



What's New in Mastercam 2026

May 2025



What's New in Mastercam 2026

Terms of Use

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

<https://www.mastercam.com/privacy/>

Be sure you have the latest information!

Information might have changed or been added since this document was published. Current versions of our documents are available on [myMastercam.com](https://www.mastercam.com/). A *What's New* and a *Release Notes* table—both available on [myMastercam.com](https://www.mastercam.com/)—include the latest information about Mastercam features, enhancements, and bug fixes.

Table of Contents

Table of Contents	3
Introduction	5
Release Highlights	5
Mastercam Resources	5
Milling Enhancements	7
Access Machine Group Setups from the Toolpaths Manager	7
Redesigned Home / Ref. Points Page	8
2D Toolpath Enhancements	9
3D Toolpath Enhancements	13
Hole-Making Toolpath Enhancements	13
Multiaxis Toolpath Enhancements	15
Tooling Enhancements	19
Turning Enhancements	27
Graphical Planes Display for Lathe Drill	27
Enhancements to PrimeTurning™	29
Support for Saving and Loading Parameters in Turning Toolpaths	33
Job Setup Improvements	35
Improvements to Part Handling Operations	37
Surface and Wireframe Toolpaths Available in Mill-Turn	39
Tooling Enhancements	41
Designing Enhancements	53
Analyze Deviations Between Mesh Entities	53
Loft Surface Enhancements	53
Solid Hole Enhancements	57
Selecting Solid Bodies, Edges, and Faces	59
Updated CAD File Support	59
Simulation Enhancements	61
Fine Tune Simulation Speeds	61

Focus on Current Operation Information	61
Reflection Effect Added to Mastercam Simulator	62
System Enhancements	65
Redesigned Levels Manager	65
Redesigned Planes Manager	68
Control Plane Display with Overlay Controls	69
Control AutoCursor Preview Display	70
Disable Inaccessible Functions	70
New Freehand Selection	71
Include Configuration In Content Files	72
Set a Default Planes Work Offset	72
Verify Geometry Selection	73
Machine Systems Enhancements	75
Link to External Files from the Machine Explorer	75
Mastercam-Specific Icons in Code Expert	76
General Enhancements	79
Mastercam Art Discontinued	79

Introduction

Welcome to What's New in Mastercam 2026! Mastercam 2026 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:

- ["Control Flat-Area Machining with Critical Depths" on page 13](#)
- ["Improved Efficiency with Tapered Helical Entry" on page 11](#)
- ["Prioritize Safety Zone Motion" on page 14](#)
- ["Void Support for Solid Holes" on page 14](#)
- ["Loft Surface Enhancements" on page 53](#)
- ["Import Lathe GTC Tool Data" on page 41](#)
- ["Link to External Files from the Machine Explorer" on page 75](#)
- ["Support for Saving and Loading Parameters in Turning Toolpaths" on page 33](#)
- ["Surface and Wireframe Toolpaths Available in Mill-Turn" on page 39](#)
- ["Enhancements to PrimeTurning™" on page 29](#)
- ["New Interface for Creating Tools and Holders" on page 20](#)
- ["Redesigned Planes Manager" on page 68](#)
- ["Redesigned Levels Manager" on page 65](#)
- ["Verify Geometry Selection" on page 73](#)
- ["Access Machine Group Setups from the Toolpaths Manager" on page 7](#)
- ["Updated CAD File Support" on page 59](#)

Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Documentation*—Mastercam installs a number of helpful documents for your version of software in the Documentation folder of your Mastercam installation.
- *Mastercam Help*—Access Mastercam Help by selecting **Help**, **Contents** from Mastercam's **File** tab or by pressing [**Alt+H**] on your keyboard.
- *Mastercam Channel Partner*—Your Mastercam Channel Partner 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, Learning** from Mastercam's **File** tab to see the latest publications.
- *myMastercam Learning Hub*— We offer additional learning resources that explore key concepts and advanced topics with Teach Me articles, Show Me videos, and more. Please visit my.mastercam.com/hubs/learning/.
- *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 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), X ([X.com/Mastercam](https://x.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 Mastercam, 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 Knowledgebase at kb.mastercam.com.

Contact Us

For questions about this or other Mastercam documentation, contact the Content Development 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.

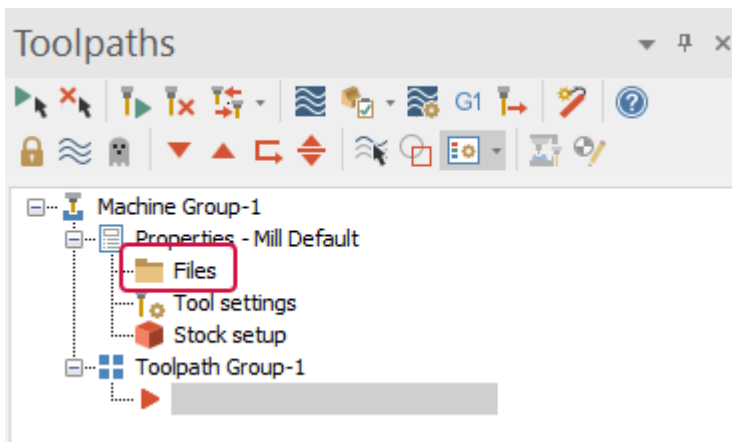


NOTE:

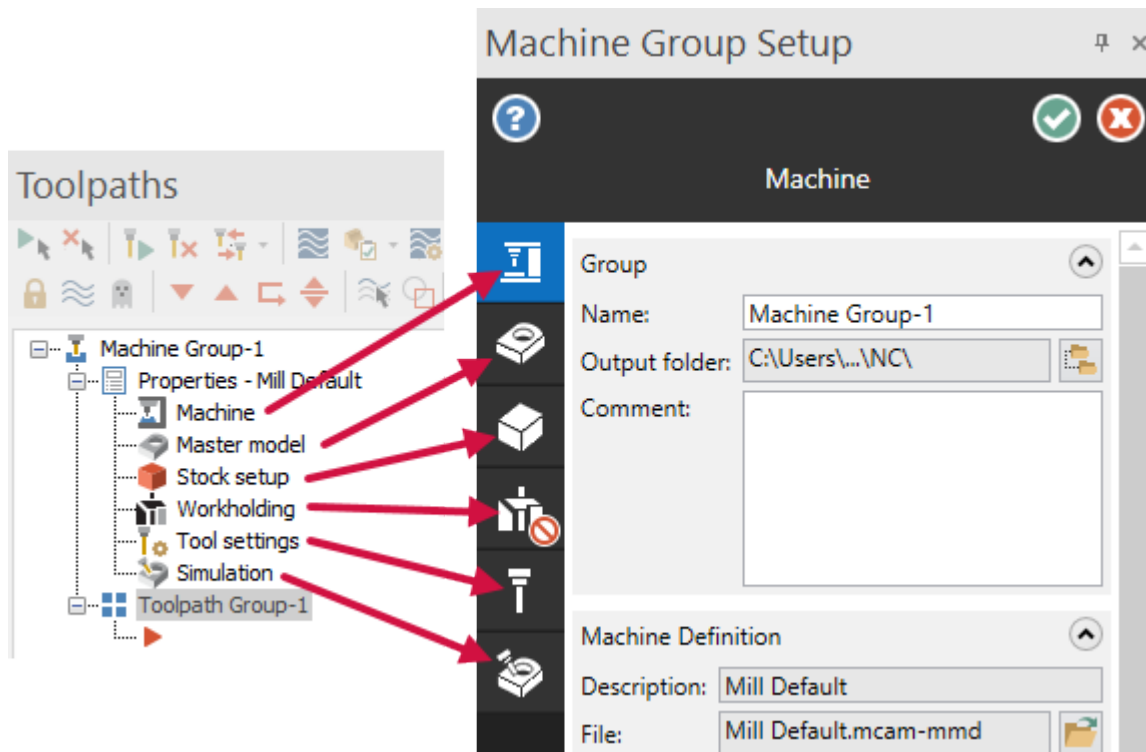
Unless otherwise stated, the new features and functionality listed in this section apply to both Mill and Router licenses.

Access Machine Group Setups from the Toolpaths Manager

In previous versions of Mastercam, the **Properties** list in the Toolpaths Manager included only **Files**, **Tool settings**, and **Stock setup**. To get to the **Machine Group Setup** pages, you clicked the **Files** entry, shown in the following image.



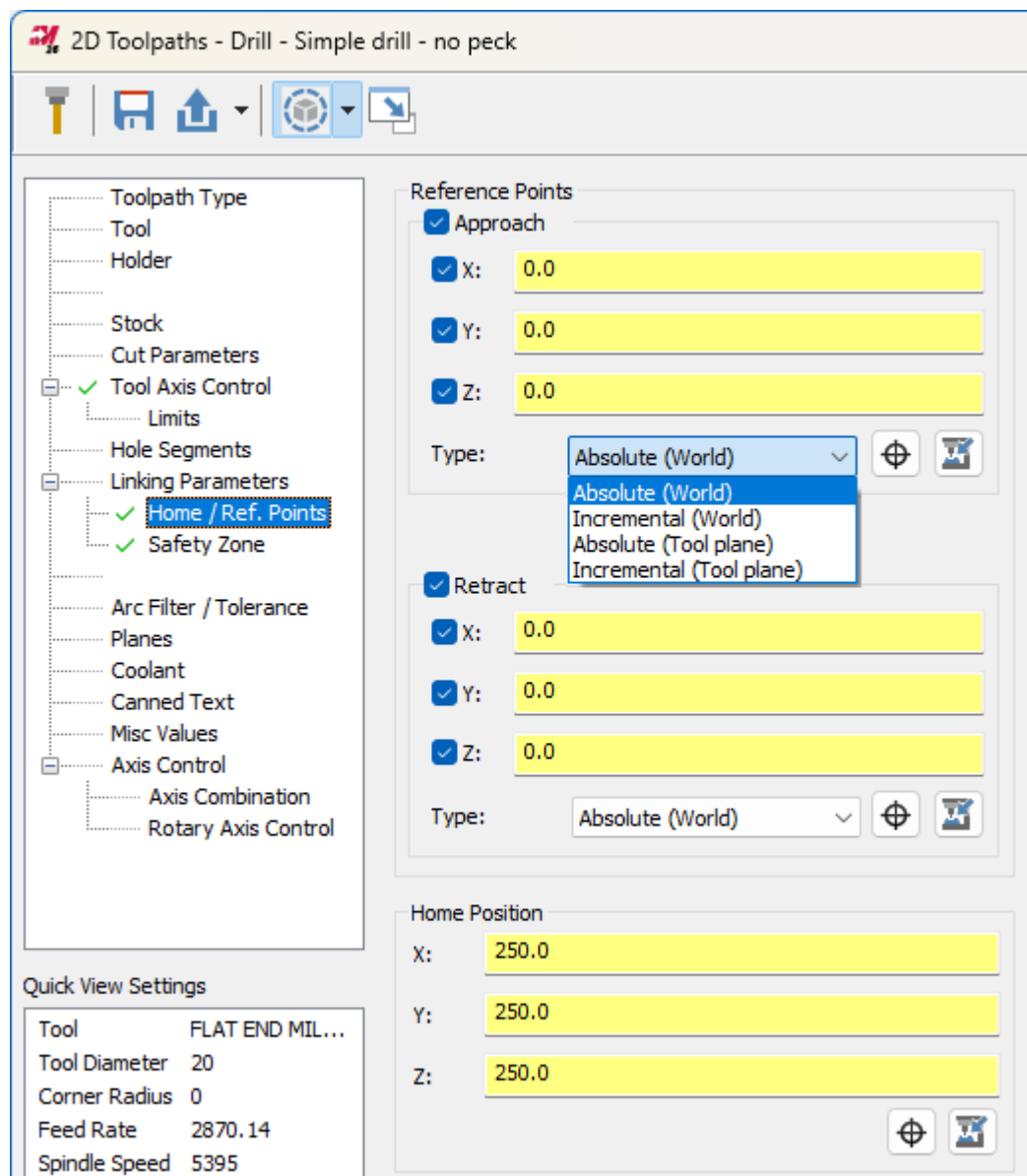
Now, for easier access, the individual **Machine Group Setup** panel pages have been promoted to the **Properties** list in the Toolpaths Manager. Selecting an icon in the tree takes you directly to the appropriate panel.



Redesigned Home / Ref. Points Page

The **Home / Ref. Points** page has been redesigned for Mastercam 2026. The updated layout aligns with the design theme found on other recently updated dialog boxes. The new page design and tool plane options let you set approach and retract points in reference to either incremental or absolute values that reference the tool plane for world coordinates.

You can find these updates in all 2D, 3D, Multiaxis, and Hole making toolpaths. In the toolpath parameters dialog box, open **Linking Parameters** to find the **Home / Ref. Points** subpage.



Following is a list of the new **Type** options:

- **Absolute (World):** Mimics the legacy Absolute functionality except that the position is displayed relative to world coordinates.
- **Absolute (Tool plane):** Allows you to enter the data in tool plane coordinates. The coordinates follow the tool plane as it changes.
- **Incremental (World):** Mimics the legacy Incremental functionality except that the position is displayed relative to world coordinates.
- **Incremental (Tool plane):** Allows you to enter the coordinates relative to the operation's tool plane. The coordinates follow the tool plane as it changes.

2D Toolpath Enhancements

Listed below are enhancements to 2D milling toolpaths.

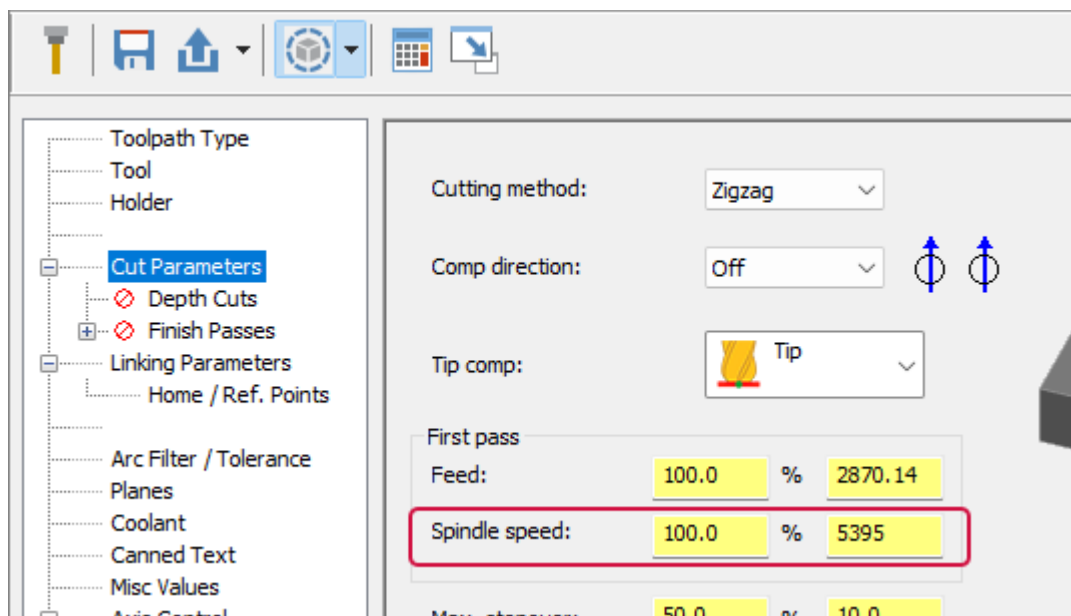
Redesigned Dialog Boxes for Area Mill and Face Mill Toolpaths

The **Cut Parameters** and **Depth Cuts** pages for Area Mill and Facing toolpaths have been updated to coincide with recent design changes. While these changes are primarily cosmetic, they refresh areas of Mastercam and improve the user experience. This update provides a more streamlined workflow and a more consistent user experience.

Customize Spindle Speed for Blend Mill and Dynamic Mill Toolpaths

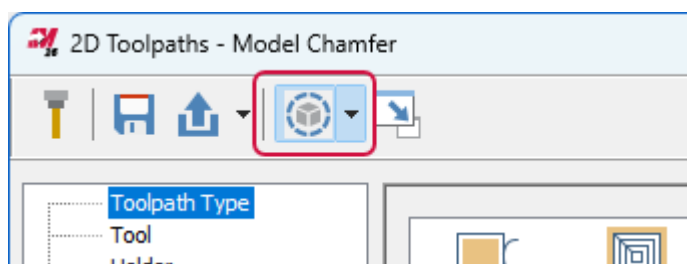
A new **Spindle speed** field has been added to the **First pass** control set for the Blend Mill and Dynamic Mill toolpaths. While in previous versions, you could input a custom feed rate on first passes, this enhancement lets you also specify an accompanying custom spindle speed. The new **Spindle speed** option provides more control over chip load and cutting conditions ensuring safe, efficient motion while improving tool life.

Find the new **Spindle speed** feature on the **Cut Parameters** page for these toolpaths.

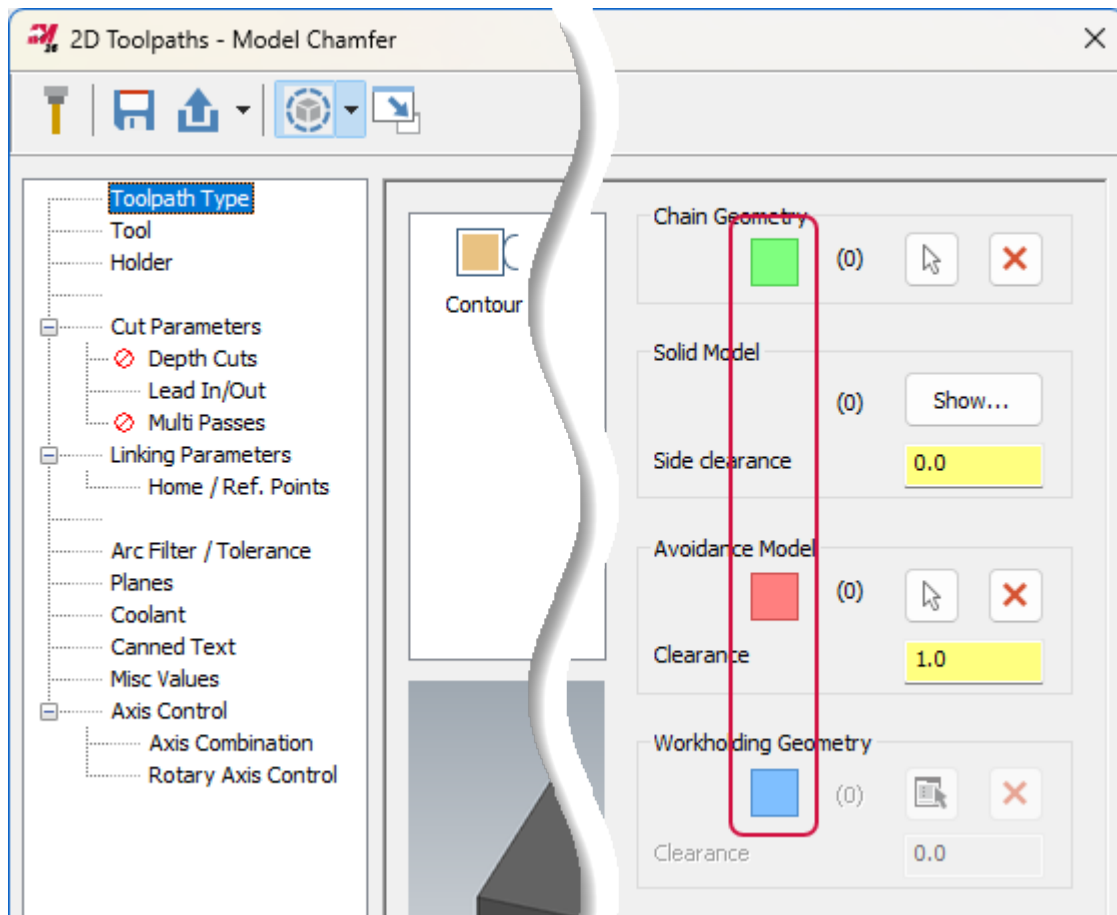


Color Selection for 2D Milling

Color selection options are now available for 2D milling. Use color selection in the **2D Toolpaths** parameters dialog box and toggle the **Display graphical elements** button on to display or hide geometry and other elements.



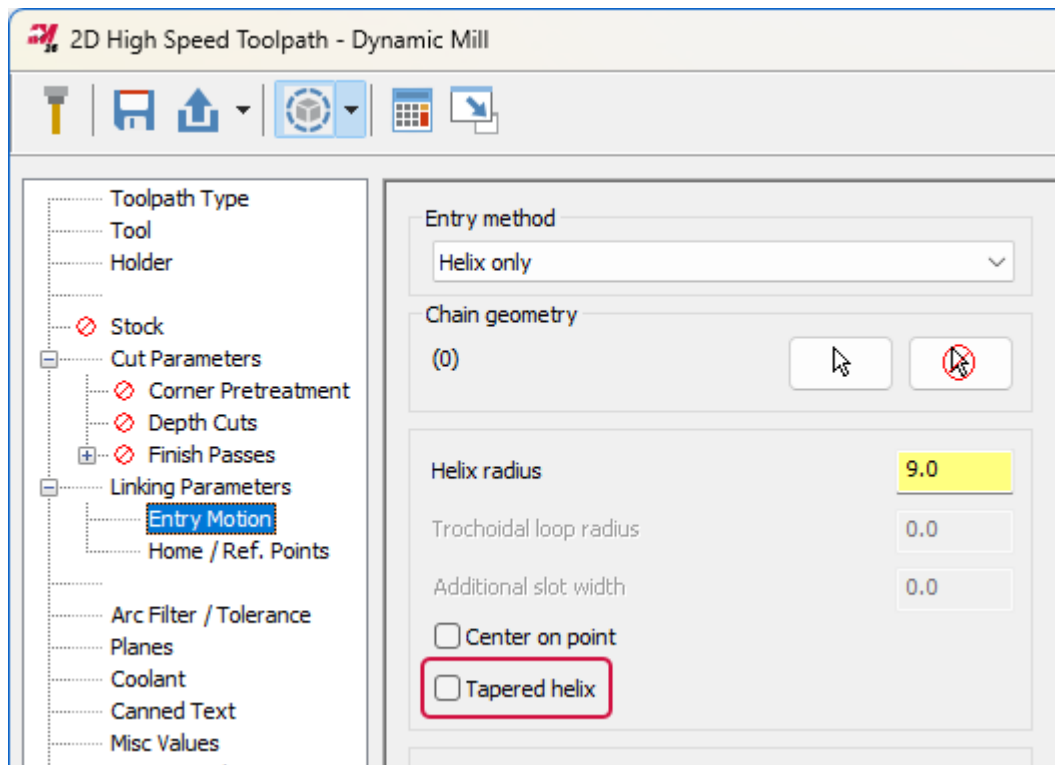
Find the new color selections on the **Toolpath Type** page of the 2D milling toolpath parameters. For more information about the **Display graphical elements** feature, please see the "Display Graphical Elements function" topic in Mastercam's Help.



Improved Efficiency with Tapered Helical Entry

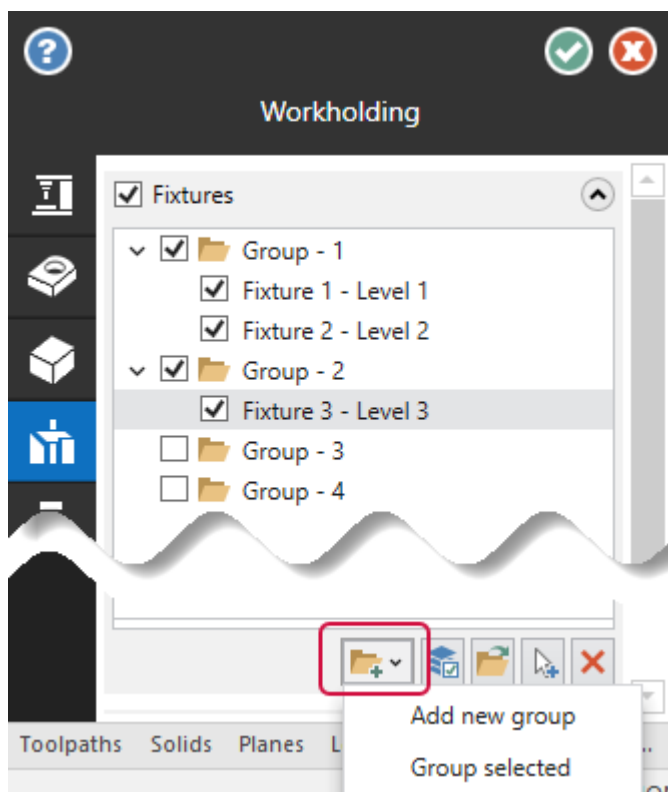
A new checkbox for the **Helix only** entry method lets you produce a tapered helix. Using a tapered helix to enter a pocket promotes better chip evacuation, coolant/air access, and is generally more efficient than a cylindrical helix. A tapered helix offers better tool life and creates more efficient toolpath creation. The taper is defined by using 95% of the tool diameter at the top and 25% at the bottom of the helix.

The new **Tapered helix** feature is located on the **Entry Motion** page in the Dynamic Mill and OptiRough toolpath dialogs, as shown in the following image:



Create and Manage Workholding Groups

We have improved the creation and management of workholding groups in Mastercam 2026 so you can better organize your workholding components. A new drop-down list allows for more capable and easier grouping. This drop-down lets you create named groups and subgroups and add selected workholding components into these groups. You can also create empty groups to use while organizing your workholding group structure. If needed, you can leave groups empty.



3D Toolpath Enhancements

Listed below are enhancements to 3D milling toolpaths.

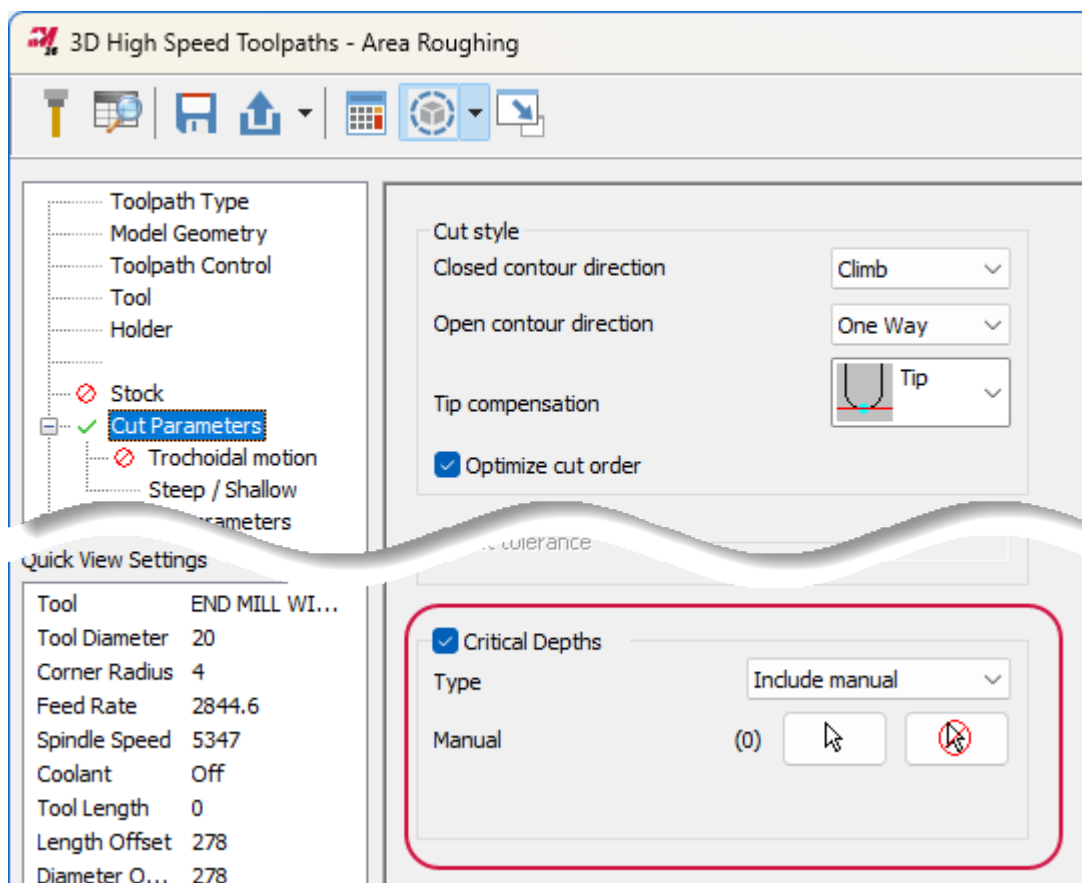
Control Flat-Area Machining with Critical Depths

Use **Critical Depths** to machine critical flat areas while performing simultaneous 3-axis machining. This new functionality has two options:

- **Include flats:** Allows Mastercam to automatically detect and machine flat areas
- **Include manual:** Lets you use the selection and deselection buttons to define the flat areas to machine

When creating a toolpath, **Critical Depths** gives you control over flat-area machining, eliminating the need to add additional flat-area operations at critical depths. Note that **Critical Depths** in a Dynamic OptiRough operation is only available for the stepdown move of the toolpath.

Find **Critical Depths** on the **Cut Parameters** page for Dynamic OptiRough and Area Roughing.



Hole-Making Toolpath Enhancements

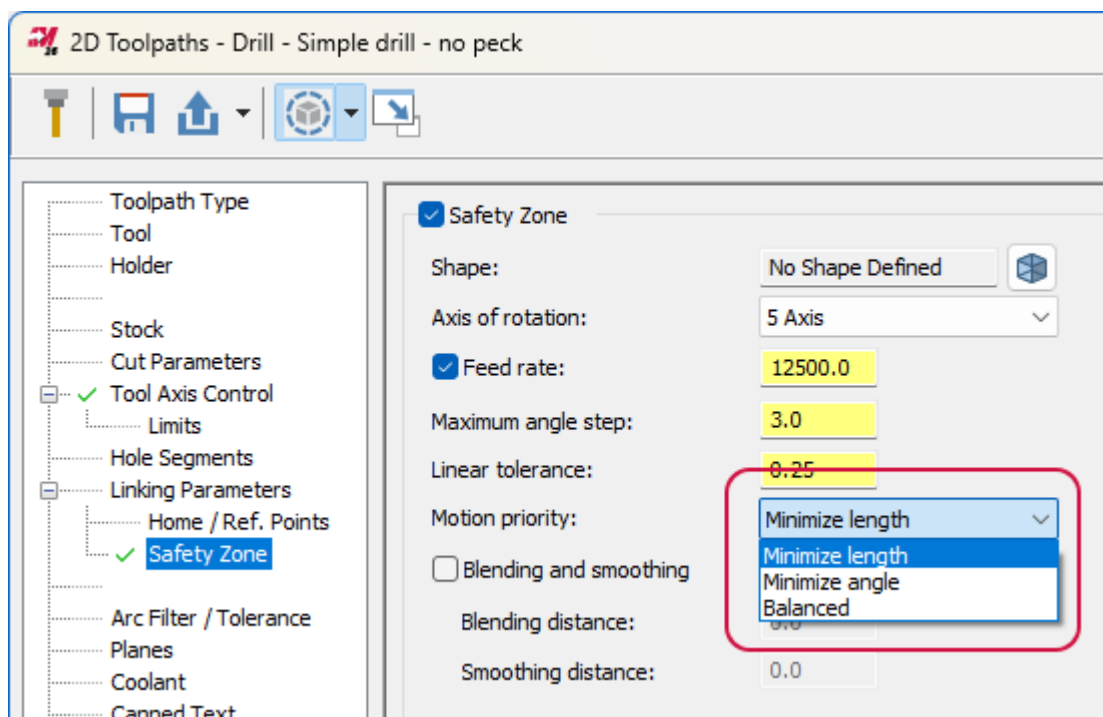
Listed below are enhancements to hole-making toolpaths.

Prioritize Safety Zone Motion

Previously, a mesh-based safety zone always selected the shortest path of the sample slices. However, this approach may result in undesirable paths that do not consider the angle change. To solve this, you can now also choose the path with the least angle change. Although many toolpaths use a safety zone, these changes are only for hole-making toolpaths.

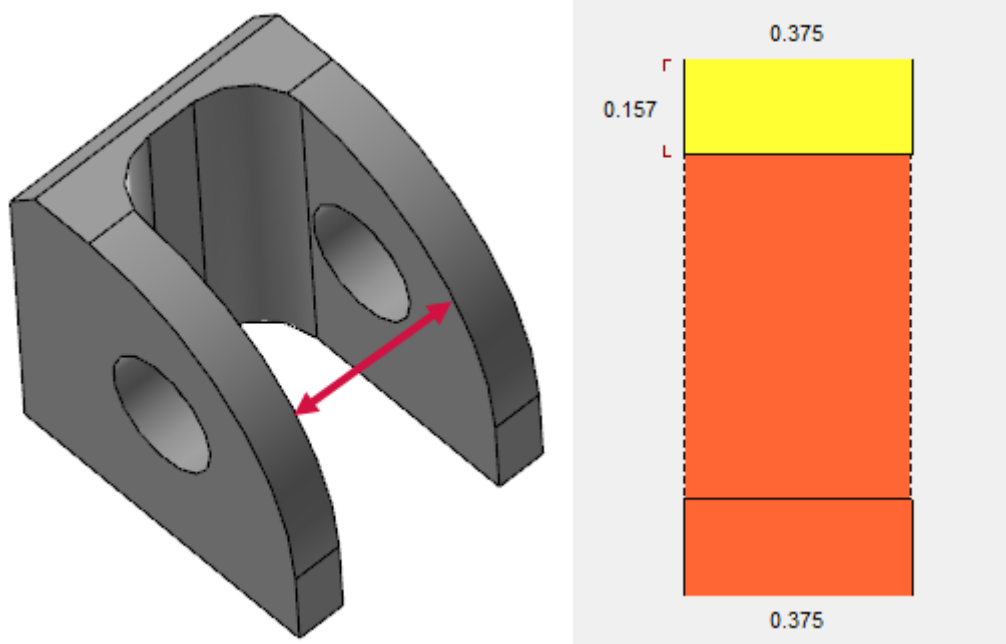
The following parameters provide three ways to prioritize safety zone motion:

- **Minimize angle:** Generates paths with the least amount of angular change
- **Minimize length:** Generates paths with the shortest distance between points
- **Balanced:** Generates paths that are balanced between the shortest distance and minimal angle change

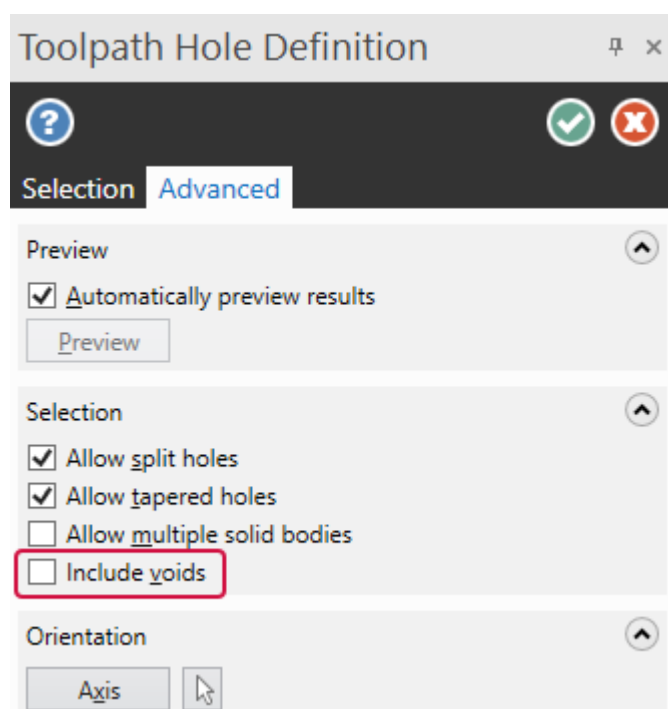


Void Support for Solid Holes

When you create a hole for a hole-making toolpath, the new **Include voids** checkbox provides selection control of the voids between hole segments. The resulting hole form includes hole segments that are collinear to the selected hole, in addition to the voids between them, comprising a single hole form. The red arrow in the first image indicates a void between hole segments. The second image shows how the hole displays in the **Hole Segments** page of the toolpath's parameters dialog box. The segment with the dashed lines is the void.



When the **Include voids** checkbox is unchecked, collinear holes with voids between them are considered individual hole forms. The following image shows the **Include voids** option in the **Toolpath Hole Definition** panel.



Multiaxis Toolpath Enhancements

Listed below are enhancements to multiaxis toolpaths.

Asymmetric Edge-Breaking Option for Deburr Toolpaths

Mastercam 2026 Multiaxis features a new asymmetric option for deburr toolpaths. **Allow asymmetric edge shape** allows for edge breaking at angles other than 45 degrees to reach undercut features. It applies to the following toolpath and tool types, as well as the machine types listed below:

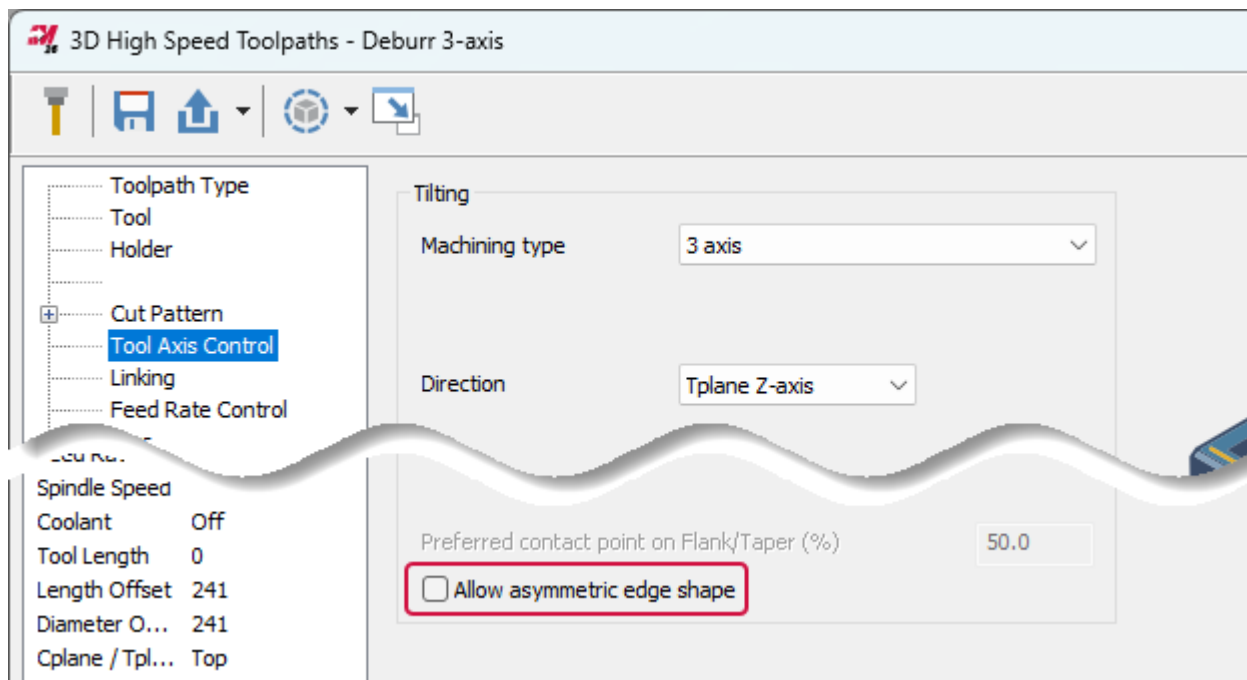
Toolpaths and tools:

- Deburr 3-axis
- Deburr 5-axis
- All Tool Types

Machines:

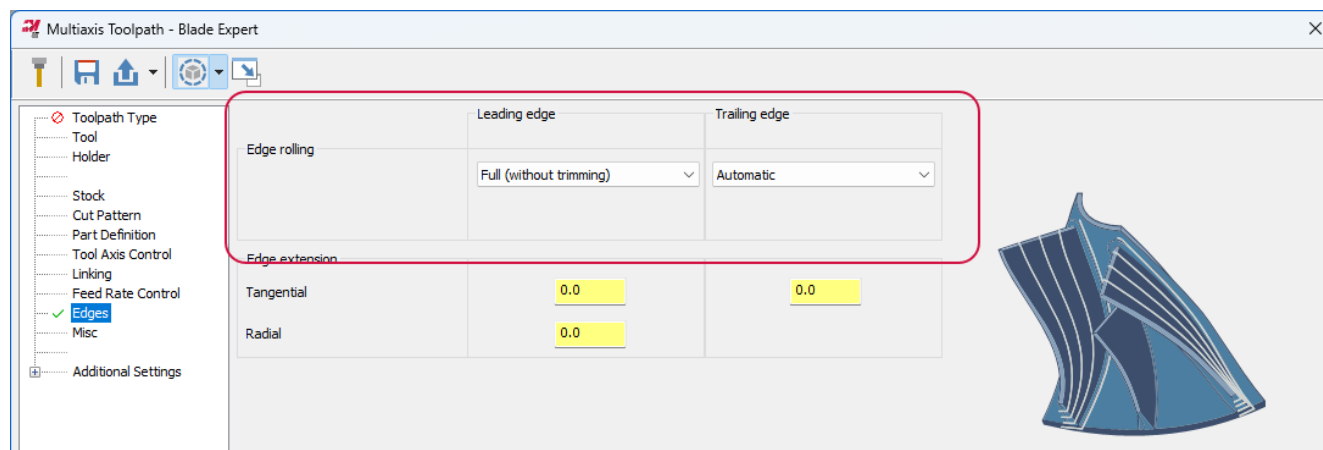
- 3 axis
- 4+1 axis
- 4 axis
- 3+2 axis

To select asymmetry, open the **Tool Axis Control** page in the toolpath's dialog box, and select **Allow asymmetric edge shape**.

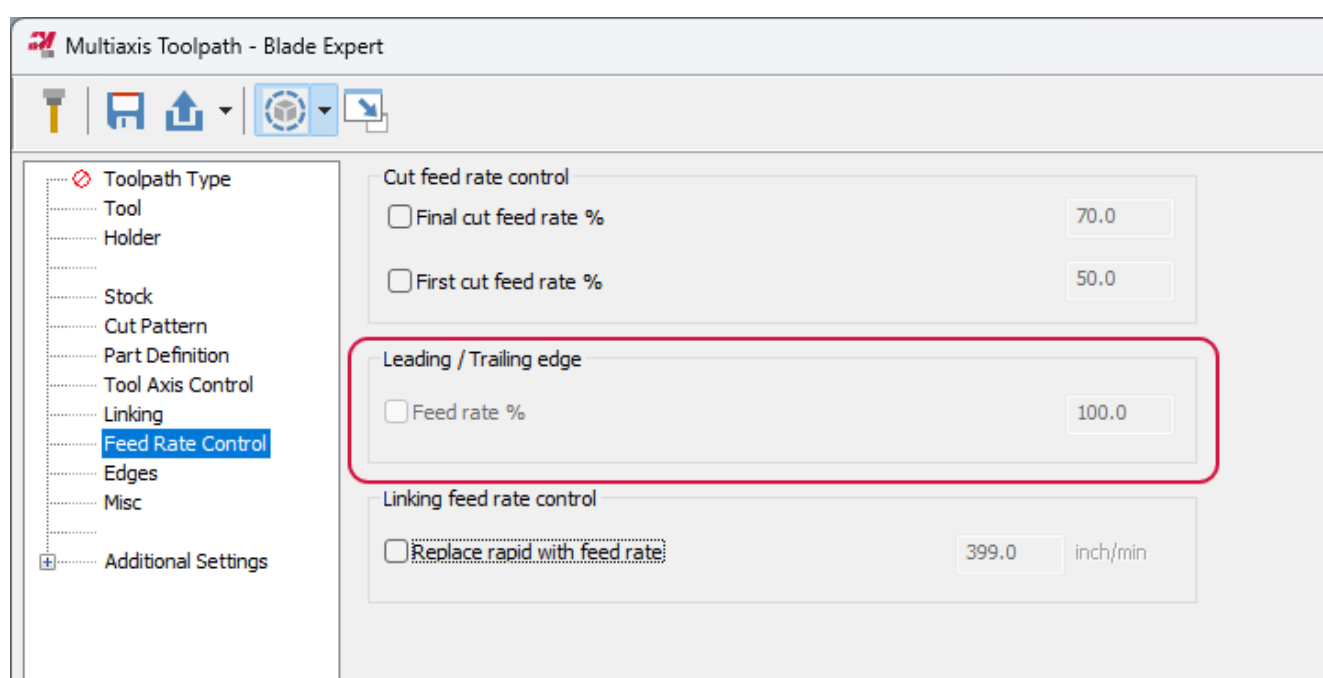


Enhanced Edge Rolling Control

In previous versions of Mastercam, any edge rolling definition you selected was applied to both leading and trailing edges. With this release, we have enhanced edge rolling in the Blade Expert toolpath to allow individual definitions for leading and trailing edges. Controlling the entry and exits allows for a much smoother finish and avoids possible gouges. Edge rolling controls are located on the **Blade Expert** dialog box under **Edges**.

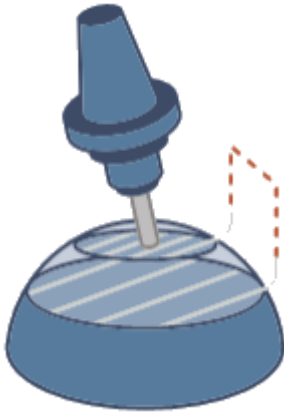


Additionally, you can now define separate feed rates for leading and trailing edges on the **Feed Rate Control** page in **Blade Expert**.

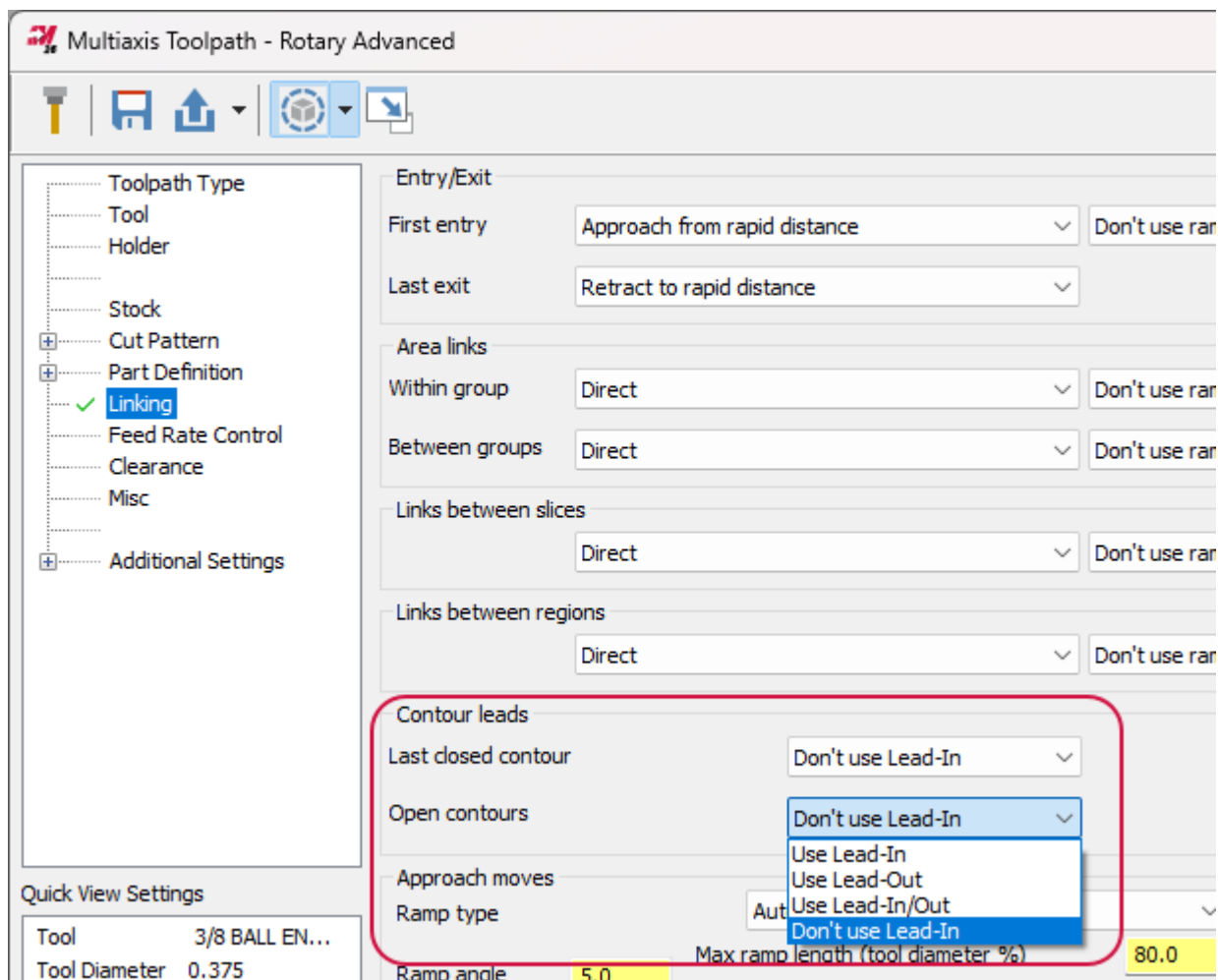


Contour Lead Enhancements for Rotary Advanced

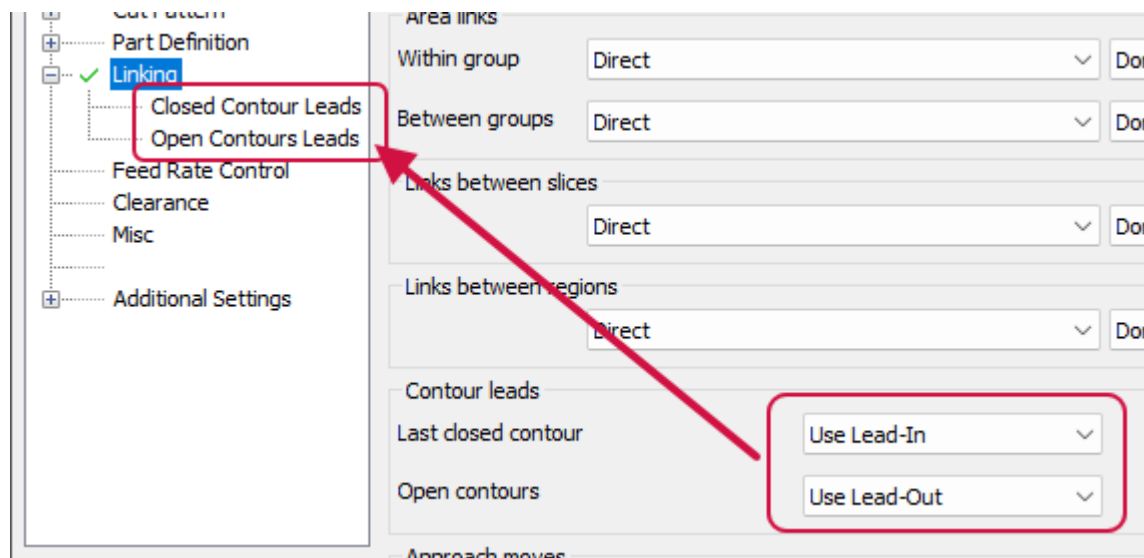
We have enhanced the Rotary Advanced toolpath to allow for lead in/out control between contours within the toolpath. Previously, control was only allowed between slices, regions, last exit, and first entry. The following image shows a lead in/out between two contours.



A new function group named **Contour leads** has been added to the **Linking** page of the toolpath's parameters dialog box, as shown in the image following. The added options for leads between contours are **Use Lead-In**, **Use Lead-Out**, **Use Lead-In/Out**, and **Don't use Lead-In**.

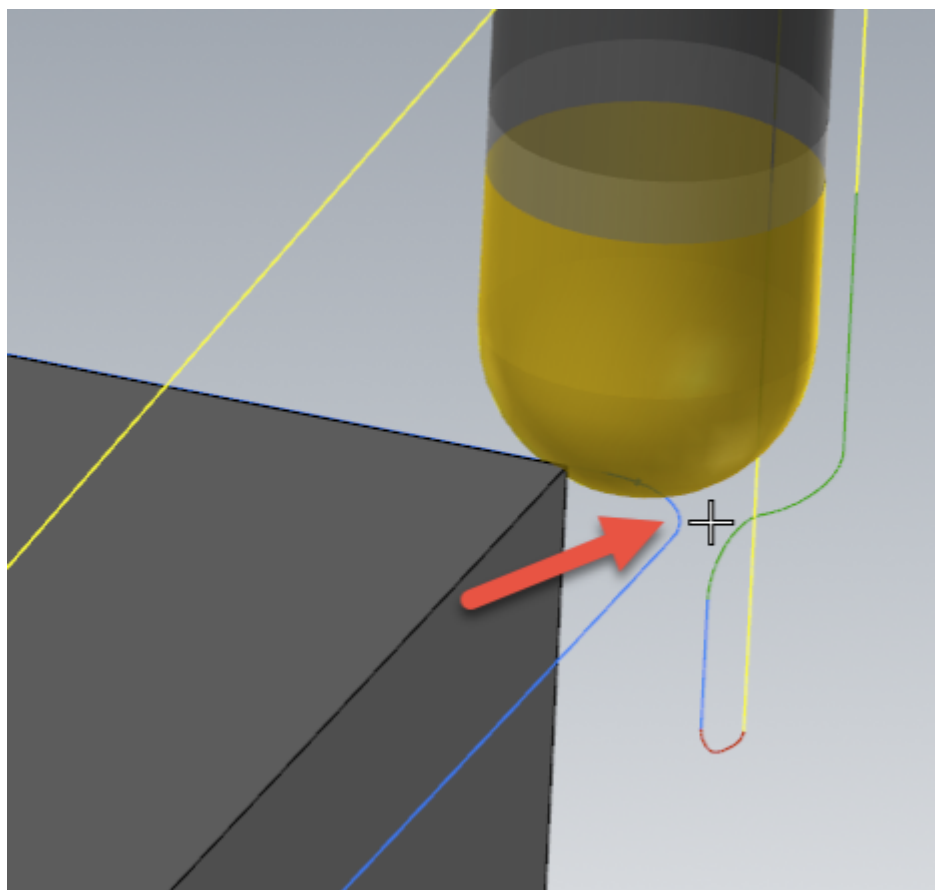


When you select an option other than **Don't use Lead-In**, new pages appear in the **Linking** section of the properties dialog box, as shown in the image following. These pages contain options for the fine control of tool movement between the toolpath contours.



Corner Arc Support for Deburr

When you used a chamfer tool in previous versions of Mastercam, the tool would not stay in a constant cut around some internal and external corners. Instead, the tool would retract and reenter on the new edge. Now in Mastercam 2026, a connecting move is automatically added to move the tool around corners without unnecessary retracts.

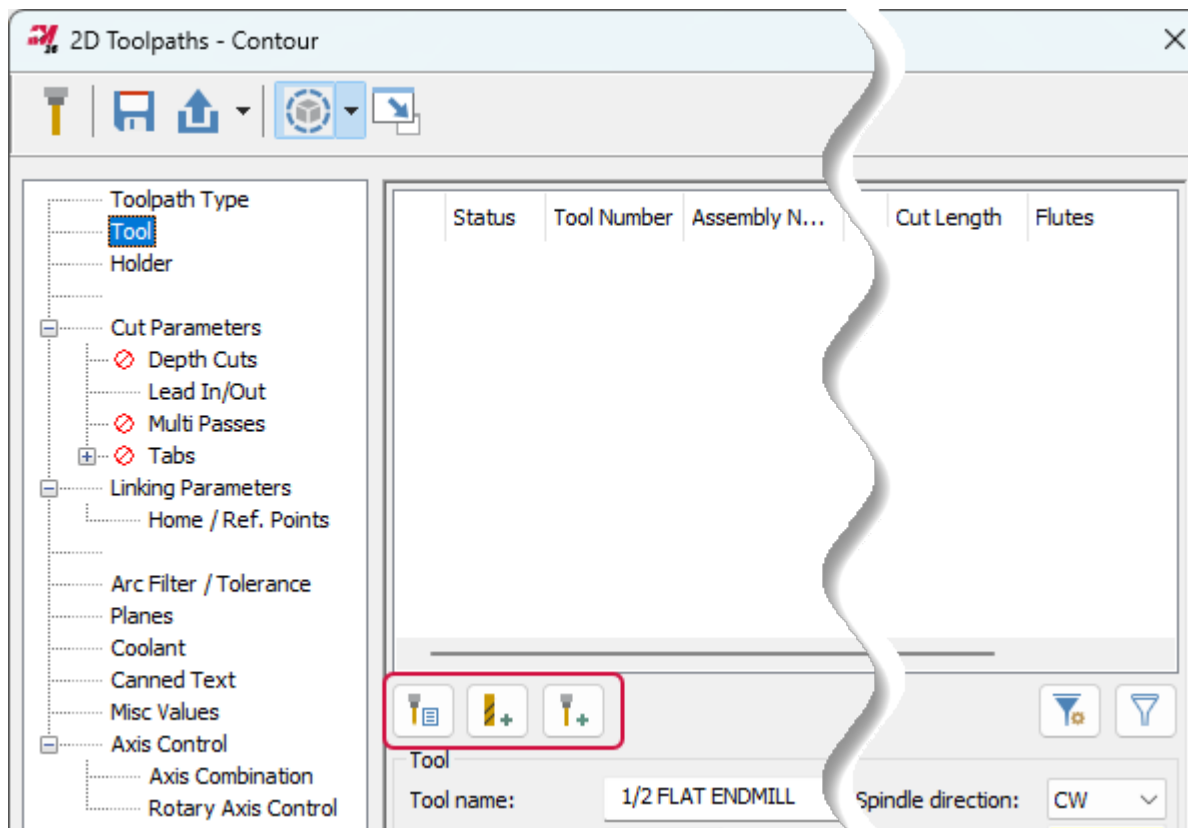


Tooling Enhancements

Listed below are major enhancements to tools and related features.

Creating New Tools and Assemblies

Create new tool and **Create new tool assembly** buttons have been added to the **Tool** page for all milling toolpaths. To accommodate the new buttons, tool-related commands are on the left side of the dialog box, as shown in the following image, while filter buttons are now on the right side. The new buttons represent commands also found in the right-click menu. These buttons reduce the number of mouse clicks needed to create a new tool or new tool assembly.



New Interface for Creating Tools and Holders

Mastercam 2026 features a redesigned interface for creating Mill tools and holders. The old wizard-style dialog boxes have been replaced by modern function panels similar to the Lathe 3D **Tool Designer**. Additional enhancements streamline your workflow even further. Use the new interface to create all types of tools for Mill and Router:

- 3D tools with component models for accurate simulation
- Simpler tools with parametric definitions that do not require solid models

The Mill Tool Designer organizes your tool settings in four pages.

Mill Tool Designer

Mill Tool Designer

Tool

Tool

Name: 10 Flat endmill

Description:

Type: Milling - Flat endmill

Use the **Tool** page to tell Mastercam about the tool and to select a tool **Type**.

Tool Assembly > Tool

Tool

Name: CoroMill® 745 face milling cutter

Description: CoroMill® 745 face milling cutter

Type: Milling - Face mill

Tool number: 1

Length offset: 1

Diameter offset: 1

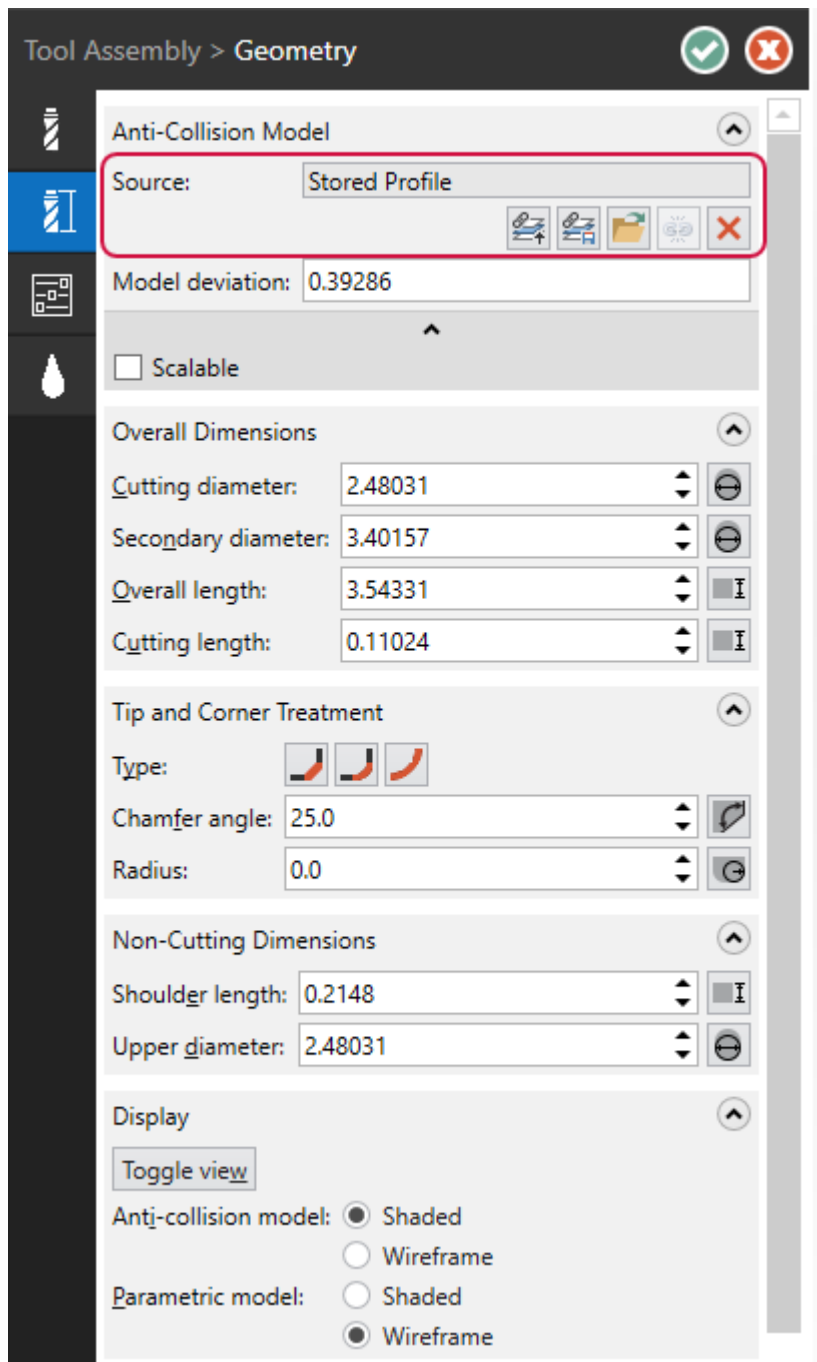
Head number: 0

Manufacturer:

Manufacturer's code: 725-063C6-21M

Grade:

The layout of the **Geometry** page is determined by the tool **Type** that you select. Use the controls at the top of the page to select and manage the solid model that will represent the tool for simulation and collision checking.



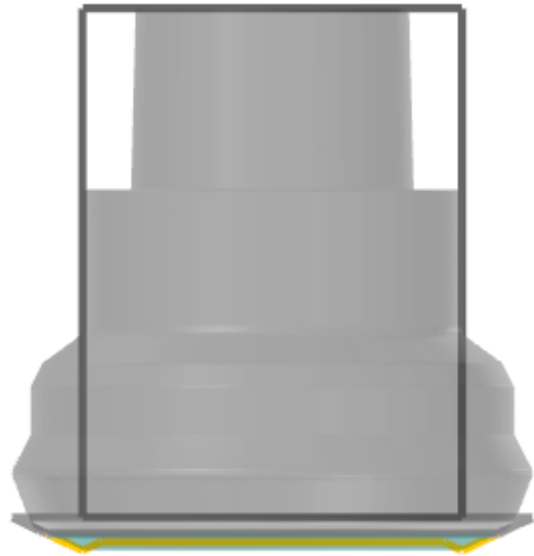
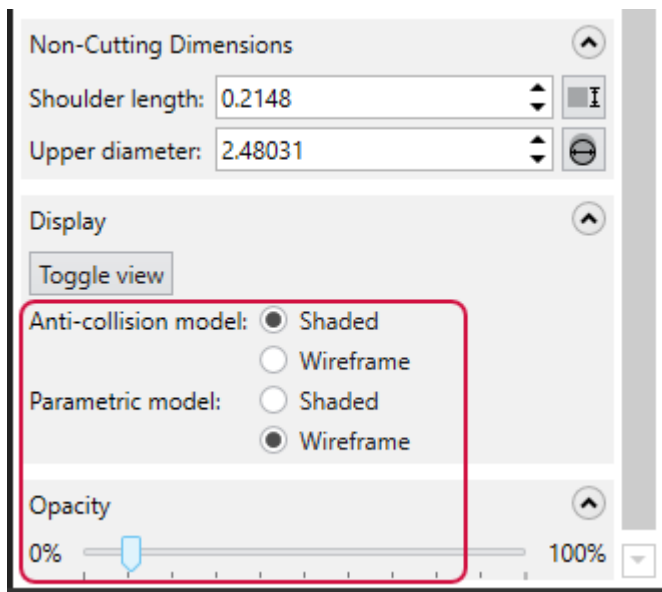
Mastercam maintains two different models of the tool:

- The anti-collision model is based on the imported solid model. Mastercam uses this model for simulation and collision checking.
- The parametric model is based on the tool dimensions as shown in this panel. Mastercam uses this model when calculating toolpaths.

If you import a solid model, Mastercam fills in the dimensions automatically based on the model. This ensures that the two models are as close as possible. However, you can edit or fine-tune the dimensions as required to improve your toolpath results. The **Model deviation** field displays the maximum deviation between the two.

You are not required to import a solid model. In this case, enter the dimensions manually and Mastercam will build the parametric model from them. For simulation and collision checking, Mastercam uses a revolved profile.

The new interface makes it easy to visually compare the two models. You can choose to display each model as wireframe or as a shaded solid. Use the **Opacity** slider to adjust the translucency so that you can see them clearly on top of each other.



Use the **Parameters** and **Coolant** pages to set default feeds, speeds, and coolant options similar to previous versions.

The new Mill Holder Designer presents a similar workflow for designing and managing tool holders.

Mill Holder Designer

?

✓

✗

Component

Component

Name:

Coromant Capto® reduction adaptor

Description:

Manufacturer:

Catalog ID:

C10-391.02-63 055

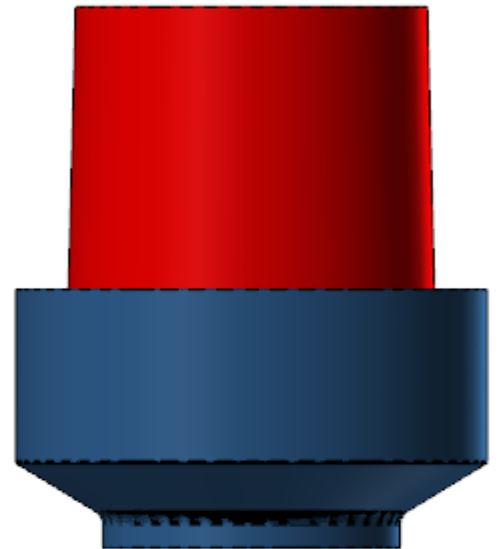
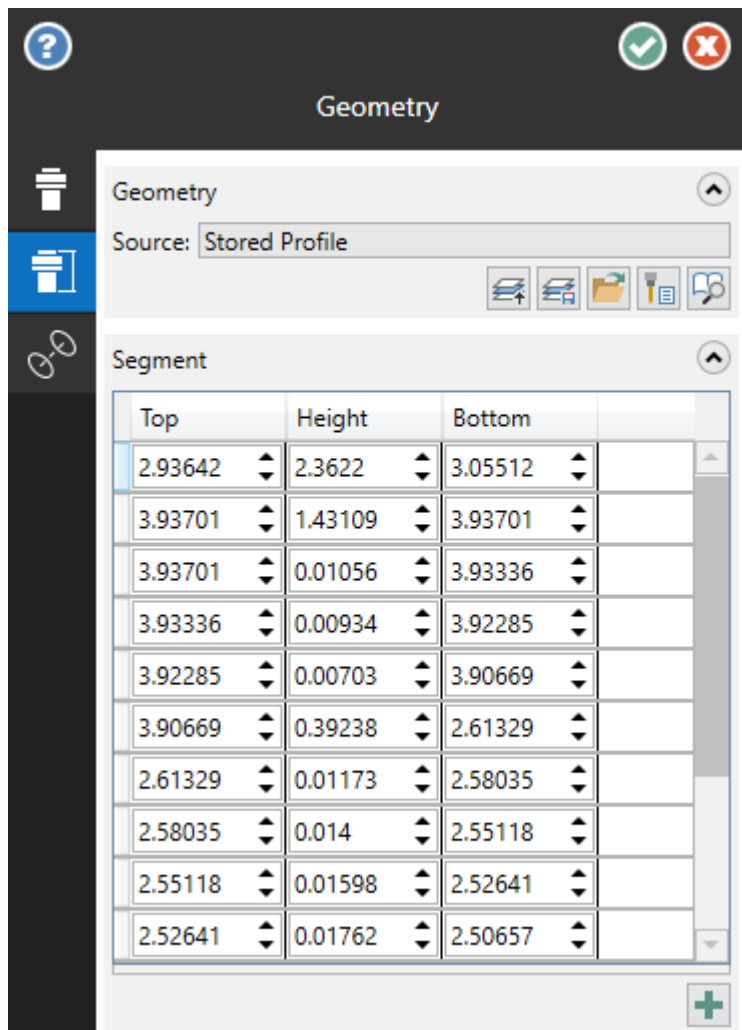
☐ Thru-tool coolant

Units

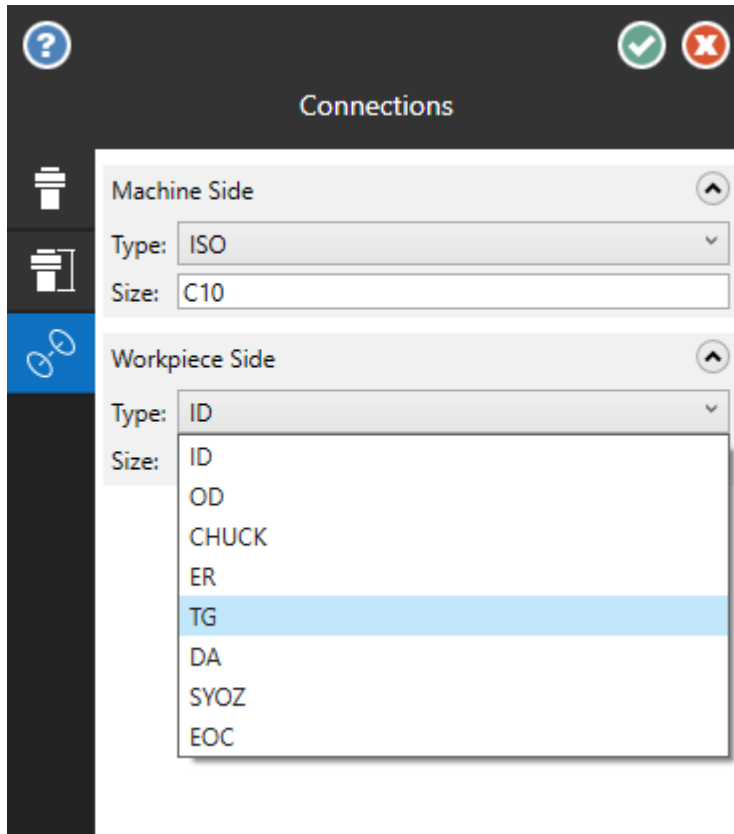
☒ U.S.

☐ Metric

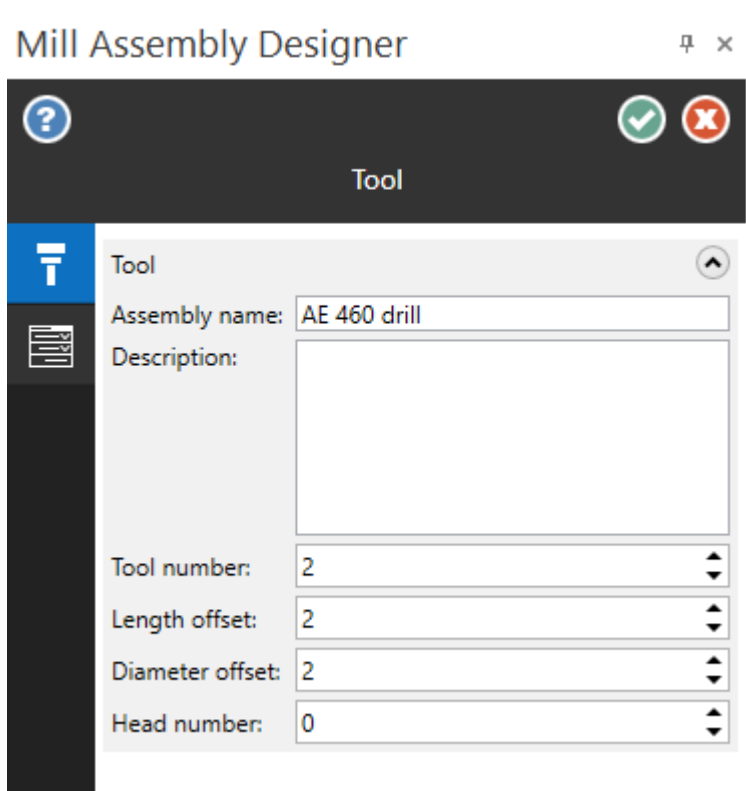
Use the **Geometry** controls to select and manage solid component models similar to the **Tool Designer** panel. The holder profile is represented as sections organized in a grid. Edit the dimensions of each section as desired, or right-click in the first column of the grid to see additional options.



Use the **Connections** page to select from industry-standard sizes and types for each end of the holder.

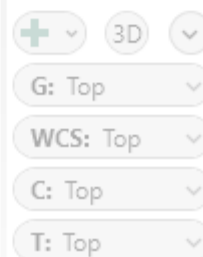
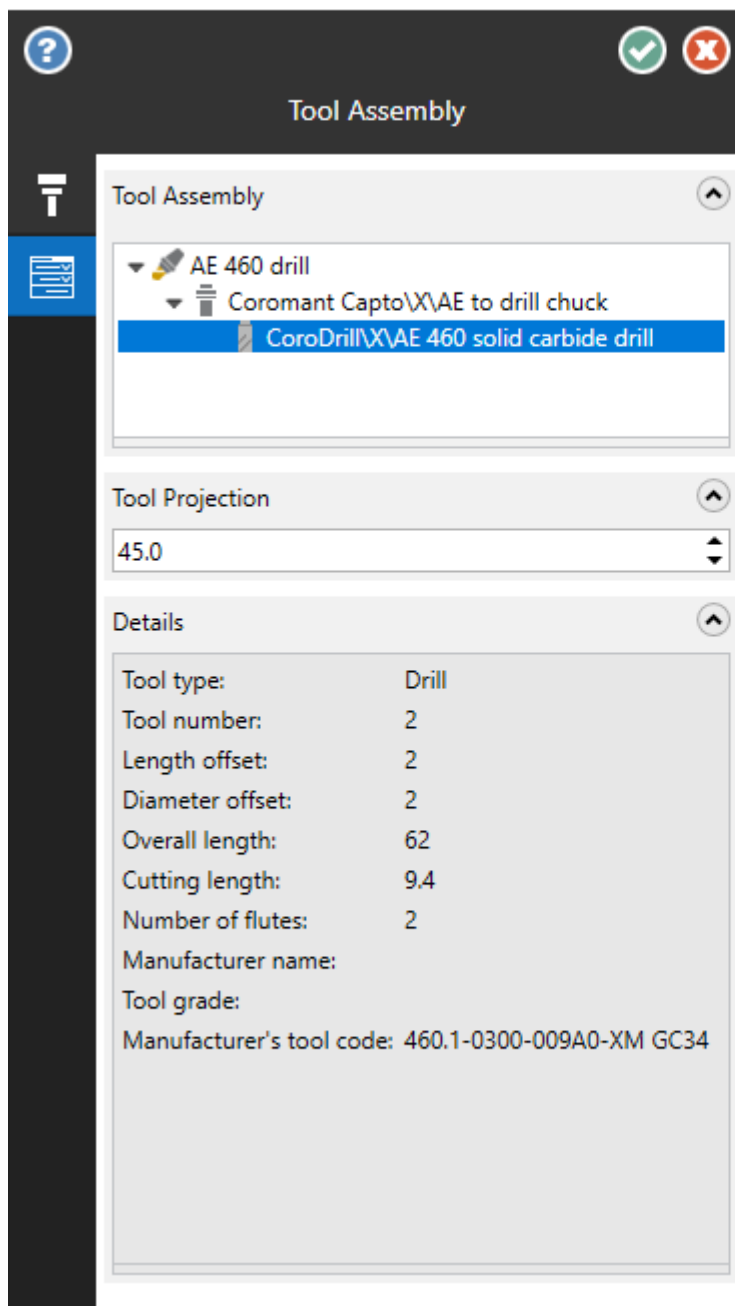


Finally, Mill tool assemblies feature additional properties that let you manage them more easily. New properties include default tool and offset numbers that Mastercam can apply when you select the assembly for an operation. The default tool number associated with the tool definition remains unchanged. This lets you create as many assemblies with the tool as you wish; each assembly can have its own tool number, independent of the number in the tool definition.



A **Tool Projection** field has also been added. This lets you easily enter a known projection value from a blueprint or data sheet instead of dragging the tool in the graphics window. If you wish, you can continue to set the projection for the tool or holder components by dragging the component in the graphics window.

Mill Assembly Designer



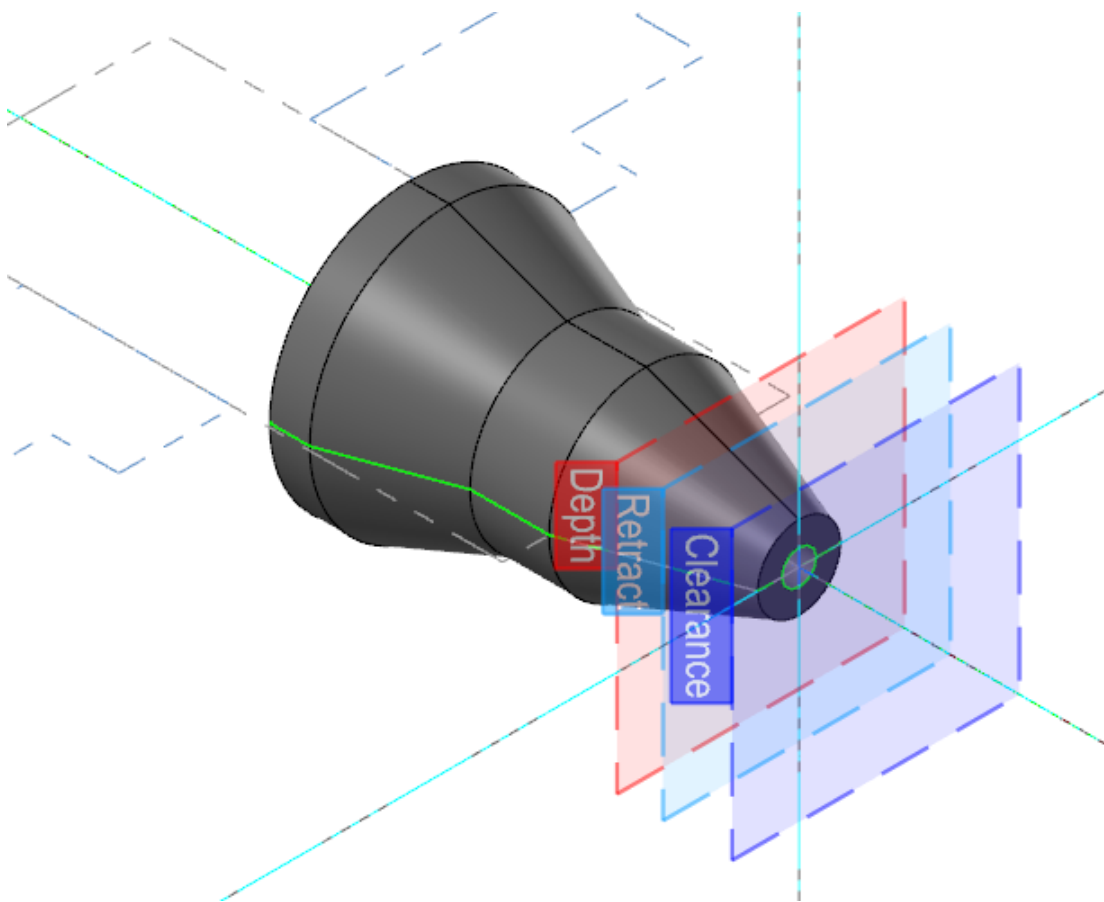
Turning Enhancements

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

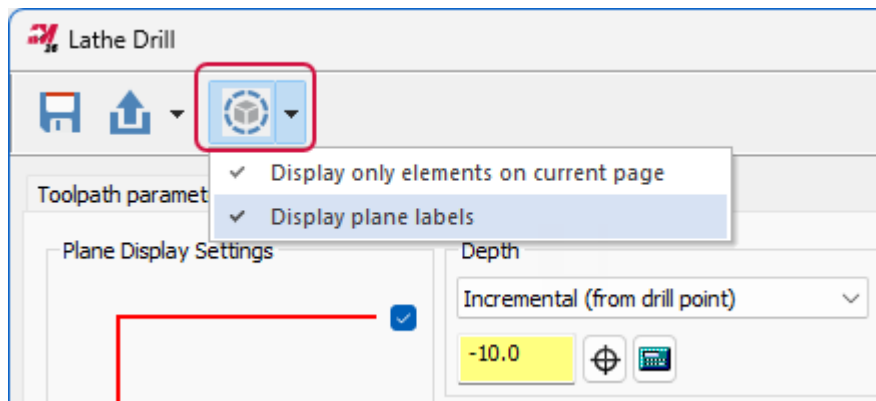
 **NOTE:**
Unless otherwise stated, the new features and functionality listed in this section apply to both Lathe and Mill-Turn licenses.

Graphical Planes Display for Lathe Drill

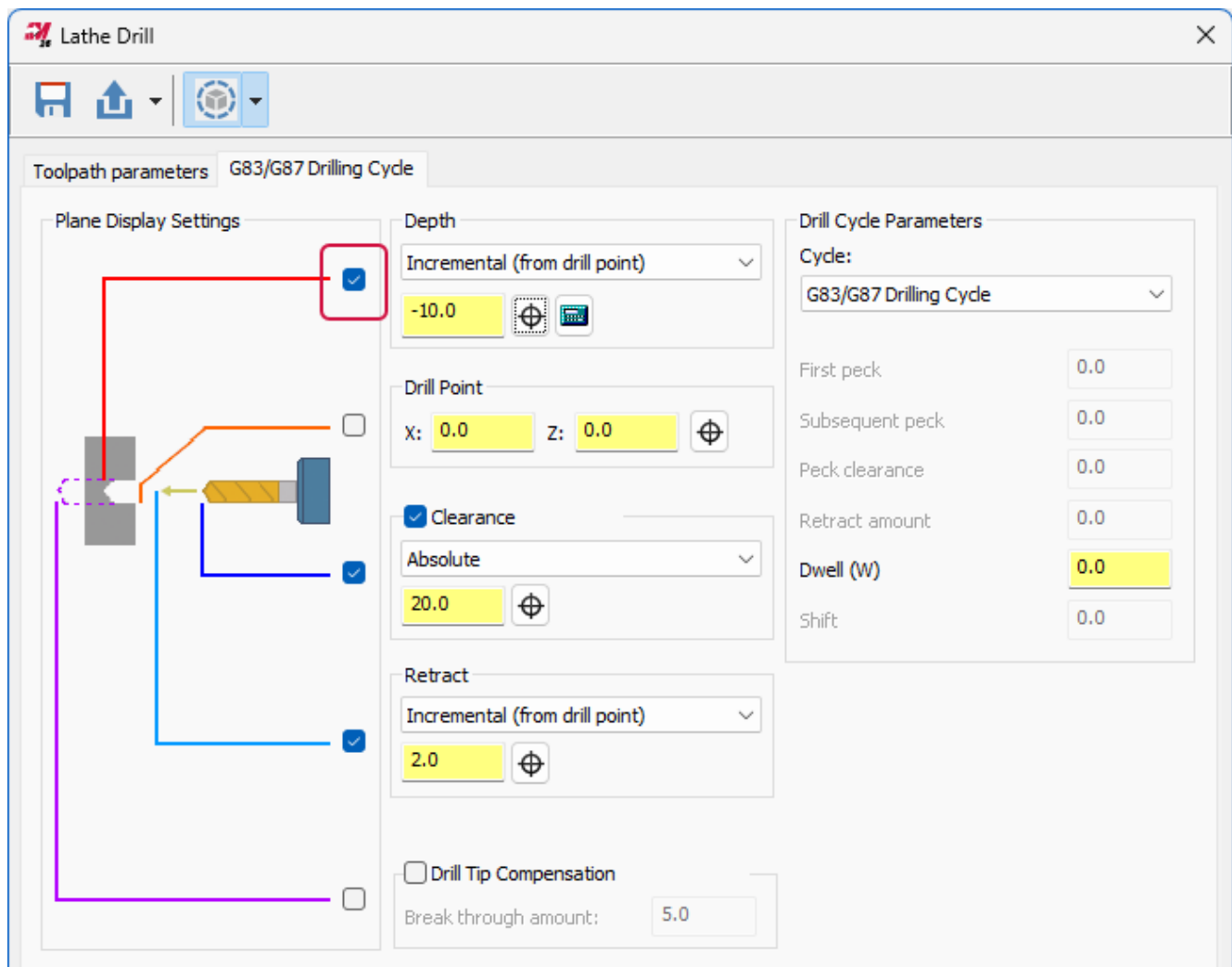
Lathe Drill operations now graphically display planes, such as clearance and retract heights, drill depth, and more. This is available for both Lathe and Mill-Turn users.



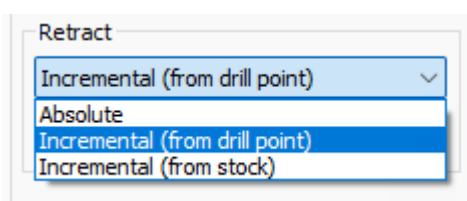
This feature works similarly in Lathe and Mill. Use the button on the toolbar to toggle the plane display and configure what is shown.



Select each plane that you wish to display by selecting it in your toolpath settings. In this example, the **Depth**, **Clearance**, and **Retract** planes are selected.



In addition, the layout of the dialog box has been updated and streamlined. For each plane, the **Incremental** and **Absolute** options have been consolidated into a single drop-down list:

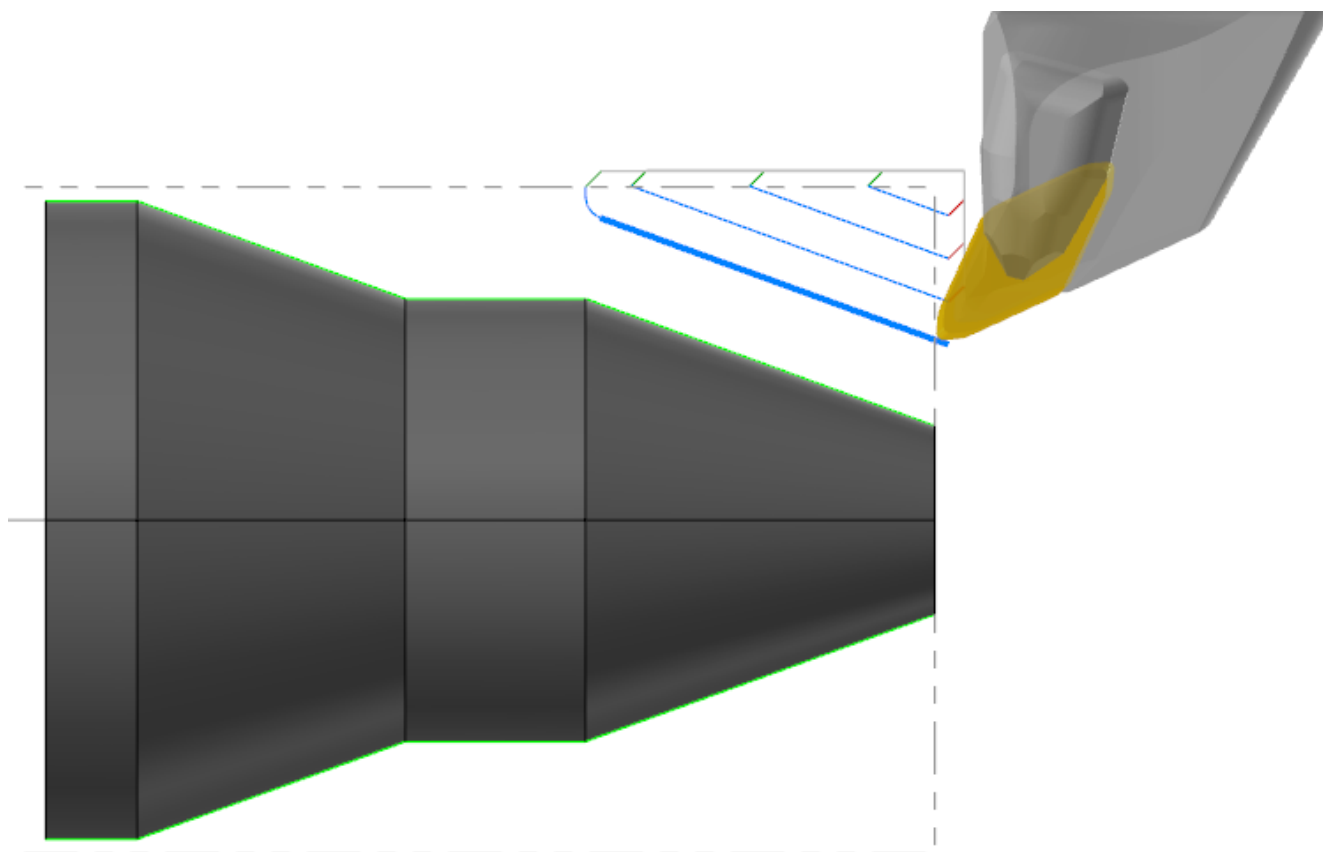


Enhancements to PrimeTurning™

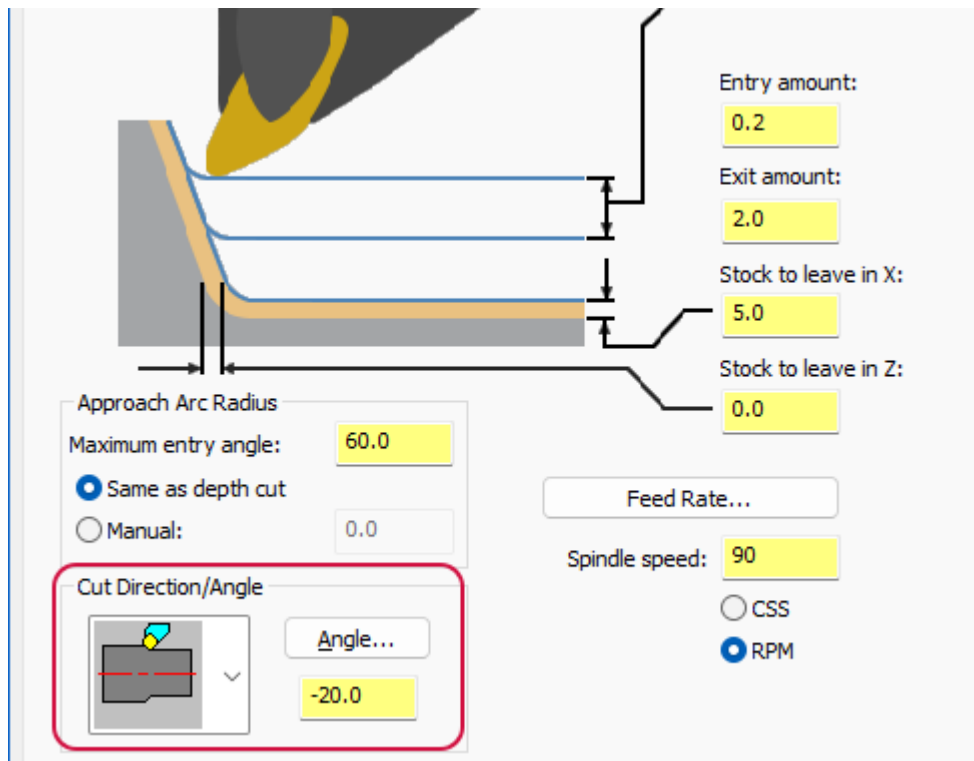
A number of enhancements have been made to PrimeTurning. These extend the range of applications in which this popular toolpath can be used and improve the overall workflow.

Angled cuts for PrimeTurning

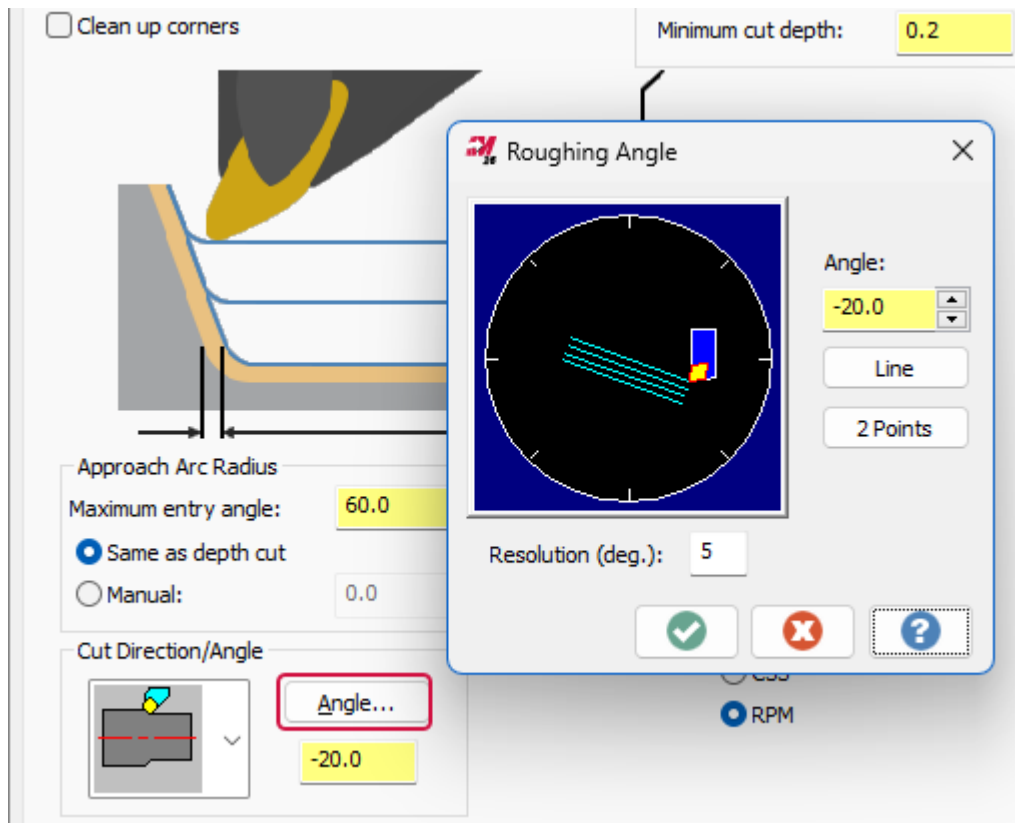
PrimeTurning now lets you create roughing cuts at an angle. This means that you can create cuts that are parallel to angled surfaces on your part, leading to much easier machining. For example, this picture shows the roughing cuts angled at -20 degrees:



Specify the angle on the **Rough parameters** tab for your PrimeTurning toolpath.

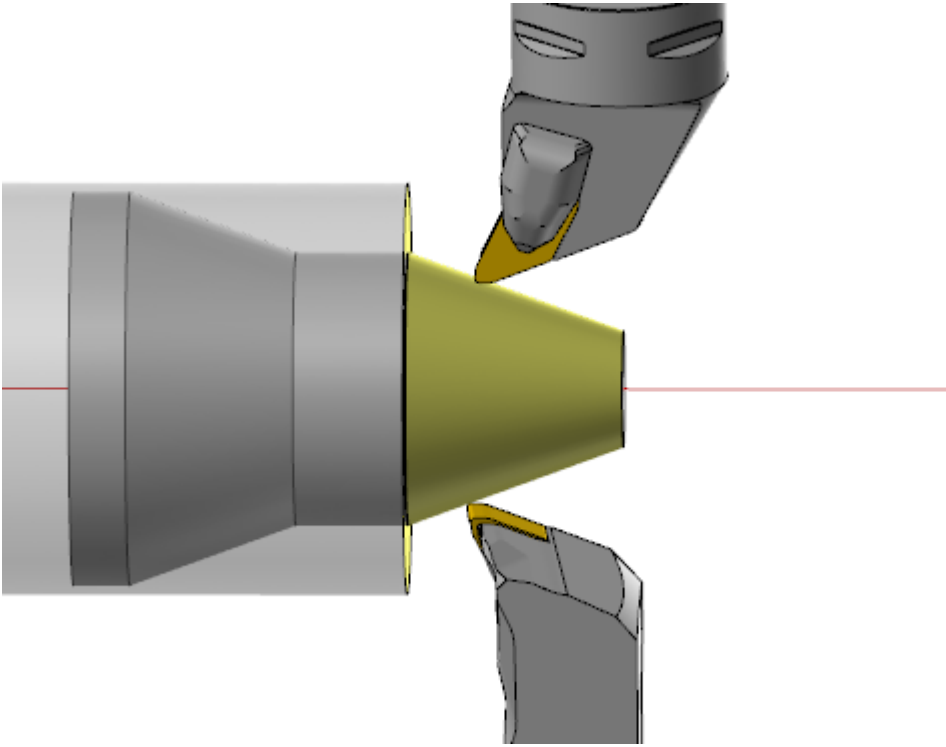


Click the **Angle** button to set the angle from a dial or by selecting geometry in the graphics window.

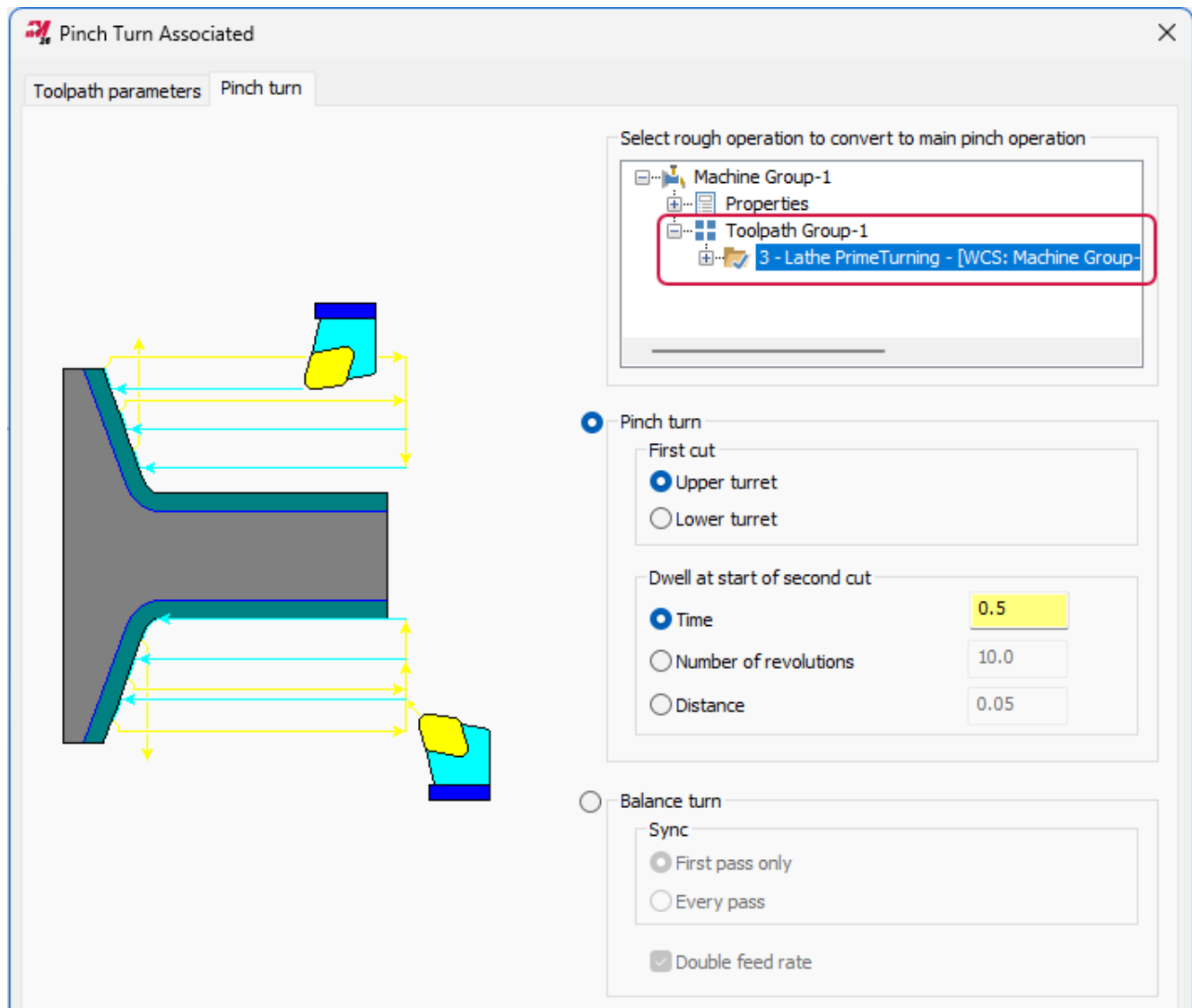


Pinch turning supported for PrimeTurning

PrimeTurning toolpaths can now be selected for use in pinch turning and balanced turning operations.



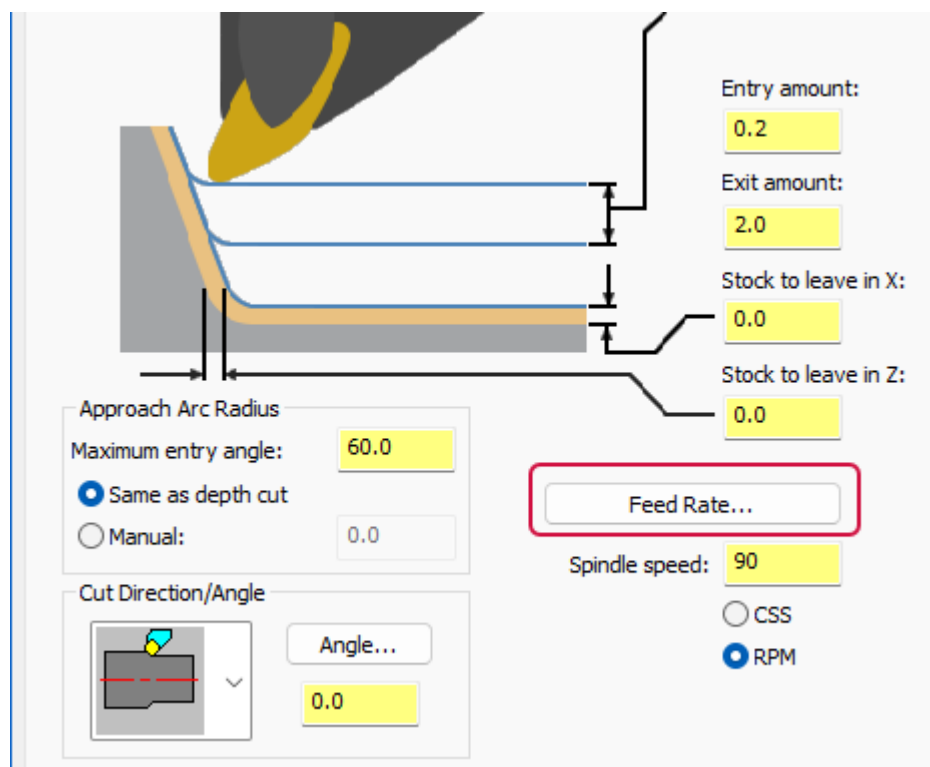
Simply select the PrimeTurning operation in the same way as any other toolpath when selecting the main pinch turn operation. Both **Pinch turn** and **Balance turn** modes are supported.



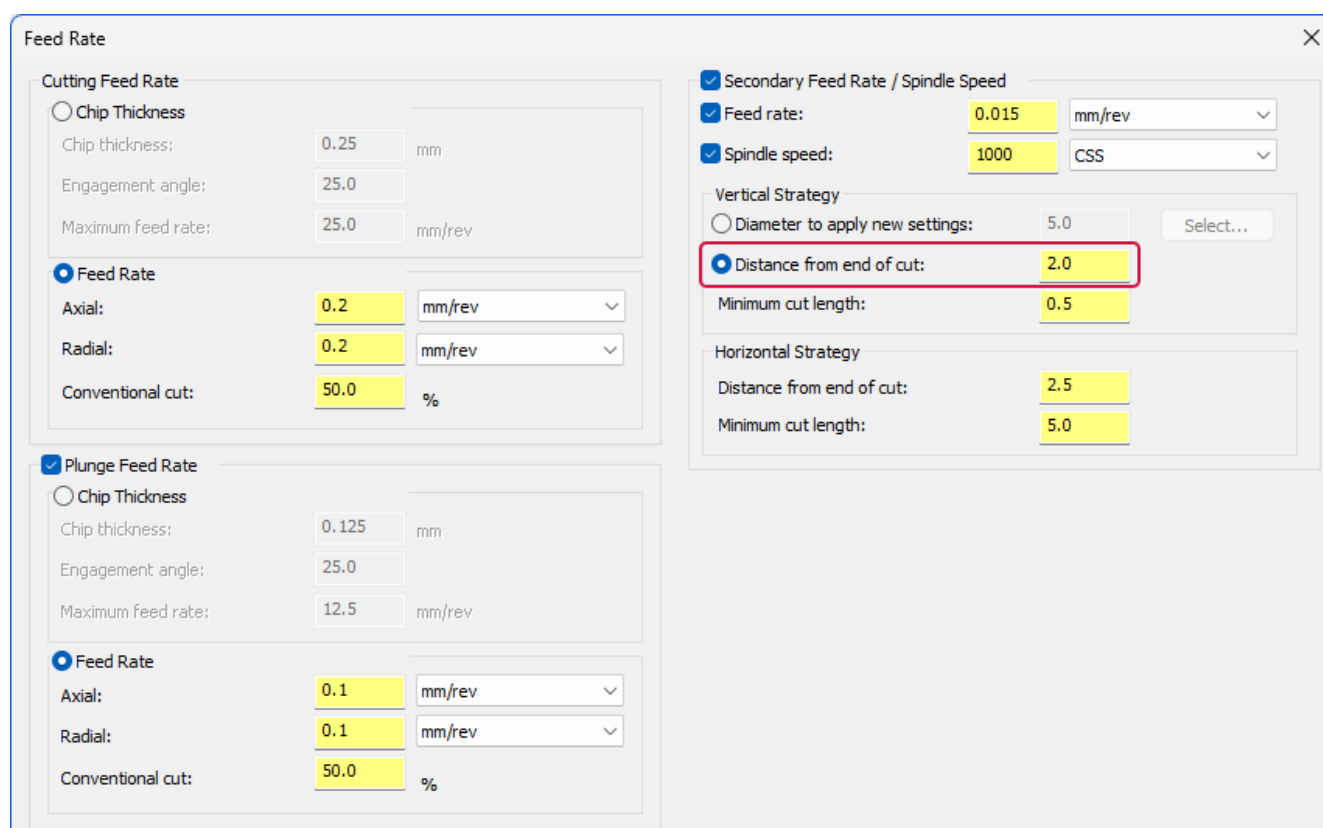
Adjusting the feed rate at the end of the cut

PrimeTurning includes an option that lets you adjust the feedrate at a set distance before the end of the cut. Before Mastercam 2026 this ability was limited to horizontal cuts; Mastercam 2026 now lets you do this for vertical cuts as well.

To do this, click the **Feed Rate** button on the **Rough parameters** tab for your PrimeTurning toolpath.

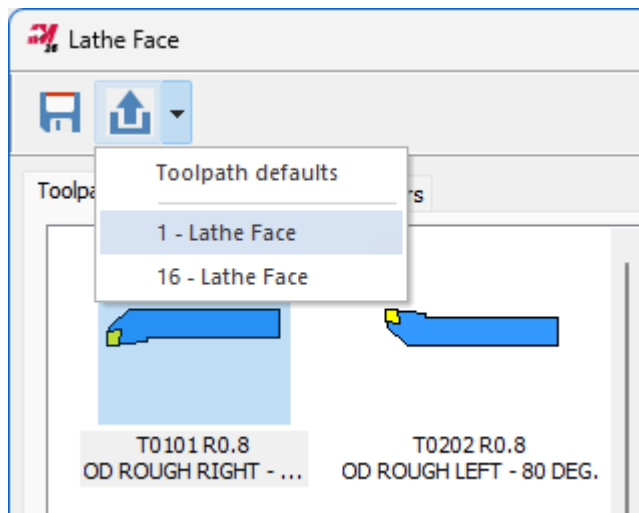


The new option is on the **Feed Rate** dialog box as shown. Notice that the layout of the dialog box has also been updated.



Support for Saving and Loading Parameters in Turning Toolpaths

Turning toolpaths for both Lathe and Mill-Turn now can save and load toolpath parameters using toolbar buttons:



- Click the left button to save the current toolpath settings to the defaults file (*.mcam-defaults).
- Click the right button to open the **Load Toolpath Settings** panel, which lets you select the source of the settings to load—either the defaults file (*.mcam-defaults) or an operations library (*.mcam-operations) that you select.

Load Toolpath Settings

⌵
✕

?

✓
✕

Operation Library ⌵

☒ Default
☐ Custom

Library: 📁

Operation Selection ⌵

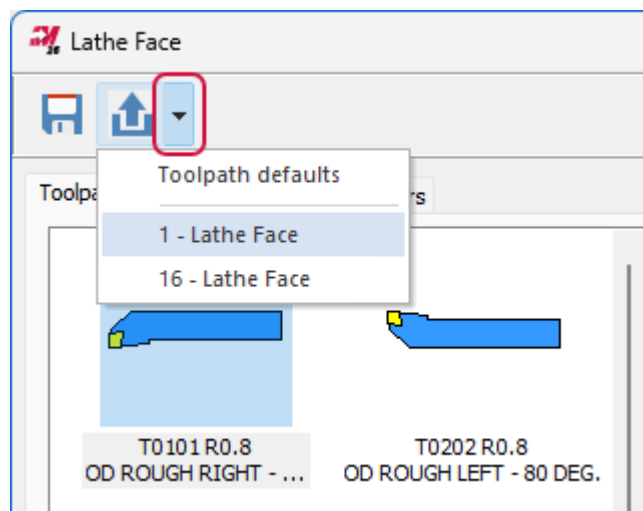
Machine Group: ⌵

Toolpath Group: ⌵

Operation:

ID	Name	Type
1	Lathe Face	Lathe Face
16	Lathe Face	Lathe Face

Click the arrow next to the right button to display the flyout menu for quick access to the current libraries.



- Select **Toolpath defaults** to load the toolpath settings from the active defaults file.
- The other options list the available toolpaths in the current operations library (*.mcam-operations). Select the desired toolpath from the list to load its settings.

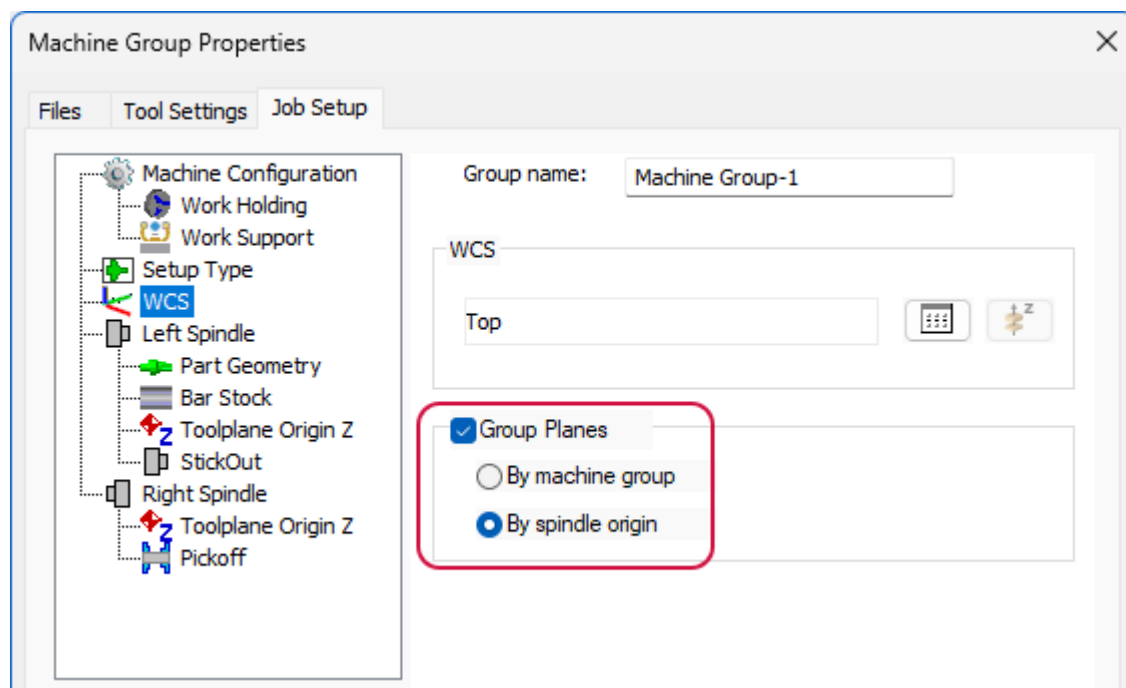
These buttons replace the context menu options that were used in earlier versions of Mastercam. These are no longer available.

Job Setup Improvements

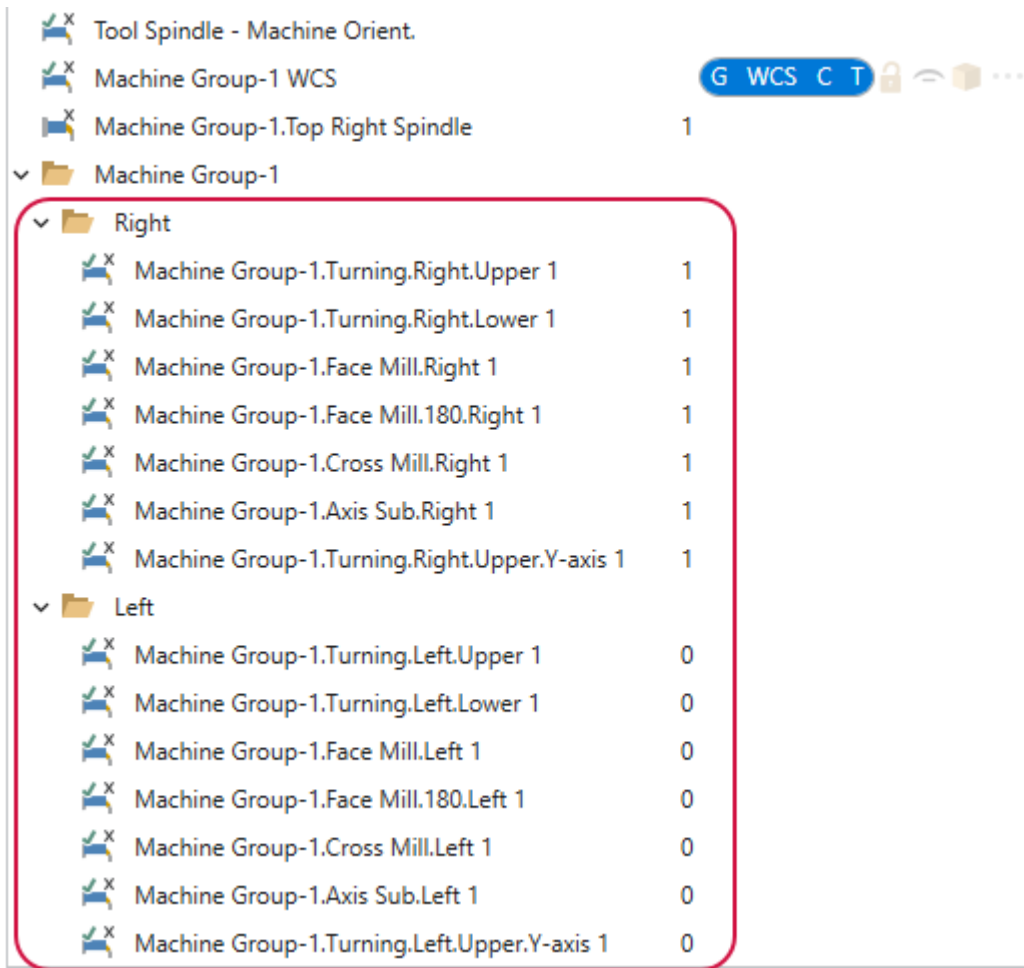
Several conveniences have been added to Job Setup to streamline the workflow for Mill-Turn users.

Grouping Planes in Job Setup

As part of the Job Setup process, Mastercam creates a number of planes for use in programming your part. Mill-Turn users can now tell Mastercam to organize these planes into groups. New options on the **WCS** page let you organize the planes by machine group or spindle origin:

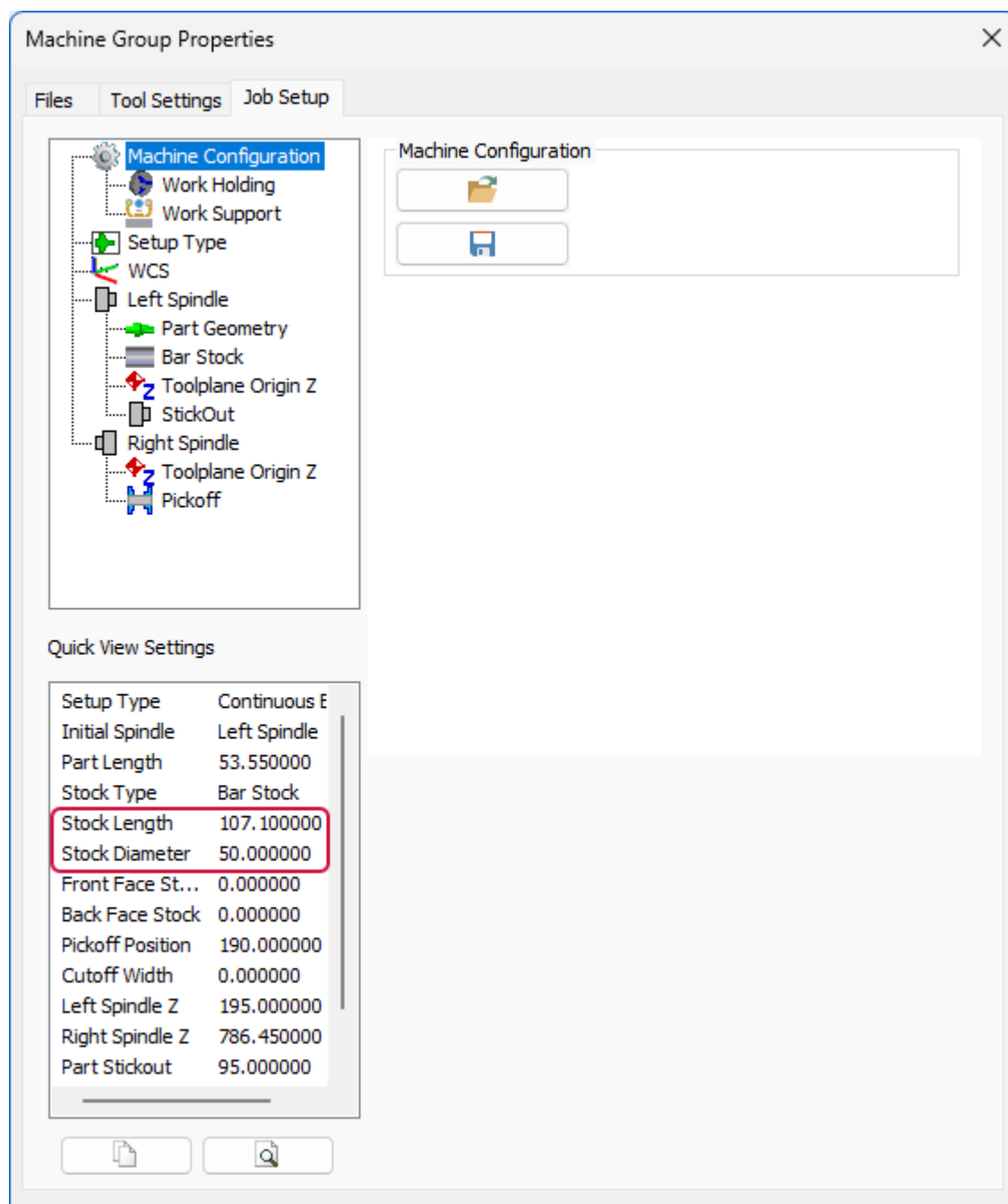


You can see the groups in the Planes Manager when you exit Job Setup:



Improved Quick View Settings

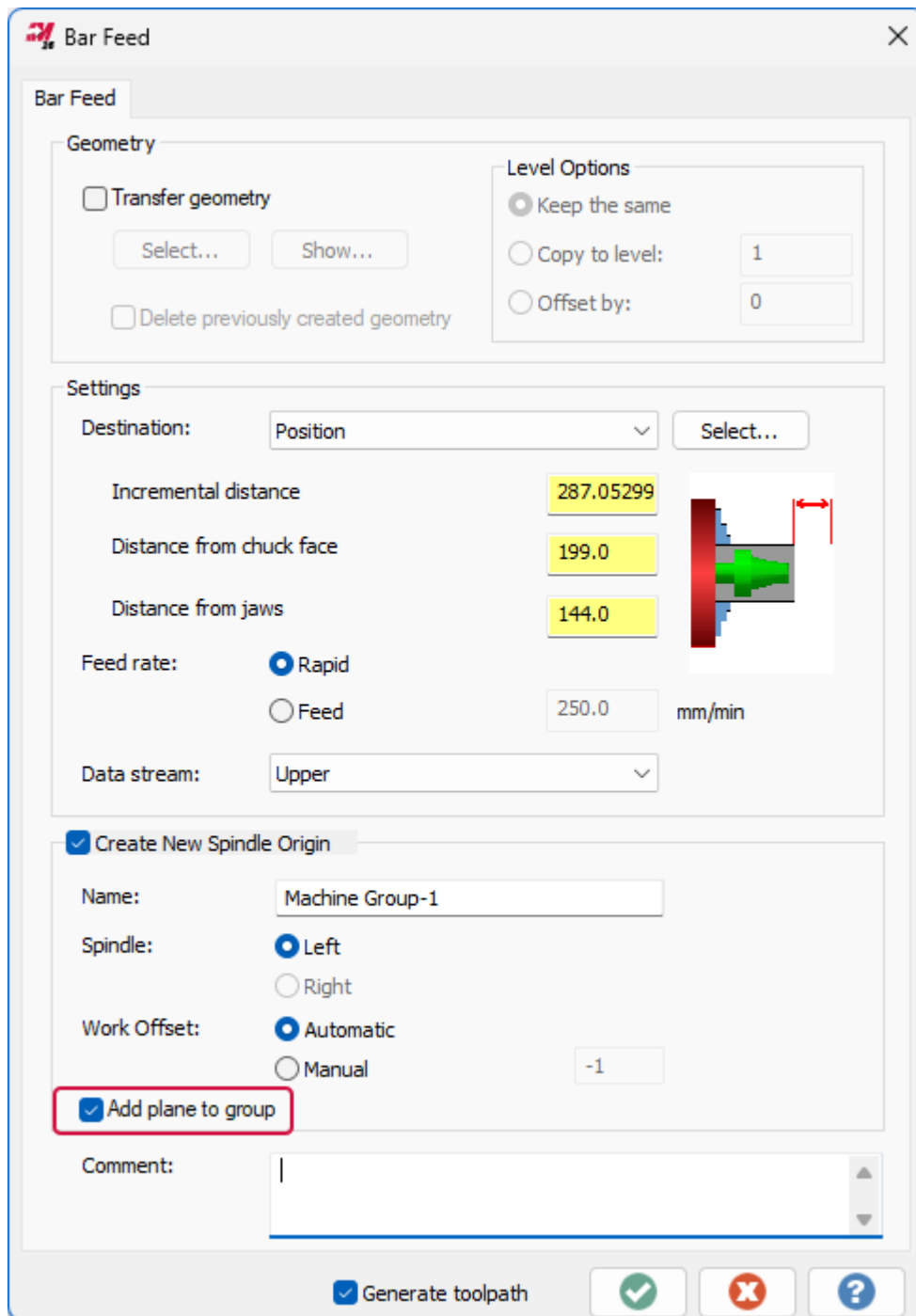
As a convenience to our Mill-Turn users, the stock length and diameter have been added to the **Quick View Settings** display.



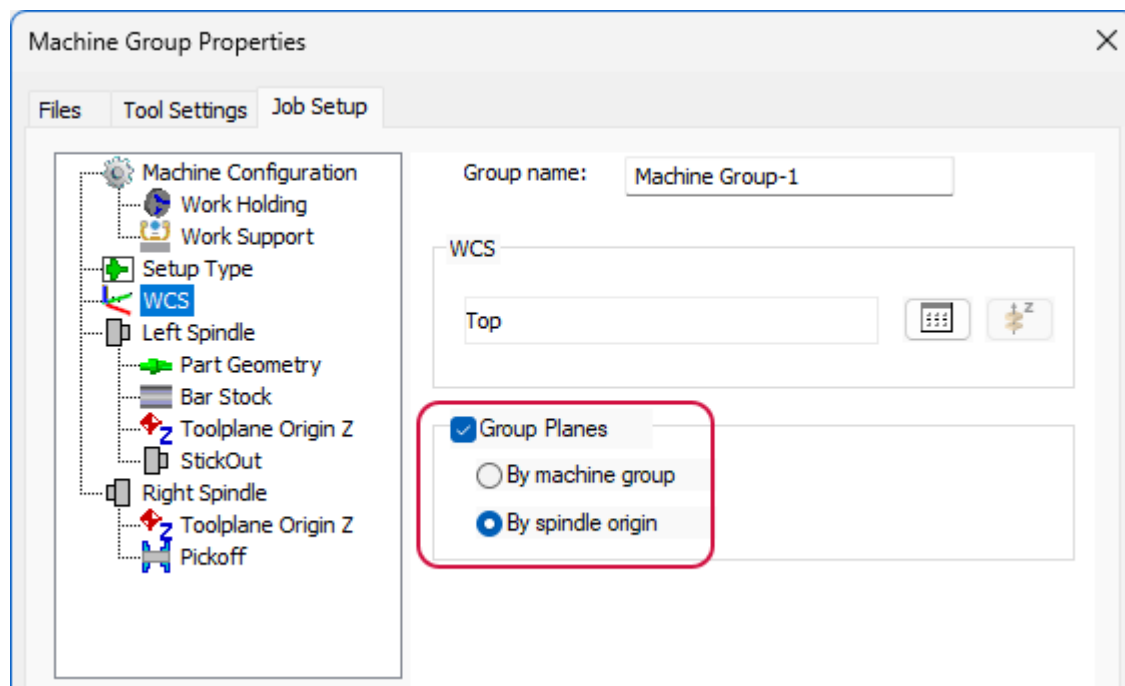
Improvements to Part Handling Operations

Mill-Turn users can now add the planes and spindle origins that are created by spindle move and bar feed operations to the same groups as the ones created by Job Setup.

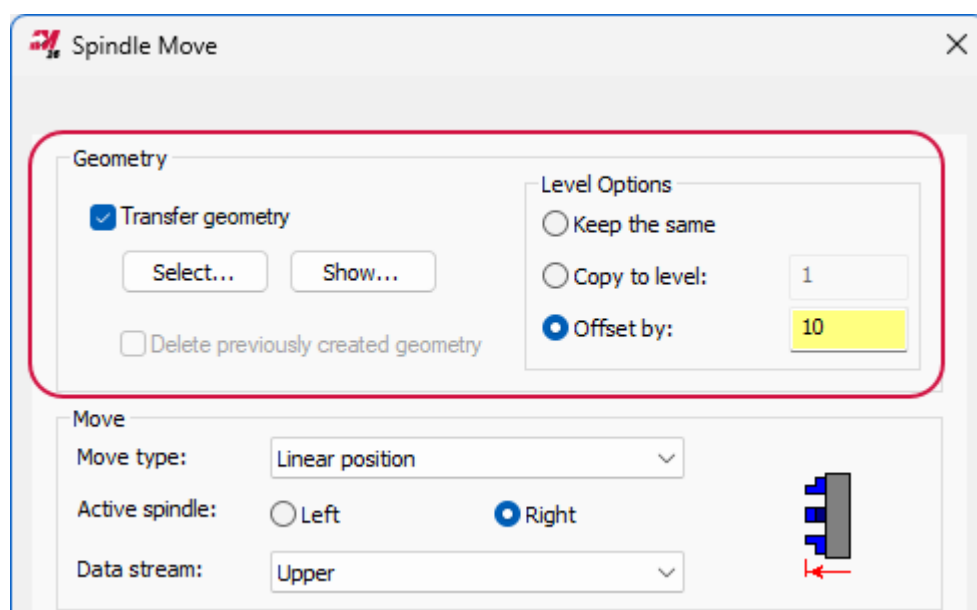
After selecting **Create New Spindle Origin**, select **Add plane to group**.



Mastercam groups the new plane according to the settings in Job Setup. If the **Group Planes** option is not selected in Job Setup, the **Add plane to group** option will be unavailable.



In addition, when creating a spindle move operation, you can now choose to transfer selected geometry along with the spindle.



Surface and Wireframe Toolpaths Available in Mill-Turn

Mill-Turn users in Mastercam 2026 can now use Mastercam's full suite of Surface rough, Surface finish, and Wireframe toolpaths. The following table lists all of the available toolpaths.

Surface rough

Parallel
Radial
Project

Surface finish

Parallel
Parallel steep
Radial

Wireframe

Ruled
Revolved
Swept 2D

Surface rough

Flowline
 Contour
 Restmill
 Pocket
 Plunge
 Multisurface Pocket

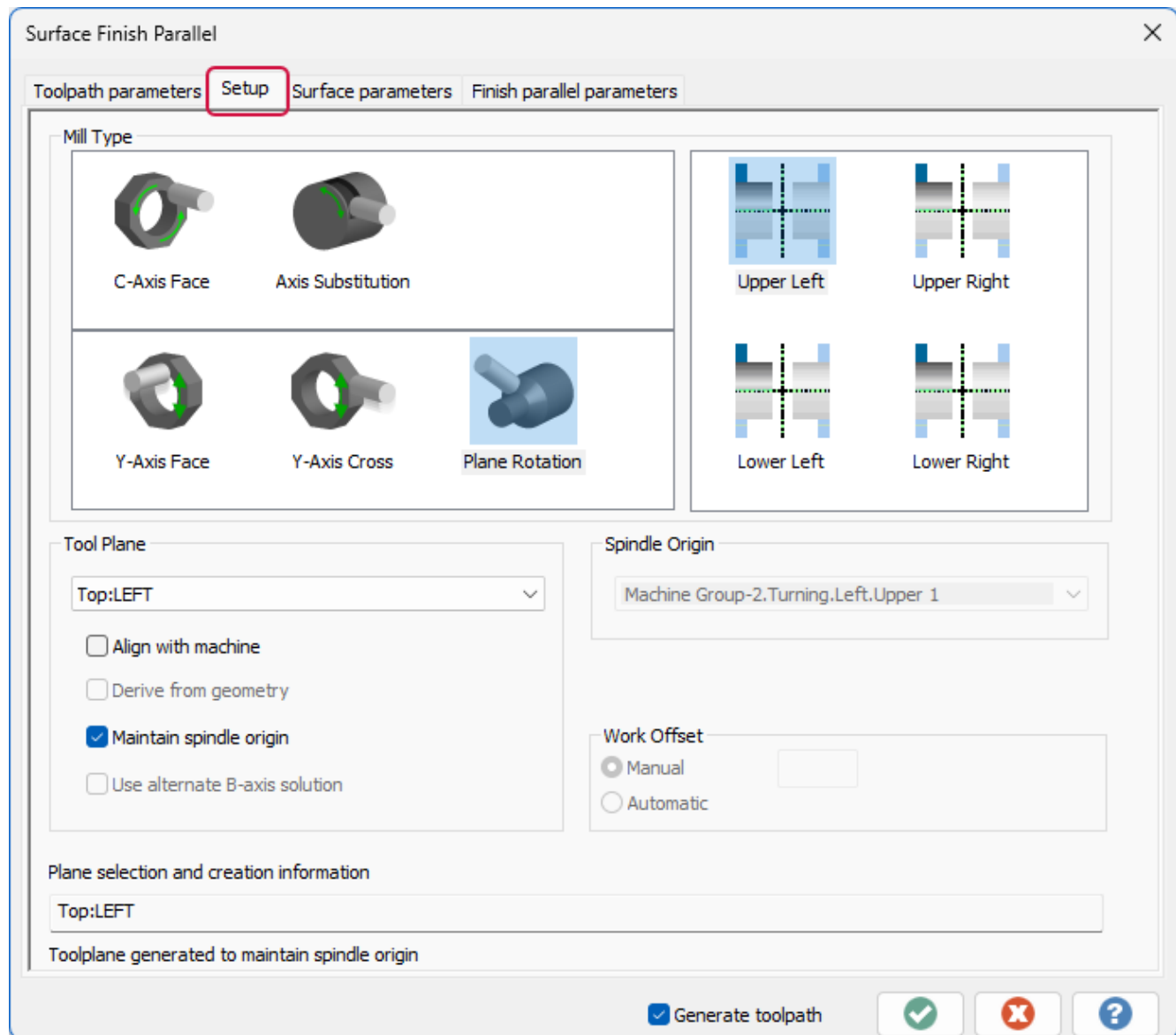
Surface finish

Project
 Flowline
 Contour
 Shallow
 Pencil
 Leftover
 Scallop
 Blend

Wireframe

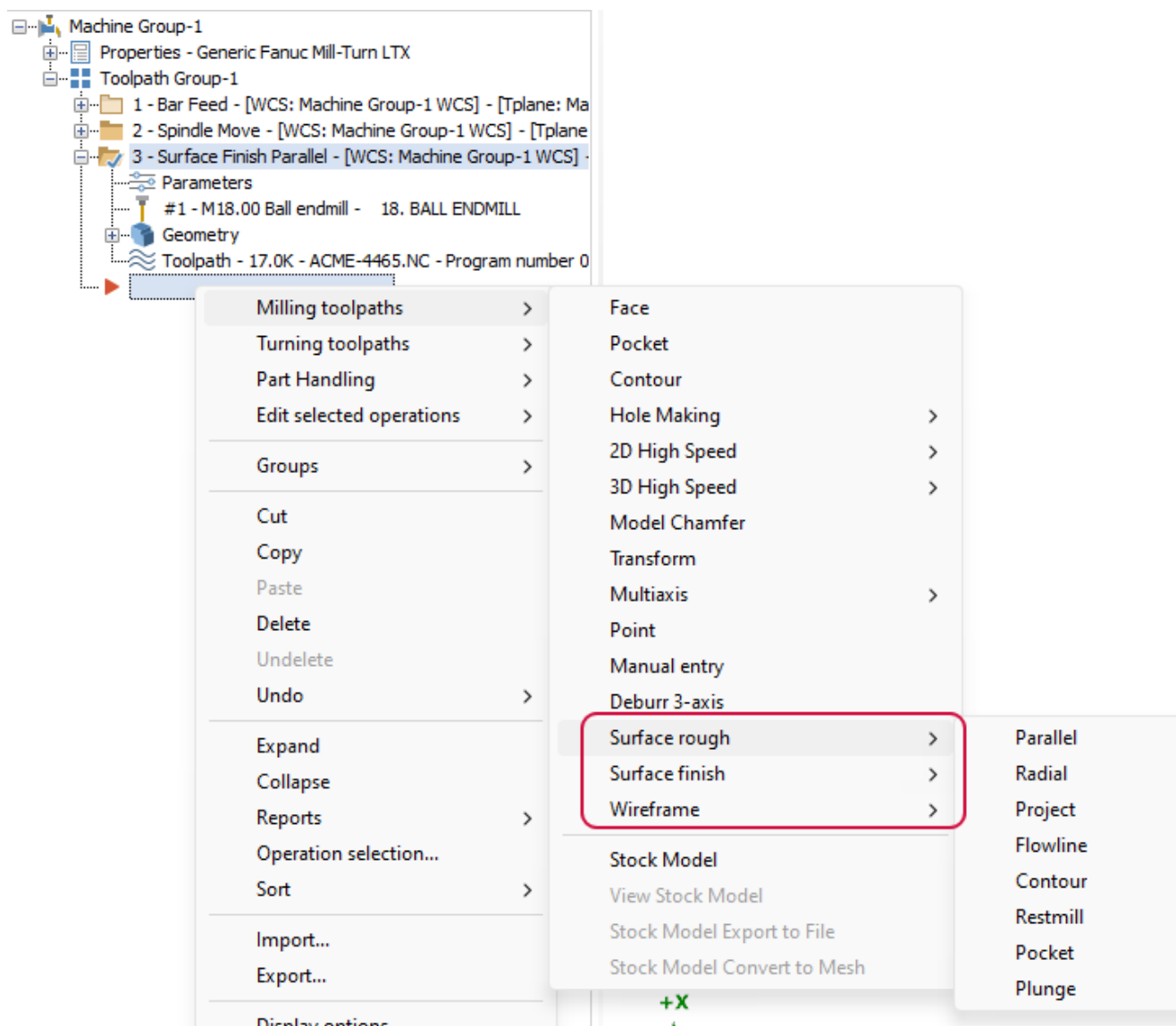
Swept 3D
 Coons patch
 Lofted

Mastercam 2026 preserves the familiar interface of these toolpaths, with the addition of the **Setup** tab which lets you tailor these toolpaths to your Mill-Turn machine.



Some of the functionality normally available on the **Toolpath parameters** tab has been moved to the **Setup** tab to be consistent with the Mill-Turn workflow. This includes selecting the axis combination, rotary axis settings, and plane selection.

To access the toolpaths, right-click in the Toolpaths Manager and select **Milling toolpaths**:



Note that several of the toolpaths allow multiaxis output when used in Mastercam Mill. This is not allowed in Mill-Turn, so the option has been removed for Mill-Turn. If you try to import such operations into a Mill-Turn machine group, Mastercam displays a warning and will not import the operations.

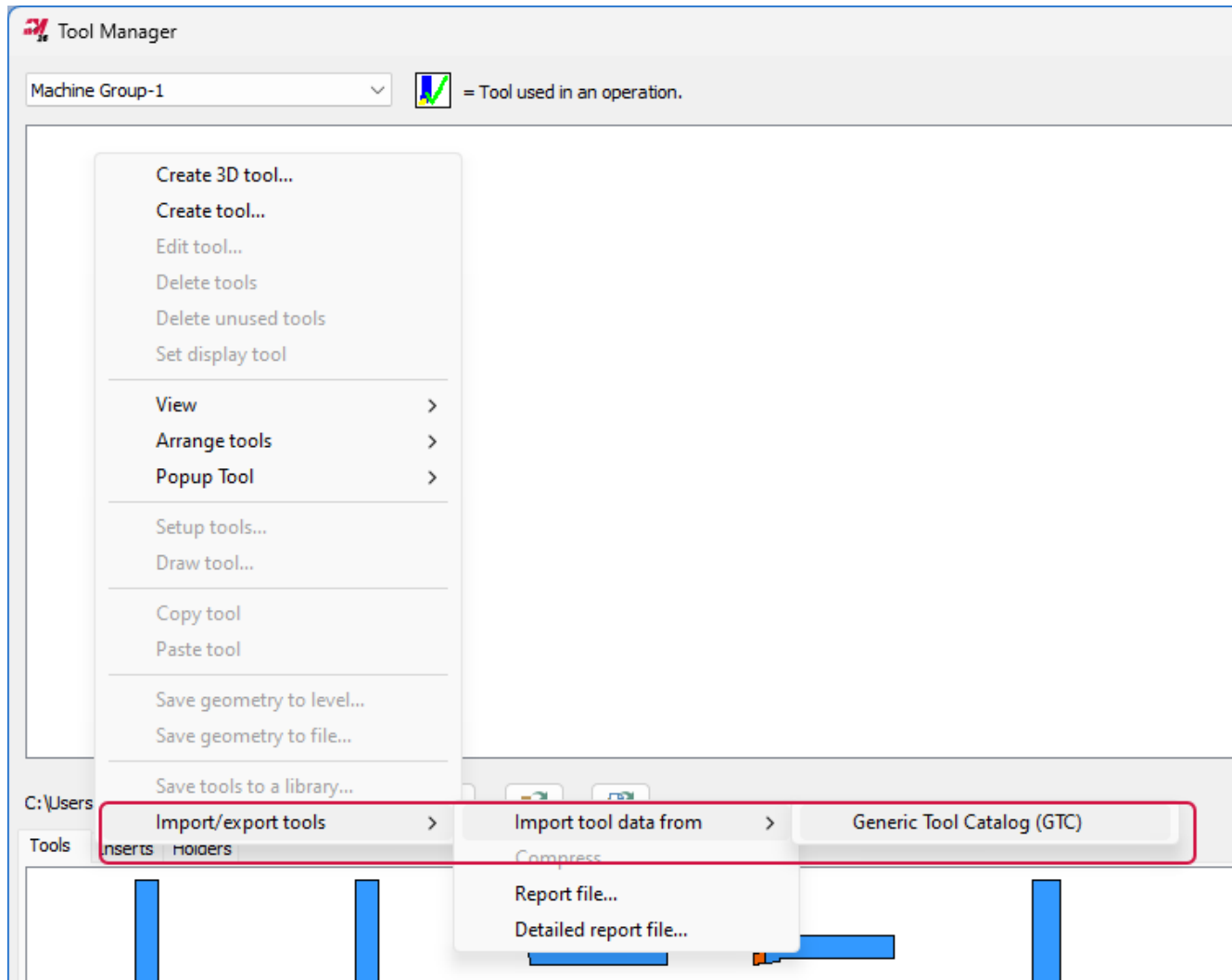
Tooling Enhancements

Listed below are major enhancements to tools and related features.

Import Lathe GTC Tool Data

Mastercam now lets you import lathe tool data from Generic Tool Catalog (GTC) packages. Previously, this ability was limited to milling tools. This greatly reduces the time that it takes to create 3D lathe tools. Similar to milling tools, Mastercam lets you select which assembly or individual components to import. It then validates the data and imports the tool or assembly directly to a Mastercam tool library (*.tooldb) or to the active machine group.

To import a GTC package, right-click in the Tool Manager and select **Import/export tools**, continuing until you select **Generic Tool Catalog (GTC)**.



Click **Open** to select the GTC package to import. For each tool assembly, Mastercam lists the individual components below it. You can also select and import the individual components. These are listed again following the assembly (below the red dashed line) with a checkbox so they can be selected. For example, if you wish you can select only the insert and add it directly to an insert library. Click **Validate** to have Mastercam inspect the data and models; it displays a green check mark next to the approved components.

Tool Importer

Open

Validate

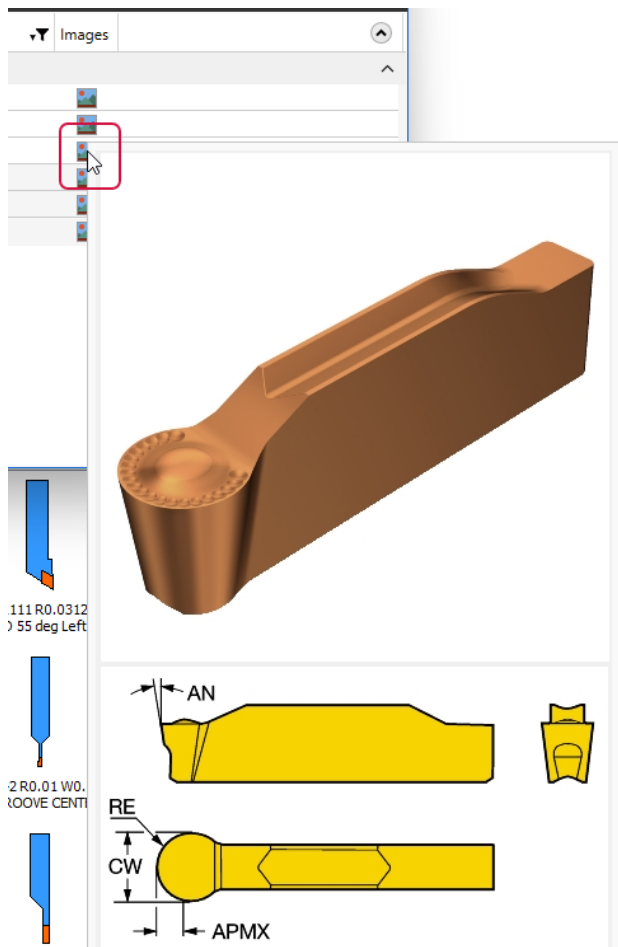
Save ▾

▼	▼	☐ Name	▼	Catalog Number	▼	Tool Type	▼	Images
	✓	☒ Boring bar/head - indexable T2 (3 items)						
	✓	HSK to cylindrical shank adaptor		392.T63-131-132-40				
	✓	CoroCut® 1-2 boring bar for profiling		LAX123J25-40B-020				
	✓	CoroCut® 1-2 insert for profiling		N123J1-0600-RM	1125			
		☐ CoroCut® 1-2 boring bar for profiling		LAX123J25-40B-020				
		☐ CoroCut® 1-2 insert for profiling		N123J1-0600-RM	1125			
		☐ HSK to cylindrical shank adaptor		392.T63-131-132-40				

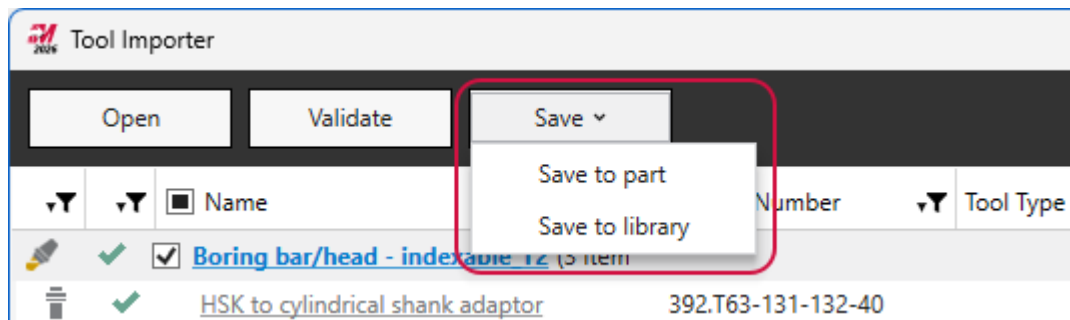
Click on the **Filter** button in any column to sort or filter the components.

Catalog Number	▼	Tool Type	▼	Images
		A ↓ Z ↓ Sort Ascending Sort Descending ☒ (Select All) ☒ 392.T63-131-132-40 ☒ LAX123J25-40B-020 ☒ N123J1-0600-RM 1125 OK Cancel		
392.T63-131-132-40				
LAX123J25-40B-020				
N123J1-0600-RM 1125				
LAX123J25-40B-020				
N123J1-0600-RM 1125				
392.T63-131-132-40				

Hover your mouse over a component's icon to see a detailed view.



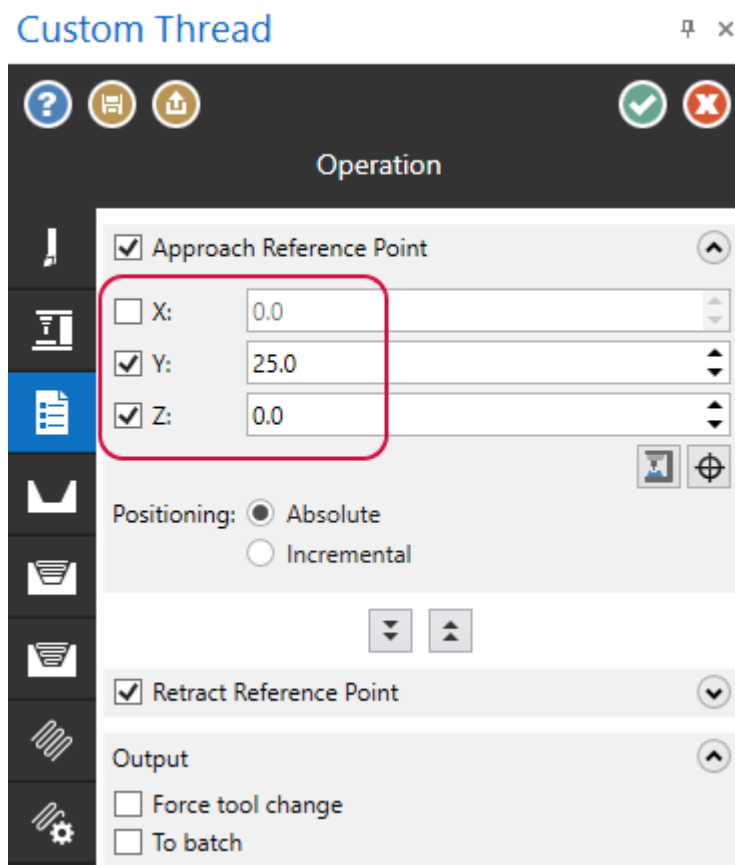
Once you are satisfied with the tool data, you can choose to save the components to the active machine group (**Save to part**) or you can select a *.tooldb file (**Save to library**).



Y-axis Tooling Support Added to Custom Threading

Mill Turn's **Custom Thread** toolpath now allows the use of Y-axis tooling. This provides more rigidity while cutting, with less vibration and longer tool life.

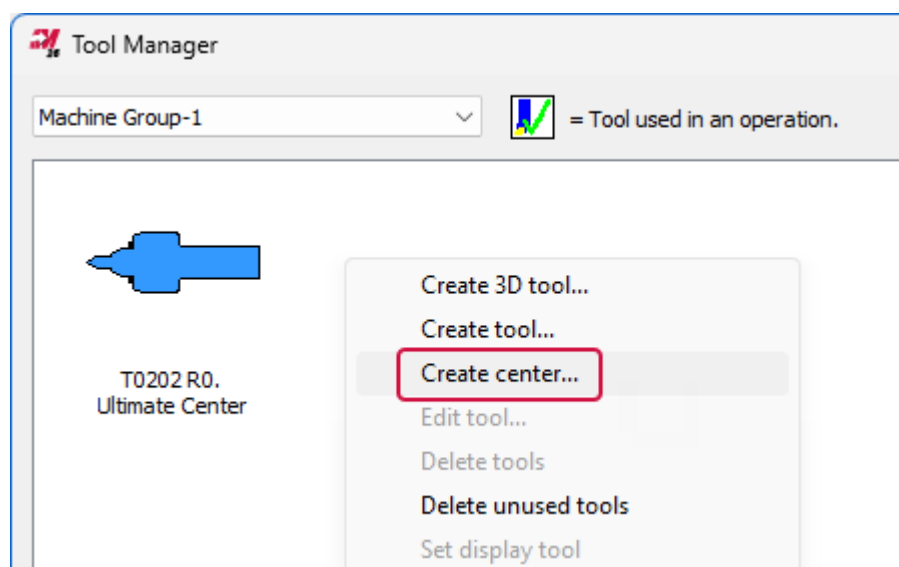
Y-axis tool support also enables all three axes for use in approach and retract reference points. When a Y-axis tool is used, all three axes are available for selection.



New Lathe Center Designer

Mastercam's new Lathe Center Designer continues the effort to create a single standardized workflow for creating tools, holders, and assemblies across different Mastercam products. It is available for Mill-Turn users only to create center components. Use it to create both live and stationary centers, and to create centers which can be mounted in either a tailstock or turret.

Access the Lathe Center Designer by right-clicking in the Tool Manager and choosing **Create center**. You can also access it while creating a Center Point operation or a tailstock part handling strategy.



The panel design is very similar to the new Mill Holder Designer interface.

Lathe Center Designer

Component

Component

Name: Ultimate Center

Description:

Manufacturer: Royal

Catalog ID:

Tool number: 2

Tool offset number: 2

Compression length: 0.0

Maximum weight: 200.0

Maximum RPM: 6000

Units

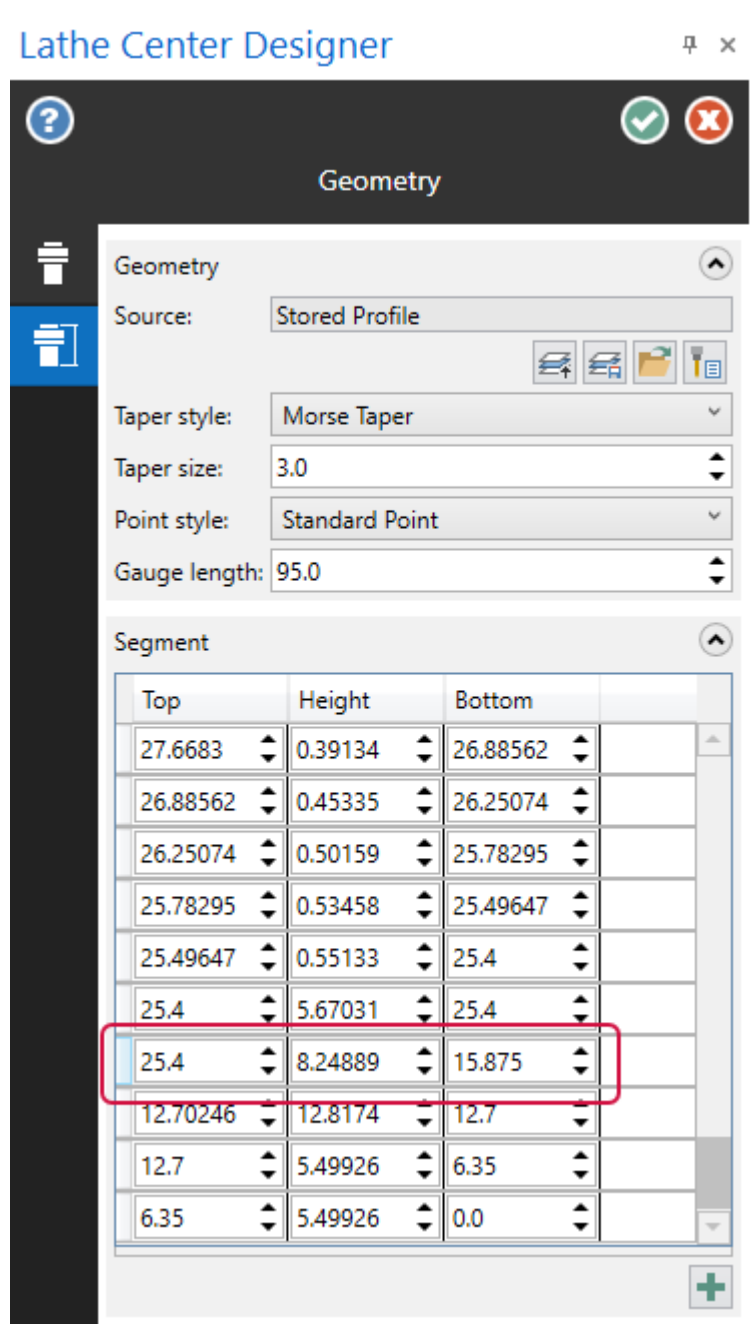
☐ U.S.

☒ Metric

Use the **Geometry** controls on the second page to select and manage the solid model that represents the center.

- The **Taper style** and **Point style** drop-down lists let you describe the center in industry-standard terms.
- The **Gauge length** specifies how much of the center projects from the quill or turret.

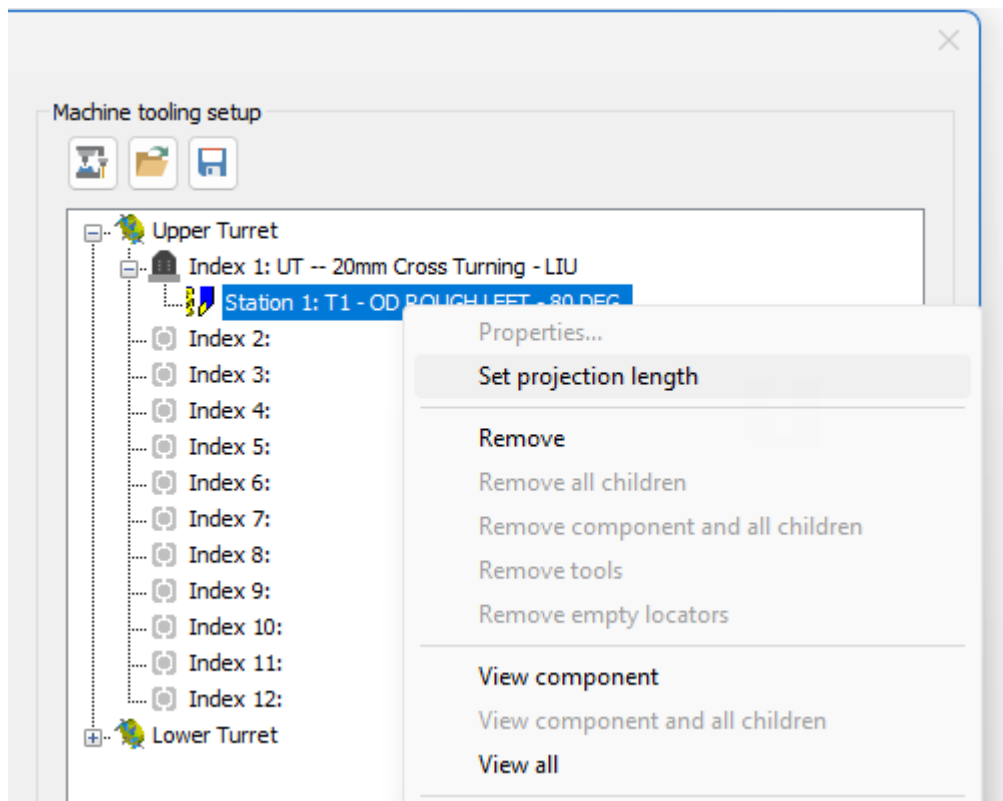
The center profile is represented as sections organized in a grid. Edit the dimensions of each section as desired to fine-tune the profile, or right-click in the first column of the grid to see additional options.



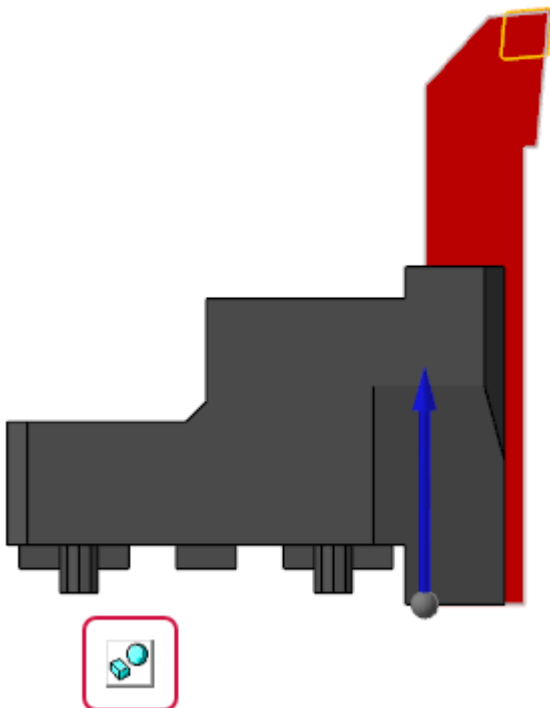
Improved Workflow for Setting Tool Projection

Mastercam Mill-Turn includes a new workflow for setting the tool's projection length. When you manually adjust the projection by dragging the tool in the graphics window, Mastercam typically displays the projection length measured from the back of the tool holder. Often, this number is not meaningful to the programmer or setup person. Use the new workflow to measure the projection from the face of the tool locator or other meaningful location, making it easy to precisely match a value from a blueprint or setup sheet.

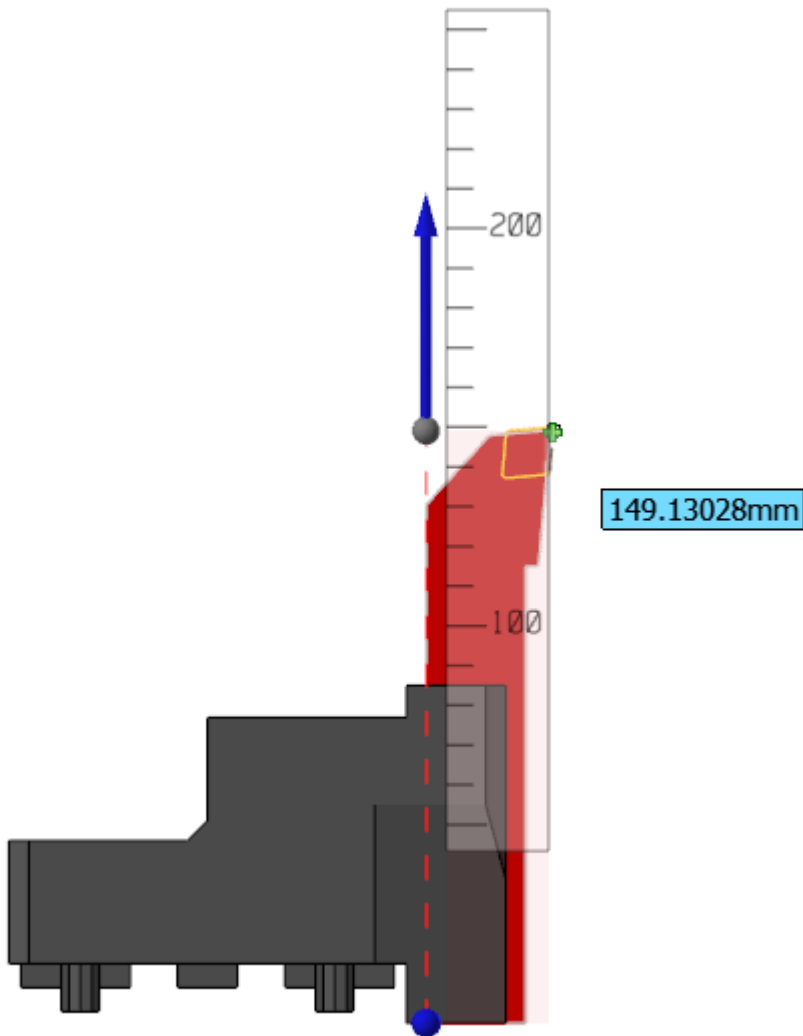
To begin, open the Tool Setup Manager. Right-click on the tool and choose **Set projection length**.



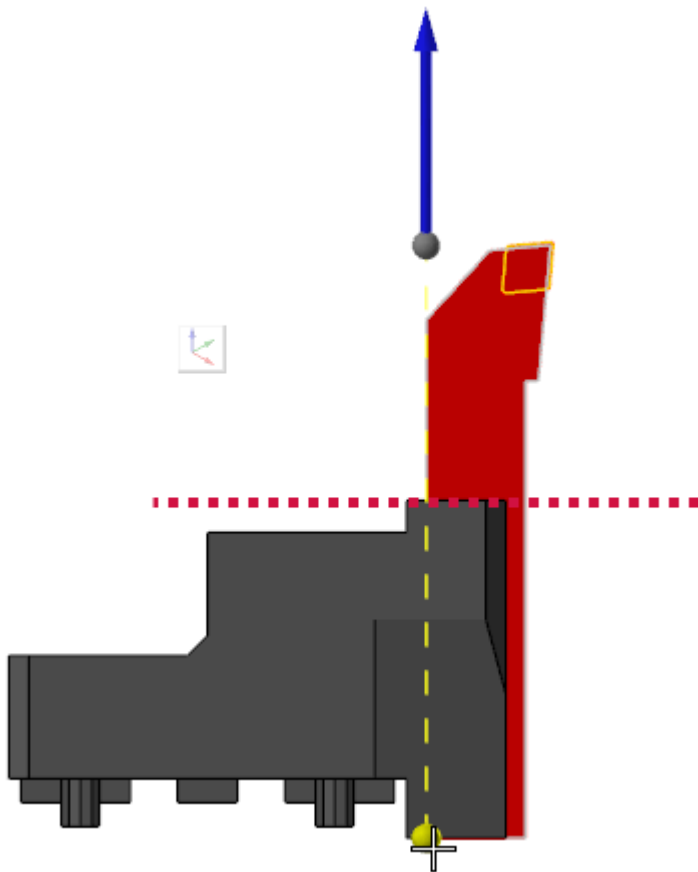
When the tool appears in the graphics window, click the new mode button. This button toggles between **Geometry** mode, where you can drag the tool, and **Axis** mode, where you can drag the gnomon arrow or origin. Click it to switch to **Axis** mode so you can relocate the zero point to the face of the locator.



Click the axis arrow and drag it to the tip of the tool, or to another point that indicates the maximum tool projection.



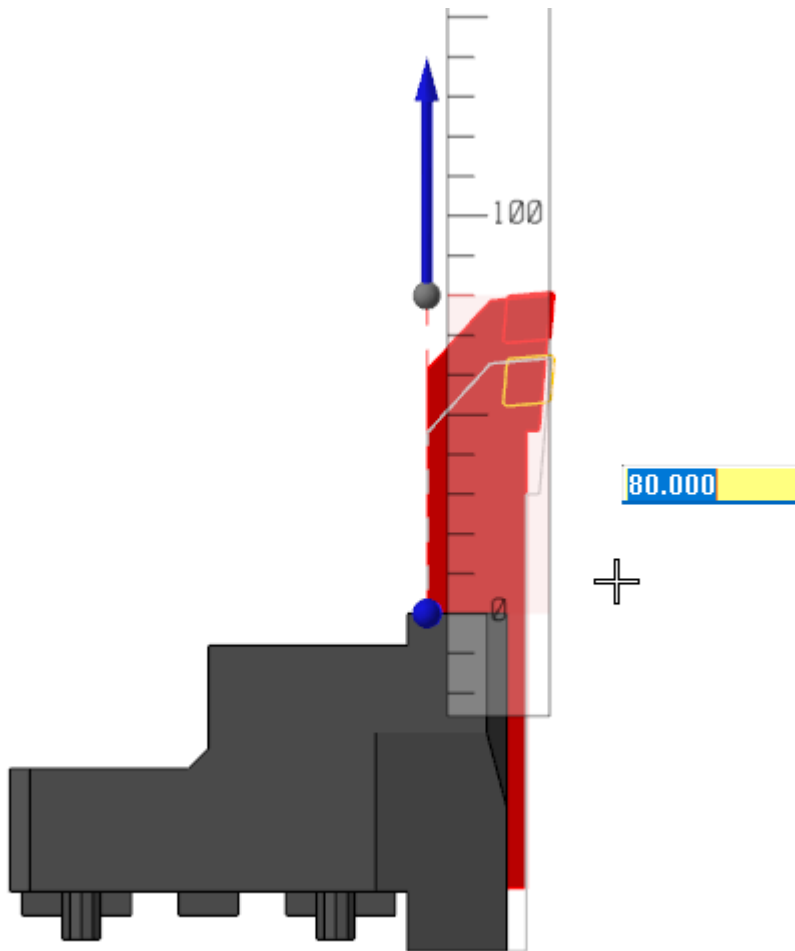
Next, drag the ball which represents the origin to the face of the locator, or to the location which represents the zero point for the projection length.



When you have successfully relocated the origin, click the mode button to switch back to **Geometry** mode.



Now you can click on the tool and drag it to the desired projection length. You can also simply type the desired distance—for example, 80mm—directly in the distance field.



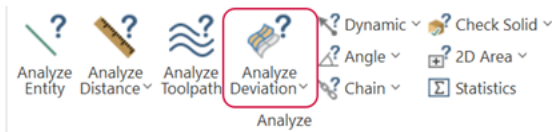
Designing Enhancements

Listed below are major enhancements to designing functions.

Analyze Deviations Between Mesh Entities

The Analyze Deviation function, which was introduced in Mastercam 2025, identifies dimensional differences between entities. In that release, analysis was limited to different combinations of points, curves, surfaces, bodies, and faces. In this release, the Analyze Deviation function now includes support for mesh to provide more flexibility and precision in your design and verification workflow.

Analyze Deviation is located on the **Home** tab in the **Analyze** group.



Under the drop-down, the ribbon control presents five entity combinations to compare. The following analysis features now offer mesh support:

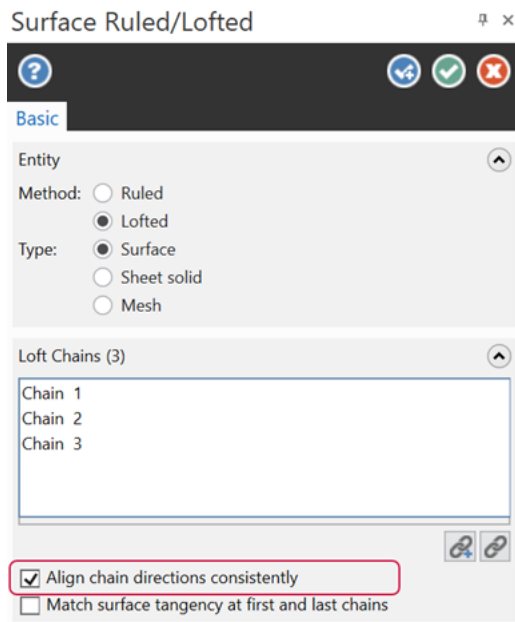
- **Surface to Surface:** Surface can now mean a surface, solid face, sheet solid, solid body, or a mesh.
- **Curve to Surface:** Mesh support has been added.
- **Point to Surface:** Mesh support has been added.

Loft Surface Enhancements

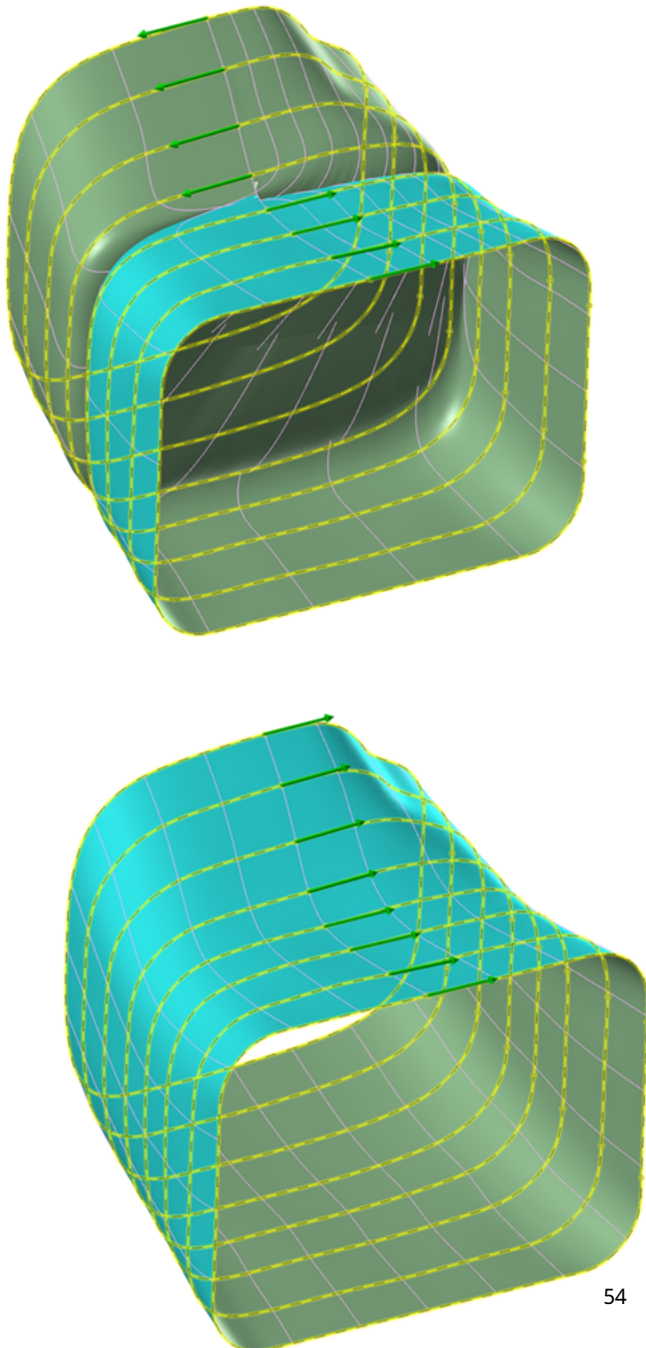
Mastercam 2026 introduces new features in **Loft**. When using wireframe geometry to create a lofted surface, you now have more controls which allow for smoother flow and blending of surfaces.

Automatic Alignment of Chain Directions

Mastercam 2026 introduces the automatic alignment of chain directions, allowing you to window select chains without worrying about their direction. This feature saves time when you need to select multiple chains. It is enabled by default, but you can disable it from the function panel by deselecting the **Align chain directions consistently** option.

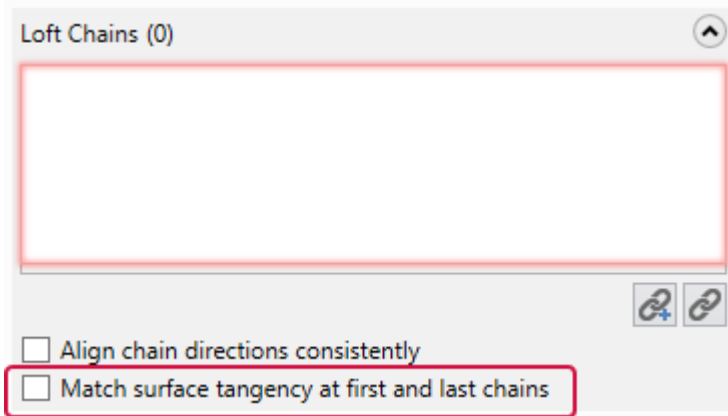


The bottom image shows how this feature aligns the disparate chain directions of the top image.

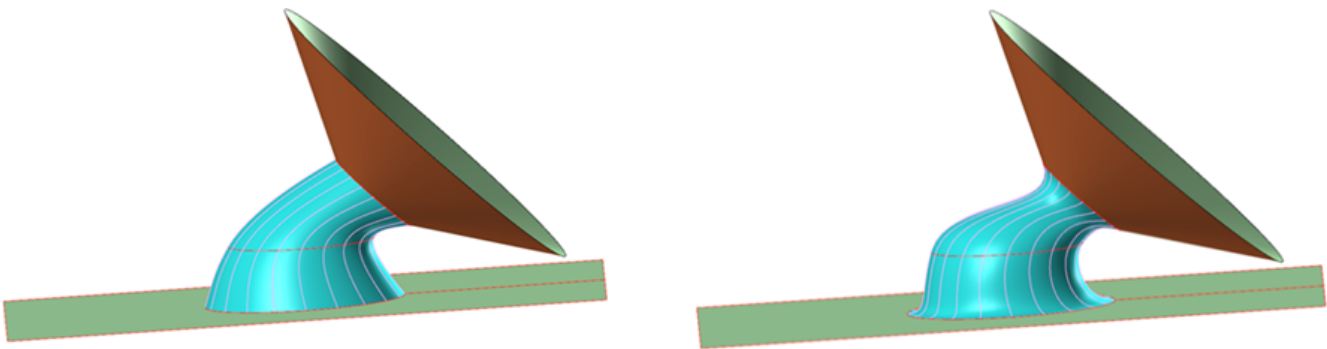


Match Surface Tangency at Start and End of Surface

The new surface tangent matching feature blends loft surfaces smoothly to existing adjacent surfaces. Control this behavior by selecting the **Match surface tangency at first and last chains** option in the Surface Ruled/Lofted function panel.



In the following image, the option is deselected on the left example and selected on the right. Note how the loft surface in each image transitions to the surfaces adjacent to the first and last chains.

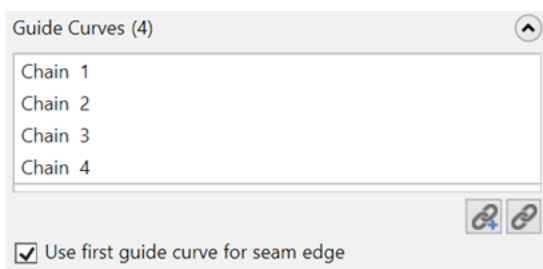


The new **Match surface tangency at first and last chains** option blends the loft with the surfaces that are adjacent to the first and last chains. There is no need to manually select these tangent surfaces, as they are recognized automatically.

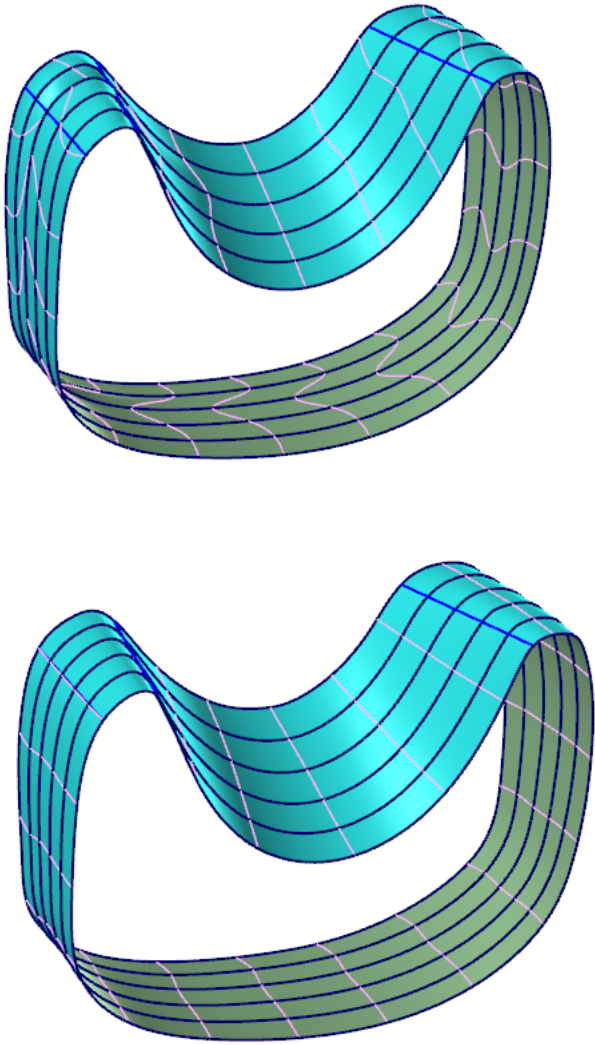
Guide Curves

Guide Curves are a new way of syncing flow across selected chains of curves. This feature eliminates the need to manually break up groups of chains to sync curves to your desired flow. Guide Curves are easy to create and do not need to intersect with loft chains.

Use the list box in the Surface Ruled/Lofted panel to add, remove, and re-select **Guide Curves**.



The images below show a chain group before and after Guide Curves are added and selected.



This feature is especially helpful if the endpoints of the curves are uneven or bumpy. It forces the flow to follow the direction you need. The shape of your surface will not be adjusted based on the added Guide Curves. They only influence the surface shape by improving flow.

Guide Curves work hand in hand with **Dynamic Sync** but are optional.

Dynamic Sync

Dynamic Sync options have been reorganized and updated.

- **Equal arc length** has been added as the default since it is the feature that will typically work the best to connect curves. It works by spacing points by relative equal arc length along the curves. This turns spline curves into constant speed curves at the evaluator level.
- **Planar slice** has replaced **Average** and only applies lofting with closed chains.

Dynamic Sync works well with **Guide Curves** to create a better, more consistent flow for a smooth surface.

Solid Hole Enhancements

Mastercam 2026 redesigns the workflow for solid hole creation. This new design and additional features discussed below allow you more control and customization for holes.

In the previous release, hole creation was limited to the options on the function panel's **Basic** tab, as seen in the following image.

Hole

Basic Advanced

Operation

Name: Simple Ø 10.00

Target: Solid

Plane Orientation

Top

Position (1)

Position 1

Template

Hole Style

Type: Simple

Diameter	10.0

Depth

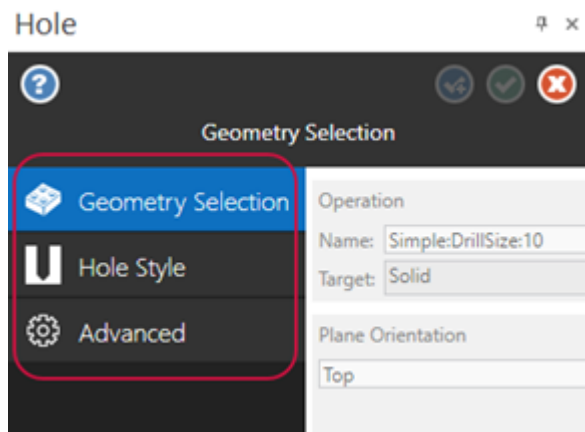
☒ Distance: 25.0

Bottom angle: 118.0

☐ Through all

☐ Top Edge Treatment

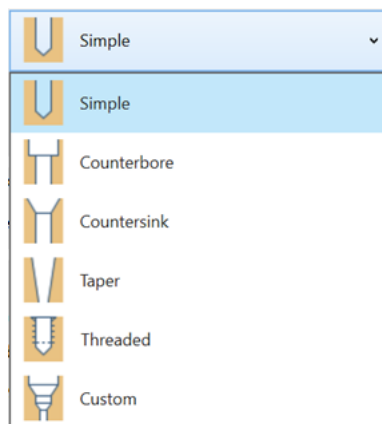
In this release, the **Hole** function panel is organized into three tabs: **Geometry Selection**, **Hole Style**, and **Advanced**. The new features and functions on these tabs allow you to design different hole types with precision and control.



Create and Save Hole Configurations with the Hole Style tab

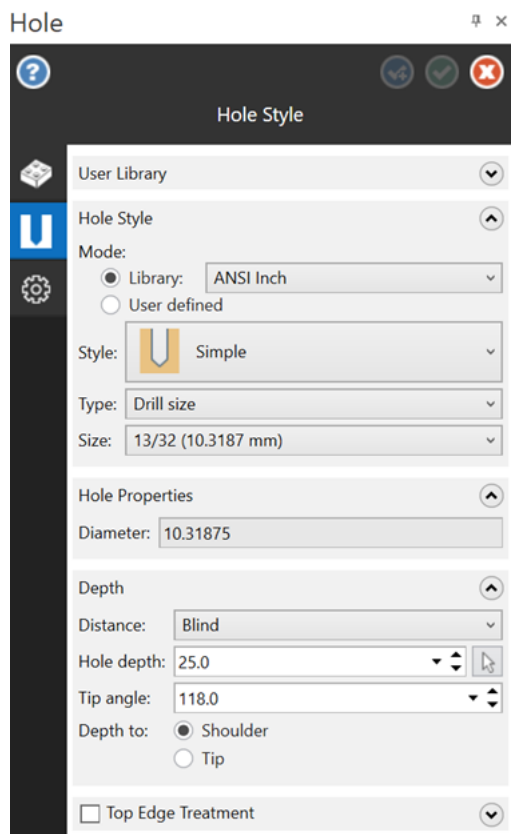
The new **Hole Style** tab offers an array of options that help you define your hole. These include **Style**, **Type**, and **Hole Properties**. Select from industry standard libraries or create a custom defined hole with the **User defined** mode. After you've defined your custom holes, you can save them to a **User Library** for future use.

Style options include **Simple**, **Counterbore**, **Countersink**, **Taper**, **Threaded**, and **Custom**.



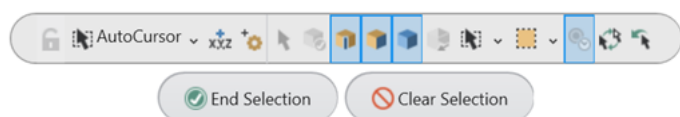
Along with these **Style** selections, you can also define the **Type** and **Size** of holes created in **Library** mode. Specific options will vary depending on the hole **Style** you select.

While in **Library** mode, most **Hole Properties** are read-only fields. **User defined** holes enable more adjustments and greater customization in the **Hole Properties** section.



Selecting Solid Bodies, Edges, and Faces

In previous versions of Mastercam you could use either the **Selection Bar** or the **Solid Selection** window to select geometry. These methods often conflicted and created problems during the selection process. In this release, we have streamlined the selection process by eliminating the Solid Selection window. Now, when solid selection is required, the Selection Bar highlights to prompt you to complete your selection.



This change is a notable usability improvement for selecting solid faces, edges, and bodies. Removing the window eliminates any issues caused by offering several selection methods and creates a smoother, more efficient workflow.

Updated CAD File Support

Mastercam 2026 includes updates to many CAD file translators.

For a complete list of supported file format versions, see the [Mastercam Translators](#) page.

Simulation Enhancements

Listed below are major enhancements to Mastercam Simulator.

Fine Tune Simulation Speeds

Users who need to adjust the speed of their simulations quickly and precisely now have enhanced control over simulation speeds.

To adjust the speed by increments of 10%, simply click on the **Slow** or **Fast** labels on Mastercam Simulator's playback bar. This enhancement provides a more intuitive and efficient way to adjust simulation speed.



Focus on Current Operation Information

Mastercam Simulator's new **Operation Info** section makes it easier for you to analyze specific operations. The new section has been added to the **Move Info** pane. It displays detailed information such as the operation number, comments, and feed time.

Operation Info displays only information pertaining to the currently simulated operation. The **Toolpath Info** section provides data for the entire set of simulated operations.

Move Info	
Move ID	1 of 946
Elapsed Time	1.00s
In-Cut Time	0.00s
Machine	5AxHeadHead
Move Type	Contour - line
Tool Number	2 - 25 Chamfer Mill
Tool Orientation	
Tool Tip Position (WCS)	
Tool Tip Position (Tplane)	
B-axis Angle	
Axis Values	X=32.450; Y=22.125; Z=301.813; C=0.000; B=0
✓ Operation Info	
Operation Name	Deburr 3-axis
Operation Number	ID: 1 (1 of 1)
Operation Comment	
Feed Length	440.043
Feed Time	3min 28.13s
Rapid Length	940.840
Rapid Time	4.52s
✓ Toolpath Info	
Feed Length	440.043
Feed Time	3min 29.13s
Rapid Length	940.840
Rapid Time	4.52s
Total Length	1380.883
Total Time	3min 33.65s
Min/Max X	-59.702 / 61.331
Min/Max Y	-27.976 / 37.257
Min/Max Z	-5.802 / 150.000

Reflection Effect Added to Mastercam Simulator

A new control has been added to Mastercam 2026 to enhance the realism of your simulations. The **Material texture** option improves simulations by applying reflective qualities to **Milling**, **Stock**, and **Workpiece** objects. This feature is only available in the **Simulate** mode of the **Mastercam Simulator**.



To use this option:

1. Select the **File** menu, and choose **Options** from the **Backstage** menu.
2. Click the **Graphics** tab.
3. In the **Milling**, **Stock**, and **Workpiece** sections, select the **Material texture** option in each section to display the reflections in the simulation window.

This feature requires a material image to be added to the machine directory and specified in the .xml file for the selected machine. For more information on this process see Help.

4. Control the intensity of reflection by entering a **Texture reflectivity** value. You can also use a the slider to adjust this value.

Milling

Holder color:	#336699
Shank color:	#666666
Shoulder color:	#999999
Flute length color:	#ddbb00
Opacity:	50.00 %
Texture reflectivity:	0.00 %
Material texture	☒ Display the material texture on the milling tool

Stock

Stock color:	#f66d6d
Opacity:	50.00 %
Texture reflectivity:	0.00 %
Material texture	☒ Display the material texture on the stock

Workpiece

Workpiece color:

Opacity:

Texture reflectivity:

Material texture

☒ Display the material texture on the workpiece

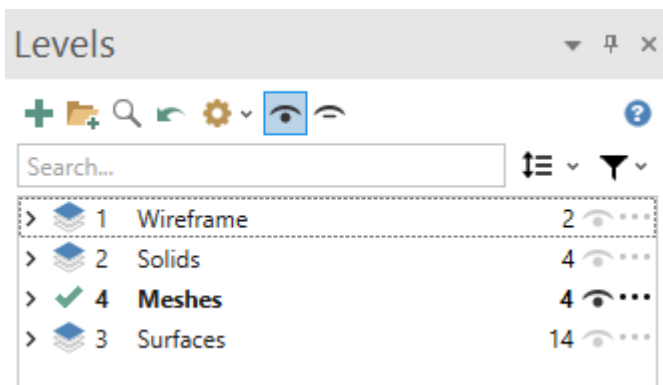
System Enhancements

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

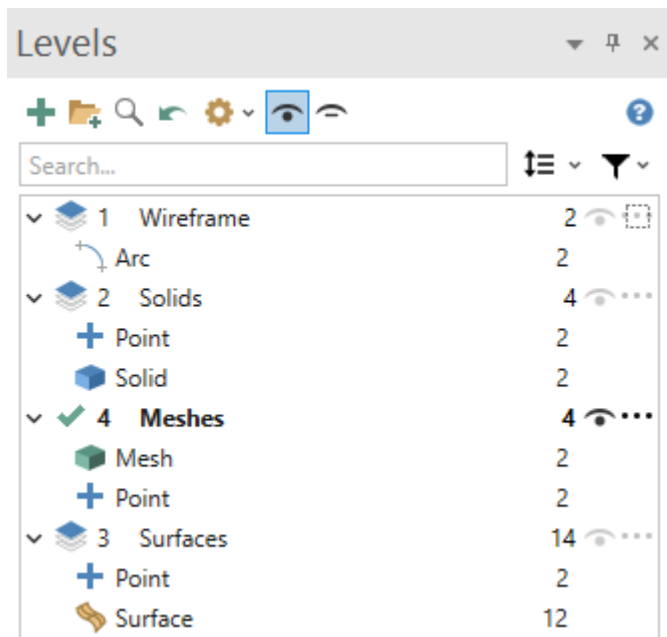
Redesigned Levels Manager

Mastercam 2026 features a redesigned Levels Manager, with levels organized into a tree structure. The new manager lets you perform such functions as those shown in the following list.

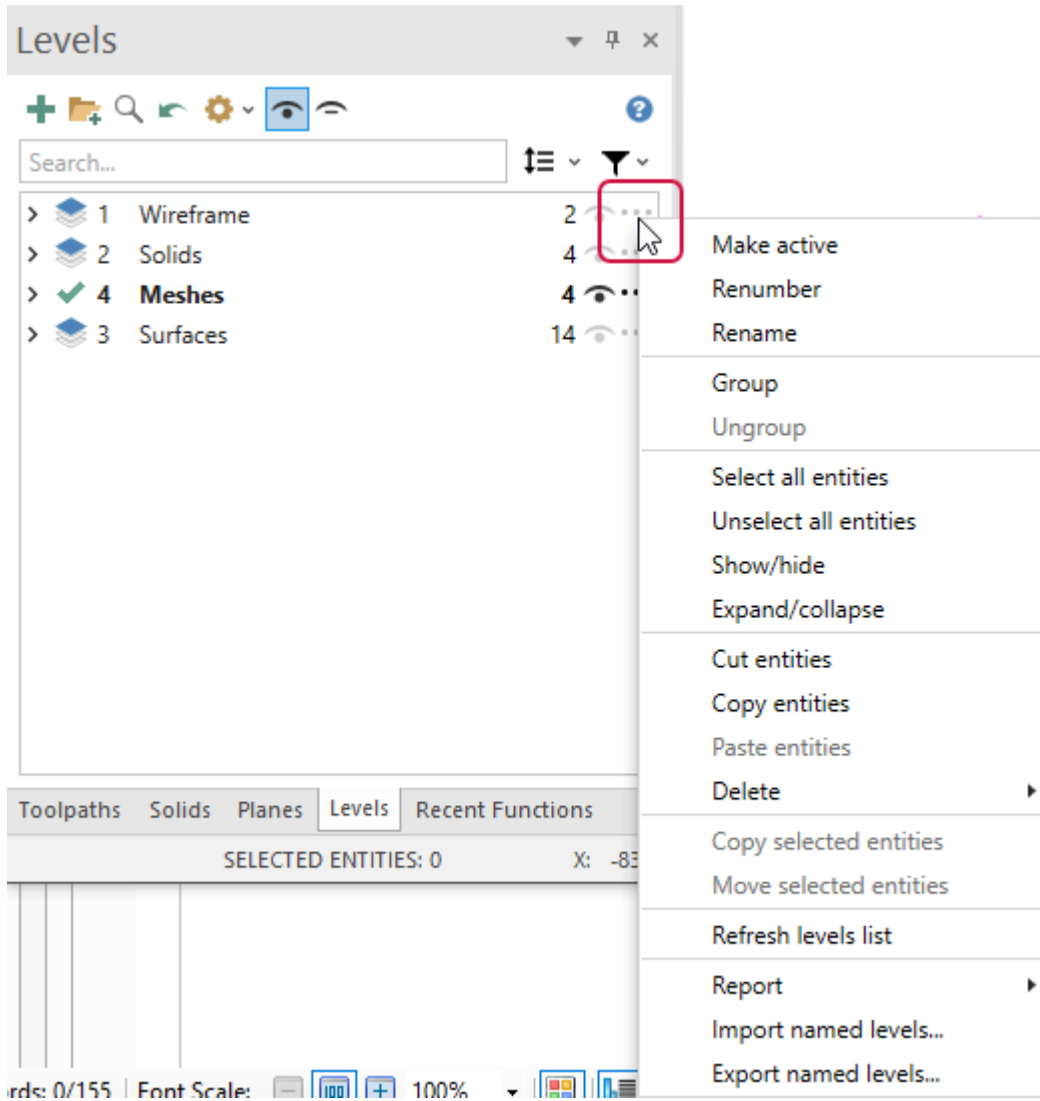
- Create and nest groups.
- Drag-and-drop within the tree.
- Search the tree.
- Filter the tree by groups.
- Expand and collapse groups for better organization.
- Find levels from geometry.
- Reset all levels.
- Customize display options.
- Turn all levels on or off.



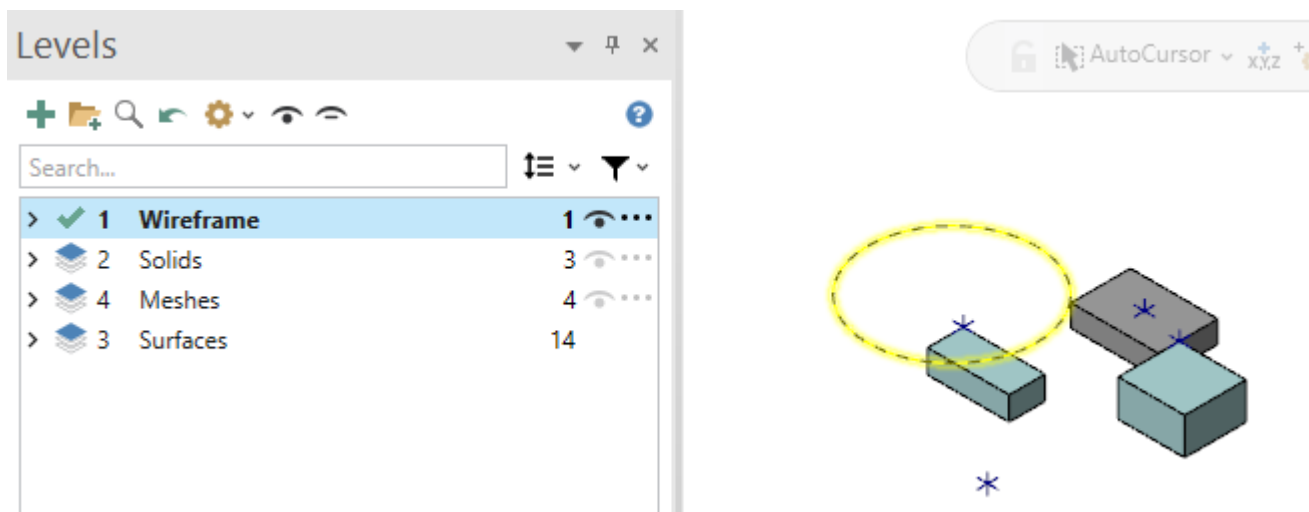
The Levels Manager's tree structure lets you see the entities on any level by clicking the arrow to the left of the level's name. The following image shows all of the levels open to list their entities. Each entity's count is also shown in the list.

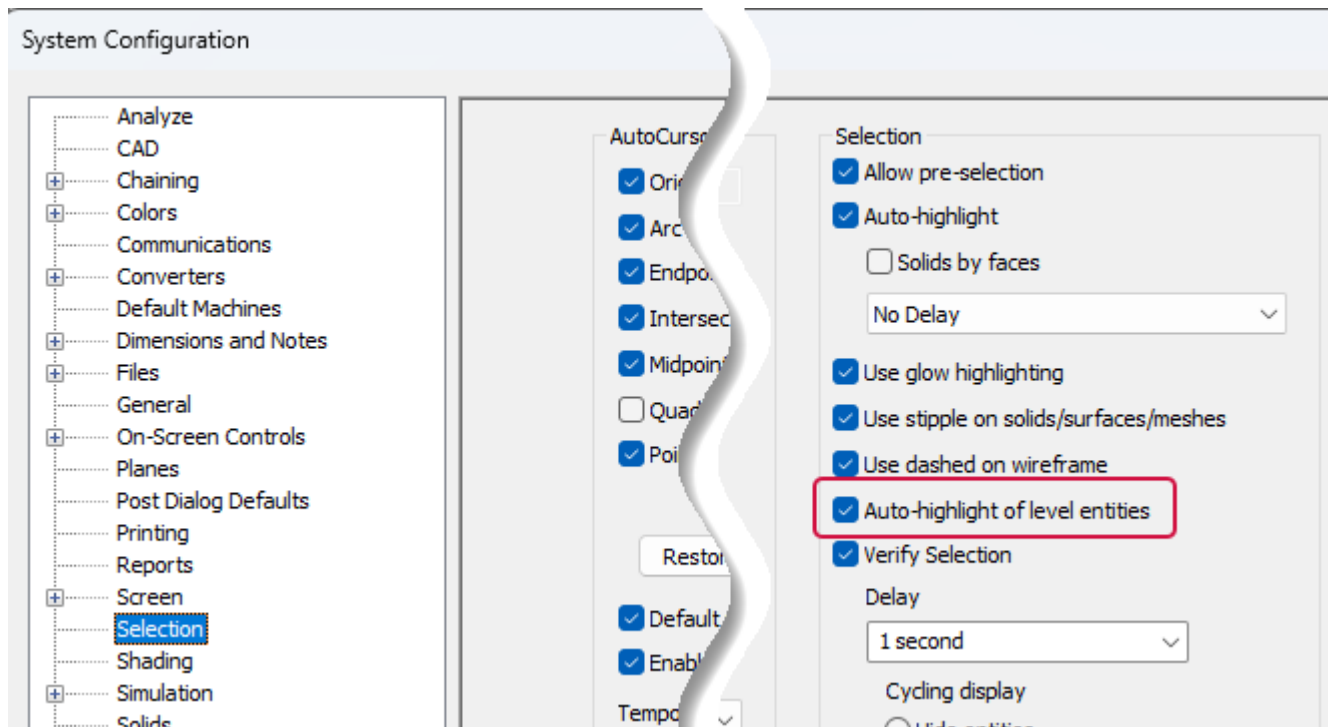


You can access functions that are specific to each level by clicking the **More actions** icon to display the menu shown in the following image. The entities within each level also feature a drop-down menu, for typical editing functions such as cut, copy, and delete. Each level displays the number of entities on the level, as well as an "eye" button for turning a level on or off.



When you click a level or a level's entity summary, Mastercam highlights the associated geometry in the graphics window. The following image shows the **Wireframe** level selected, with the associated wireframe geometry highlighted in the graphics window. The second image following shows the **Selection** page of the **System Configuration** dialog box, where you can toggle the auto-highlighting function, as needed.





Redesigned Planes Manager

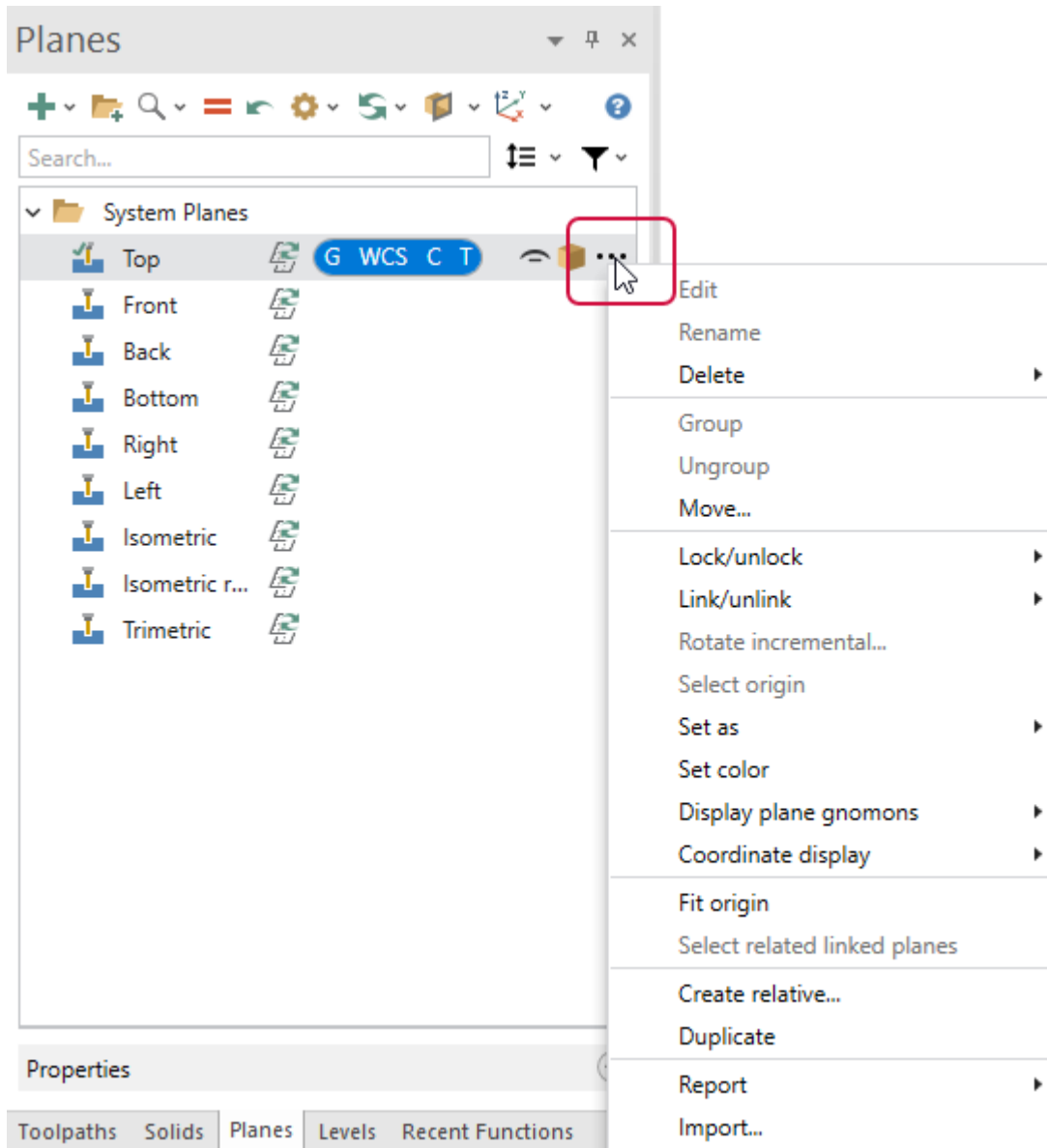
Mastercam 2026 features a redesigned Planes Manager, with planes organized into a tree structure. The new manager lets you perform such functions as those shown in the following list.

- Create groups and nested groups.
- Drag and drop within the tree.
- Find a plane based on geometry or a plane in the graphics window.
- Search the tree.
- Filter the tree by groups.
- Expand and collapse groups for better organization.
- Customize the properties display.
- Sort planes on various criteria.
- Customize display options.
- Set "follow" rules.
- Manipulate section views.
- Customize gnomon display.

You can access functions that are specific to each plane by clicking the **More actions** icon to display the menu shown in the following image. You can also get to this menu by right-clicking a plane. Each plane in the list features the following:

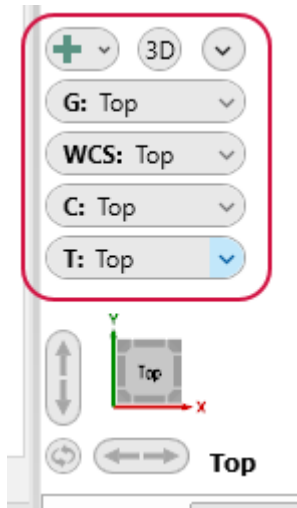
- Plane icon
- Plane name

- Work-offset number (or an icon for automatic offsets)
- Type of plane (graphics, WCS, tool, or construction)
- Gnomon toggle
- Section view indicator



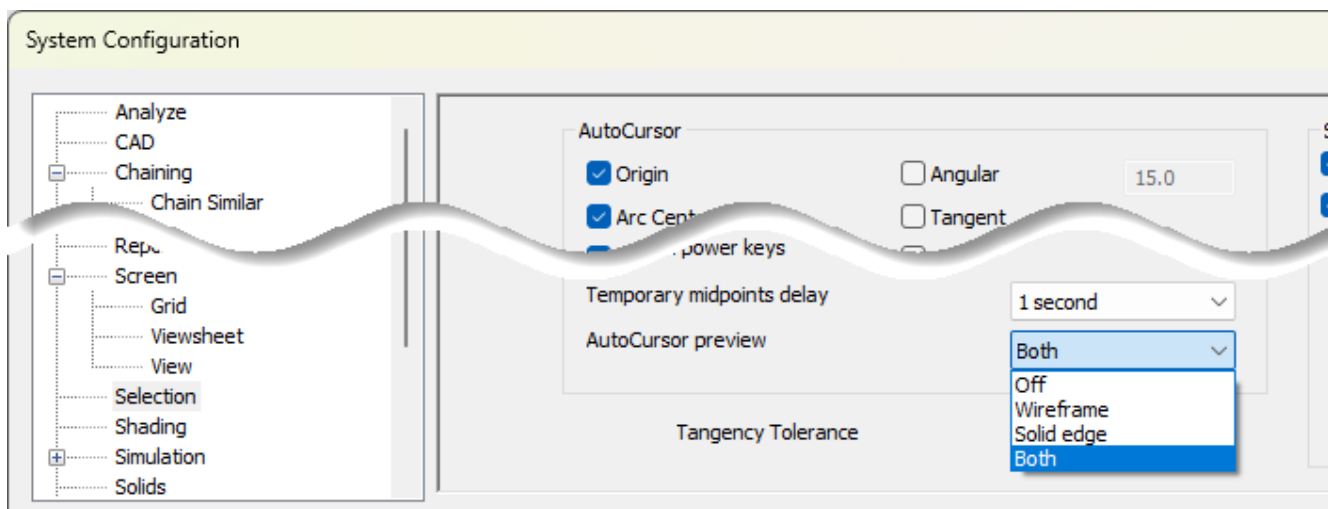
Control Plane Display with Overlay Controls

Mastercam 2026 adds plane overlay controls to the graphics screen. These controls let you to create a plane or change the WCS, construction, or tool plane without opening the Planes Manager. You can also change the 2D/3D control, as well as the graphics view.



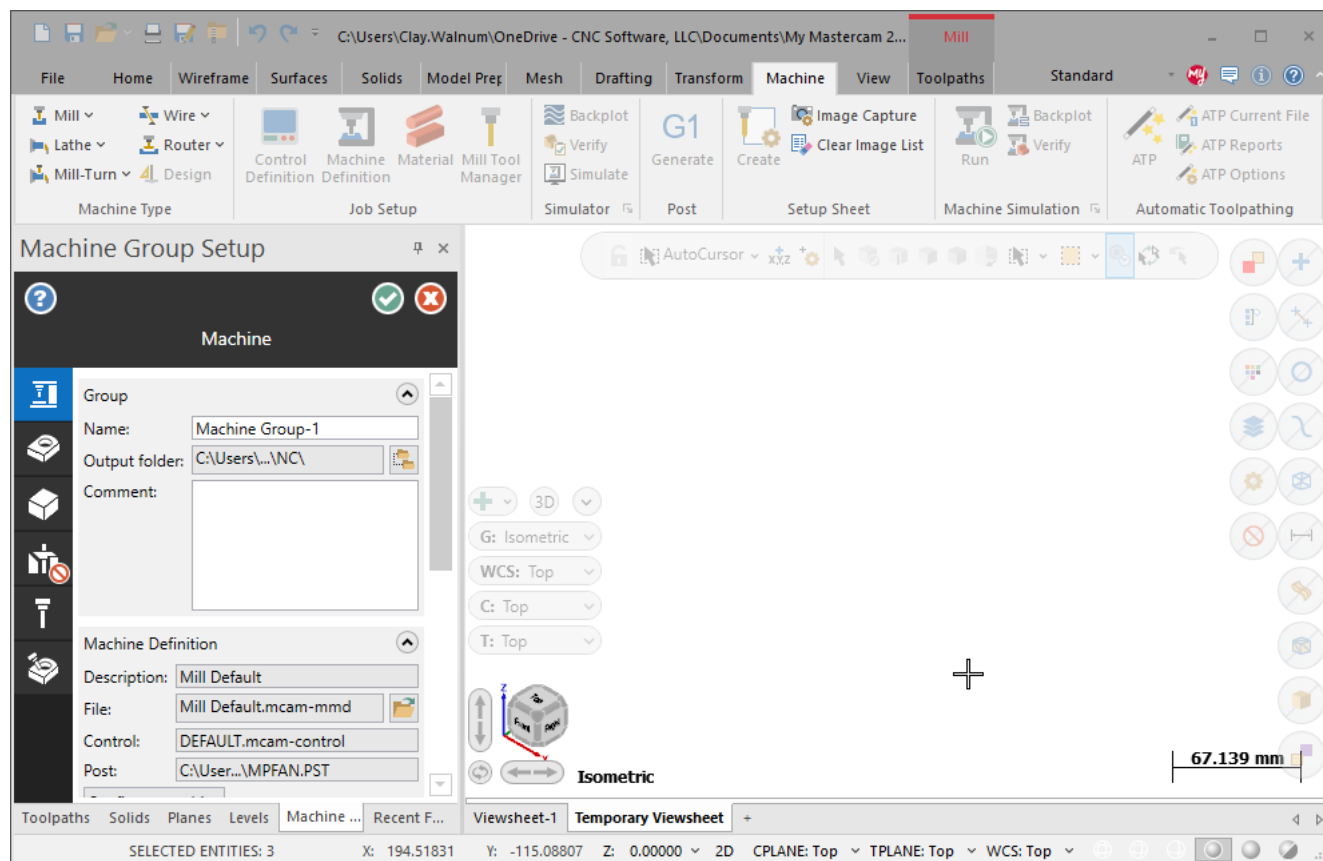
Control AutoCursor Preview Display

Mastercam now lets you customize the AutoCursor preview, which appears when you move the mouse over entities. In the **Selection** page of the **System Configuration** dialog box, select an option to indicate wireframe, solid edges, or both. You can also turn off the preview cursor.



Disable Inaccessible Functions

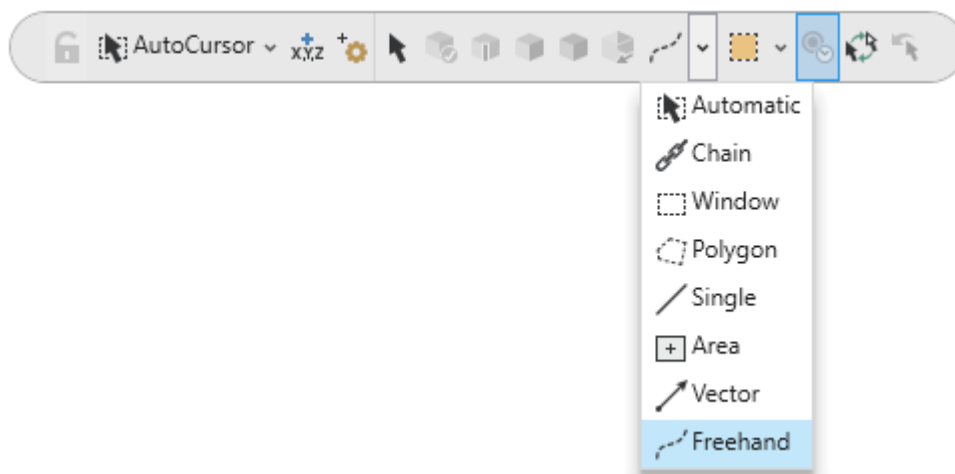
Mastercam 2026 disables functions that are inaccessible when you are working in other functions. For example, the following image shows Mastercam inside the **Machine Group Setup** panel, which disables many functions in the main Mastercam ribbon.



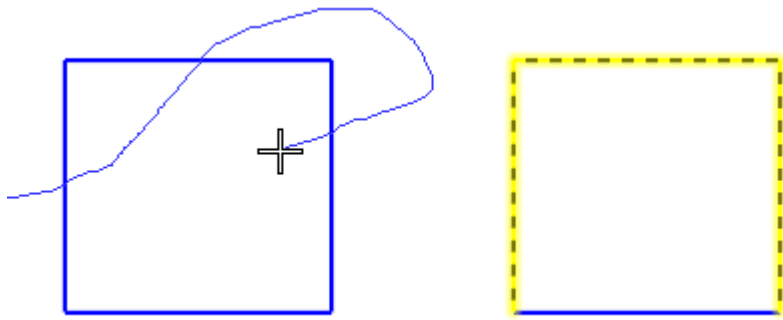
New Freehand Selection

Mastercam 2026 offers freehand selection of entities in the graphics window. To use this function:

1. Select **Freehand** from the Selection Bar.



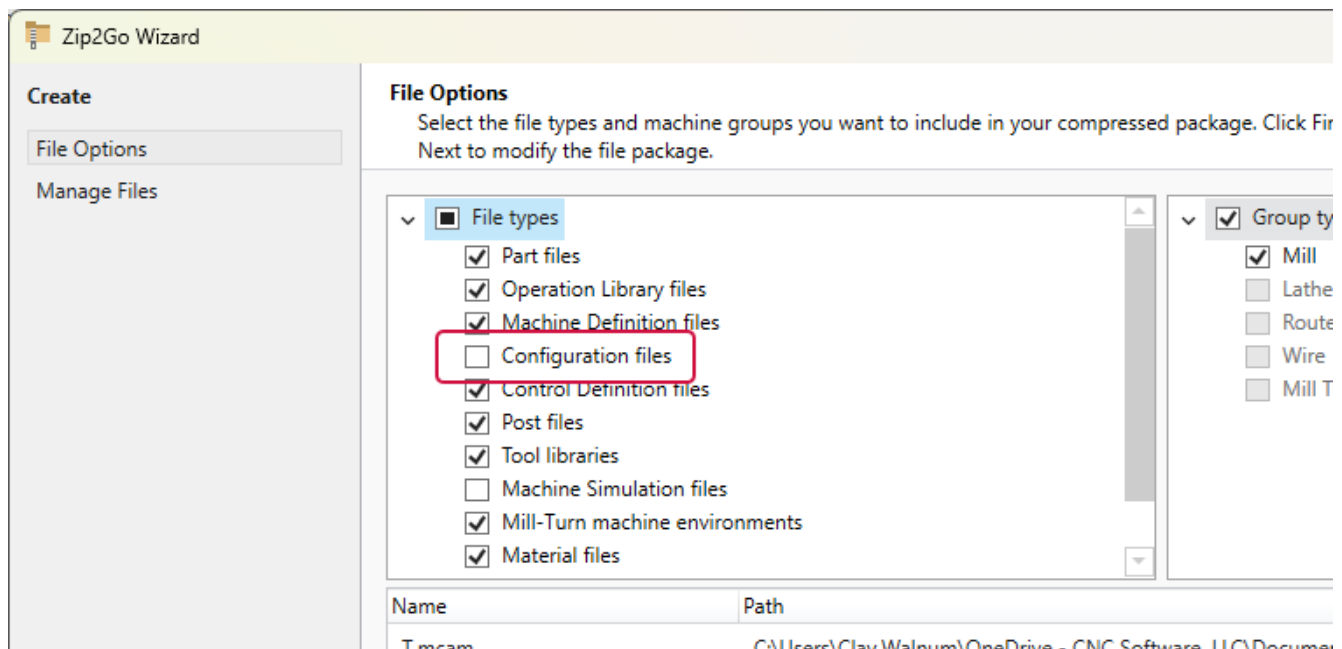
2. Hold down the mouse button and draw over the entities to select them.



3. Click to finish.

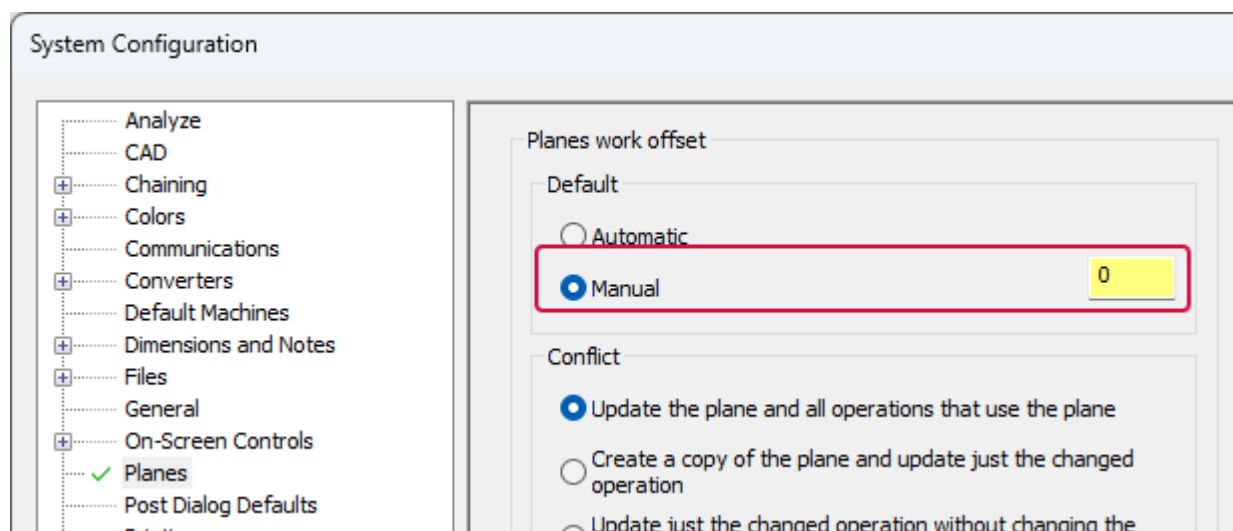
Include Configuration In Content Files

Mastercam now has an option in Zip2Go to include configuration files in MCAM-CONTENT files. In previous versions, Mastercam automatically added the configuration files. Now, **Configuration files** is an option in the **File Options** menu. By default, this option is disabled.



Set a Default Planes Work Offset

You can now set a default **Planes work offset** in your configuration. To do this, go to the **Planes** page of the **System Configuration** dialog box, and change the **Planes work offset** setting to **Manual**. You can then type an offset into the box to the right of the **Manual** button. Note that the new default offset will apply only to newly created operations; it will not change the work offset for existing operations.



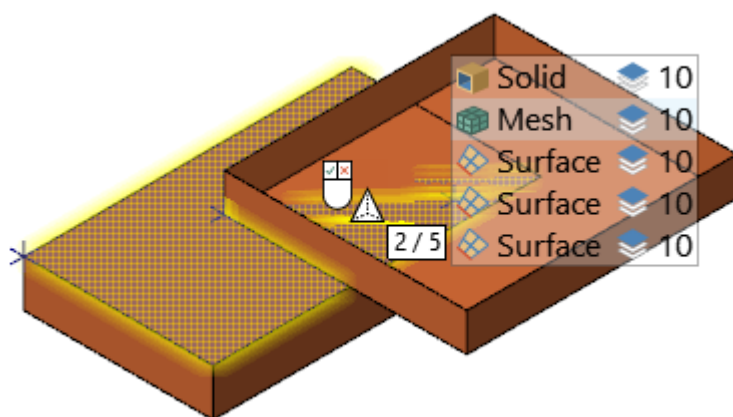
Verify Geometry Selection

The Verify Selection function has been updated to make geometry selection easier and more accurate. As in previous versions, when you move your mouse cursor over geometry in the graphics window, the cursor changes to show the type of geometry over which the cursor passes; a click selects that geometry.

In Mastercam 2026, if you hover over geometry, after a short delay, the cursor changes to show the number of selections (including hidden geometry) you can make.

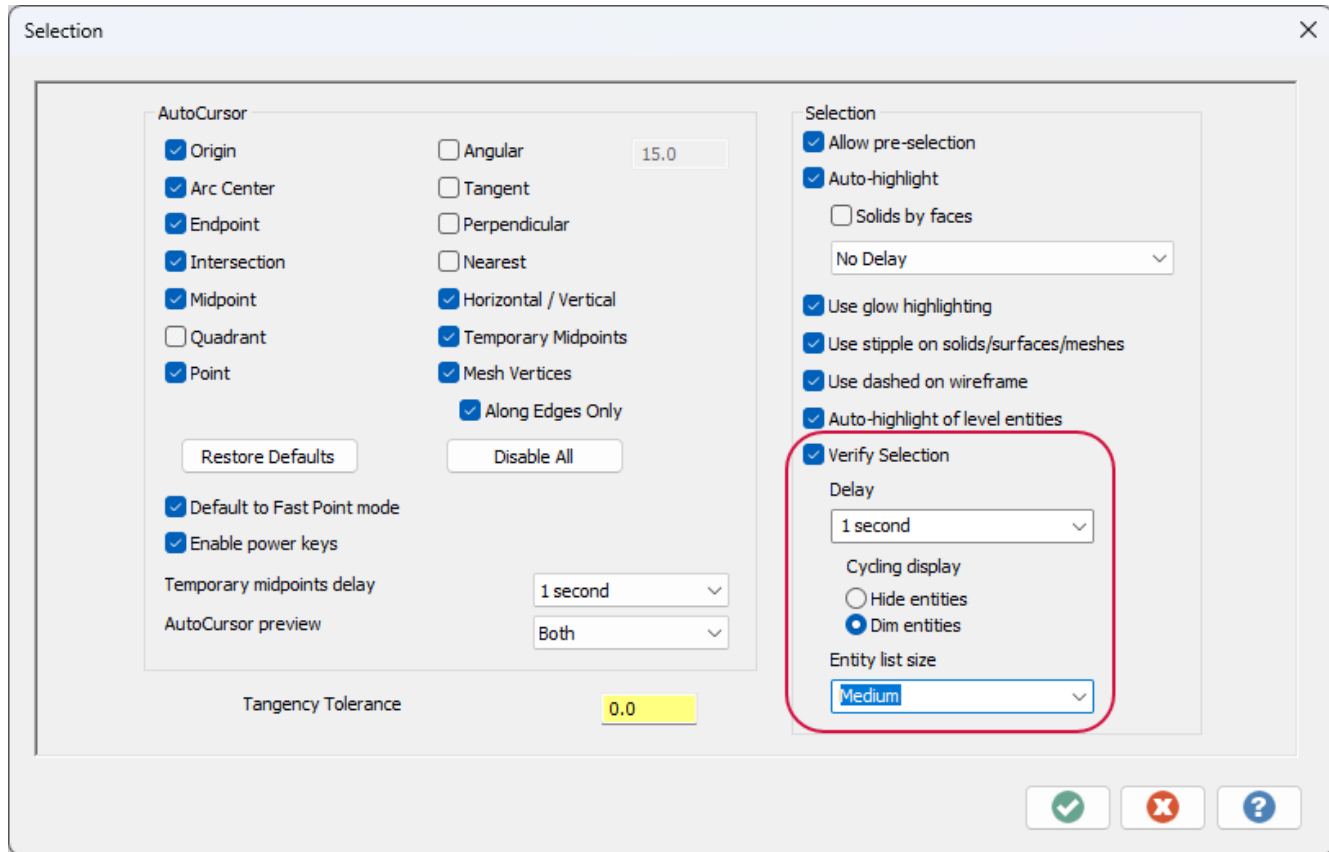
- Left-click to select the geometry that the cursor is on.
- Right-click to display a menu of possible selections, as shown in the following image.

From the right-click menu, choose the entity you want, and then double-click to finalize the selection. Cycle through the choices by scrolling the mouse wheel, by right-clicking, or using the cursor keys. The entity list also displays the levels on which geometry appears.



NOTE: Use Quick Masks to limit the types of geometry the selection verification shows in the geometry list. For example, if you select the surface-only and mesh-only Quick Masks, only those types of entities will appear in the geometry list.

Customize the selection function by choosing **Selection Settings** in the Selection Bar. In the **Selection** dialog box, you can change the delay time, how entities display, and the size of the menu. These customizations are also available in the **Selection** page of the **System Configuration** dialog box.



Machine Systems Enhancements

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

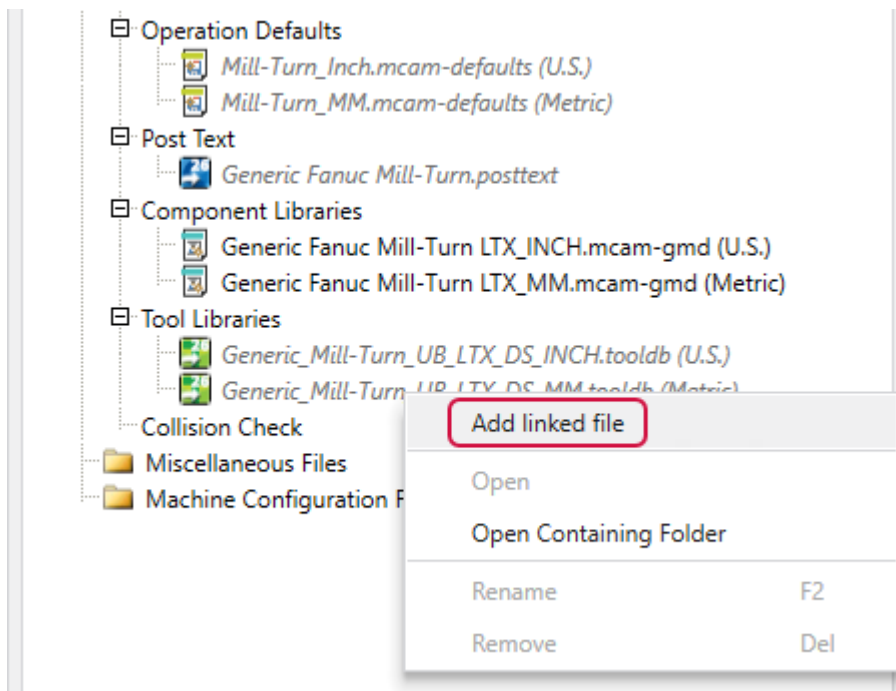
Link to External Files from the Machine Explorer

Normally, all of the individual files listed in the Machine Explorer are serialized into the .machine file and are all saved together. However, some users prefer to use tool libraries, defaults files, or other resources that are stored on a server or central workstation. For these cases, Mastercam 2026 lets you include a link to the external file in your .machine file, rather than the file itself.

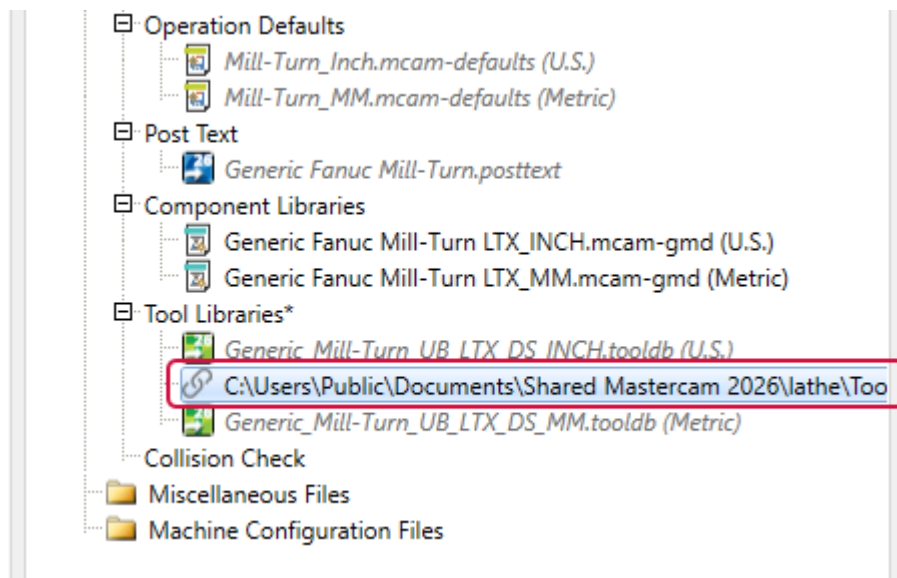
This functionality is supported for the following file types:

- Tool libraries (*.tooldb)
- Operation libraries (*.mcam-operations)
- Component libraries (*.mcam-gmd)
- Operation defaults files (*.mcam-defaults)

To create the link, right-click on the file that you wish to replace, and choose **Add linked file**. In this example, the metric tool library is being replaced.



Mastercam prompts you to select the new file. When you return to Code Expert, Mastercam displays the file with a special icon.

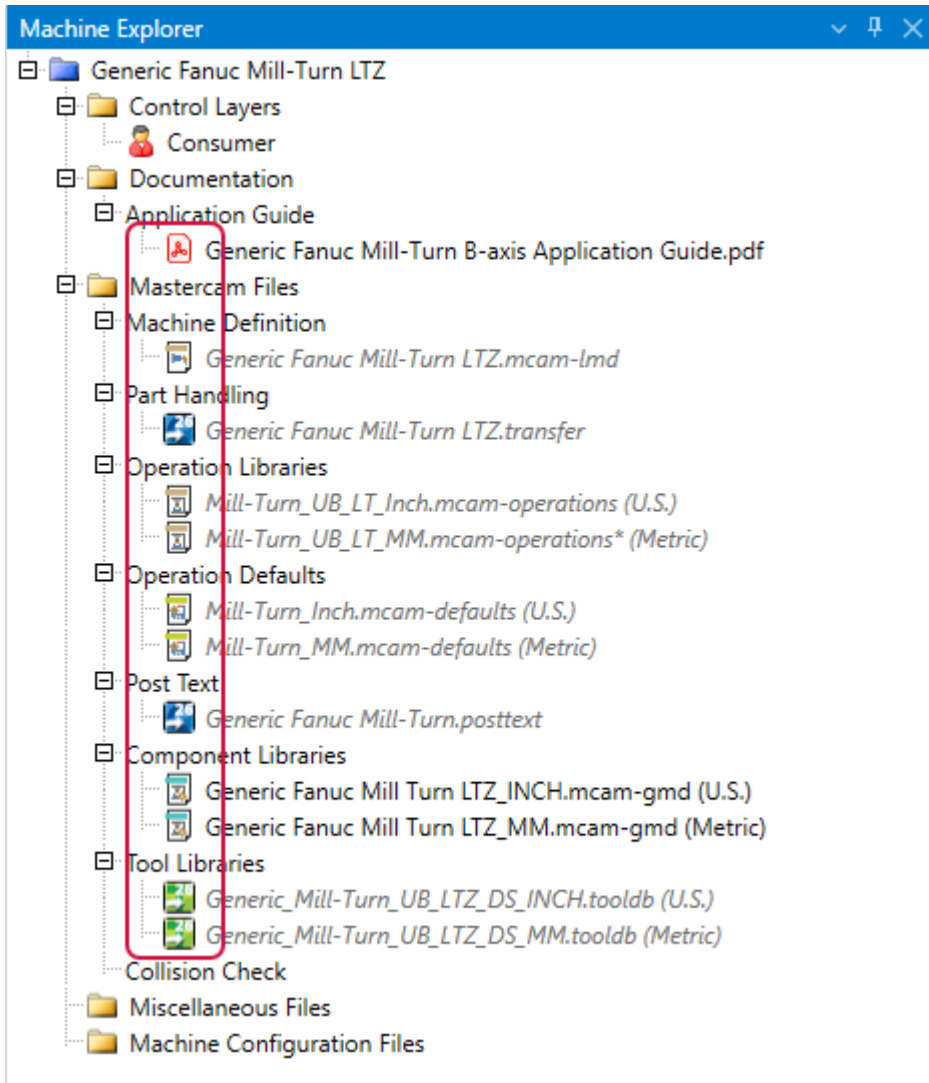


- You can replace each of the U.S. (inch) and metric files individually.
- Users with a **Consumer** license can add links to files even though they cannot normally edit or replace the original.

The original files remain in the .machine file. If a user loads the .machine file and Mastercam cannot find or access the linked file, the original file will be used as a fallback.

Mastercam-Specific Icons in Code Expert

Code Expert in Mastercam 2026 features new and refreshed icons for the Machine Explorer interface. Previous icons were generic and did not conform to modern standards. Code Expert now displays standard Windows icons for file types such as PDF files. Additionally, new easy-to-understand icons have been created for Mastercam-specific file types. The new icons provide for a more intuitive experience in Code Expert and will help users manage the many file types included in a typical .machine file.



General Enhancements

Listed below are general enhancements to Mastercam.

Mastercam Art Discontinued

We have removed Mastercam Art from Mastercam 2026. The product has been discontinued and is no longer supported.

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