st	Nexus 2	Single Spindle / Single Turret / Tailstock
CMZ		
TD35 Y Z1350_16st BO	Fanuc 31i-A	Single Spindle / Single Turret / Tailstock / Steady Rest
TD45 Y Z3200_12st BOT		
TD45 Y Z2200_12st BOT		
DMG Mori Seiki CTX		
CTX gamma 3000_R – Gen 1	Siemens 840D	Single Spindle / Single Turret / Tailstock / Steady Rest
Okuma LB		
LB3000 EX II L x1000C	OSP-P300L	Single Spindle / Single Turret / Tailstock
Haas ST		
ST-30 Y_S_BMT65x24_v2	Haas CNC	Dual Spindle / Single Turret
ST-15_S_Hybrid_v2		
DMG Mori Seiki NL		
NL2500SY 700_12st	Fanuc 31i-A	Dual Spindle / Single Turret
Mazak Quick Turn		
Quick Turn 350MSY 1500U_VDI	Nexus 2	Dual Spindle / Single Turret
Hwacheon Hi_TECH series		
230AL YSMC_BMT65X24	Fanuc 0i-MF	Dual Spindle / Single Turret
230BL YSMC_BMT65X24		
230CL YSMC_BMT65X24		
DMG Mori Seiki NLX		
NLX3000MC 3000TSY_R_12st BOT	Mitsubishi (CELOS)	Single Spindle / Single Turret / Tailstock / Steady Rest
NLX1500MC 500SY_12st BOT		Dual Spindle / Single Turret
NLX2000MC 500SY_12st BOT – Gen 2		Single Spindle / Single Turret / Tailstock / Steady Rest
NLX2000MC 1250TSY_R_12st BOT		Dual Spindle / Single Turret
NLX1500MC 500SY_20st BOT		Dual Spindle / Single Turret

Mill-Turn machine environments

The following machine environments have been released for Mastercam Mill-Turn. These machines require a full Mill-Turn license.

Machine	Control	Configurations
Emco Hyperturn		
Emco HT 665MC-Plus TCMY	Siemens 840D-SL	Dual Spindle / Dual Turret
Victor		
Victor Vturn-X200	Fanuc 31i-A5	Dual Spindle / Tool Spindle / Lower Turret
Citizen Miyano		
ABX-64SYY2_v2	Fanuc 31i-B	Dual Spindle / Dual Turret
Eurotech		
Rapido B436-Y2	Mitsubishi M700	Dual Spindle / Dual Turret
Takisawa TMX		
TMX-2000S	Fanuc 31i-B5	Dual Spindle / Tool Spindle
TMX-4000ST		Dual Spindle / Tool Spindle / Lower Turret
Mazak Integrex j-series		
j-200 500U	Smooth G	Single Spindle / Tool Spindle / Tailstock
J-200S		Dual Spindle / Tool Spindle
j-300		Single Spindle / Tool Spindle / Tailstock
j-400		Single Spindle / Tool Spindle / Tailstock
DMG Mori Seiki CTX		
CTX beta 1250 TC 4A – Gen 2	Siemens 840D (DMG Structure)	Dual Spindle / Tool spindle / Lower Turret
CTX gamma 3000 – Gen 1		Single Spindle / Single Turret / Tailstock
CTX gamma 1250 TC 4A – Gen 2		Dual Spindle / Tool Spindle / Lower Turret
CTX gamma 2000 TC 4A – Gen 2		Dual Spindle / Tool Spindle / Lower Turret
CLX 450 TC_S		Dual Spindle / Tool Spindle
DMG Mori Seiki NTX		
NTX1000 SZM – Gen 2	Siemens 840D (DMG Structure)	Dual Spindle / Tool Spindle / Lower Turret
NTX2500 1500SZM – Gen 2_ BMT60x10		
Nakamura-Tome		
Super NTJX	Fanuc 31i-A5	Dual Spindle / Tool Spindle / Lower Turret

Machine	Control	Configurations
NTY3-100	Fanuc 31i-B	Dual Spindle / Triple Turret
Okuma LU		
LU4000 EX-MY 2SC x2000	OSP-P300L	Single Spindle / Dual Turret / Tailstock
LU35-II 2SC x1500		Single Spindle / Dual Turret / Tailstock
Okuma Multus U		
U4000 2SW x1500_Compact-H1	OSP-P300	Dual Spindle / Tool Spindle / Lower Turret
U4000 2SW x2000_Compact-H1		Dual Spindle / Tool Spindle / Lower Turret
U4000 1SW x1500_Compact-H1		Dual Spindle / Tool Spindle
U5000 1SW x3000		Dual Spindle / Tool Spindle
Mazak Integrex e-series		
E-500 H-II 4000U_R	Matrix 2	Single Spindle / Tool Spindle / Tailstock / Steady Rest
E-650 H-II 4000U_R		Single Spindle / Tool Spindle / Tailstock / Steady Rest
E-500 H-SII 3000U_R		Dual Spindle / Tool Spindle / Steady Rest
E-650 H-SII 3000U_R		Dual Spindle / Tool Spindle / Steady Rest
Mazak Integrex i-series		
i-200HST 850U	SmoothAI	Dual Spindle / Tool Spindle / Lower Turret
i-250HST 1500U		Dual Spindle / Tool Spindle / Lower Turret
i-250HS		Dual Spindle / Tool Spindle
i-200HS 850U		Dual Spindle / Tool Spindle

Attention! Updates may be available. Please refer to my.mastercam.com/product-downloads for the latest downloads.

CNC Software, LLC
671 Old Post Road
Tolland, CT 06084 USA

www.mastercam.com