

WHAT'S NEW IN MASTERCAM 2023

March 2022 © 2022 CNC Software, LLC – All rights reserved. Software: Mastercam 2023

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/legal/

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

Table of Contents	
Introduction	
Release Highlights	
Mastercam Resources	
Contact Us	
Milling Enhancements	11
New Behavior for the Apply Button in Milling Toolpaths	11
Re-designed Tool Pages and Tabs for Milling Toolpaths	12
New Mill Tool Icons	14
Overall Process Improvements	14
3D Toolpath Enhancements	15
Detecting Undercut Stock When Machining	15
Identifying and Machining Material in Corners	16
Creating Automatic Boundaries for Toolpath Containment	16
Machining with Zigzag Motion	17
More Powerful and Flexible 3D Roughing Linking	19
Preventing the Tool from Retracting Between Passes	
Trimming Passes to Fit Transition Moves	21
Rest Roughing Against Stock Model Improvements	21
Automatically Detecting and Including Stock for Z Depths	
Setting the Entry and Exit Move Helix Arc Height	23
Adding Stock Awareness to Holder Checking	
Machining from Bottom to Top	24
Machining at Maximum Depth	25
2D Toolpath Enhancements	
Removing Restrictions with Slot Mill Geometry	25
Overriding the Feed Rate for Profile Ramp Entry and Exit	26

Dragging and Dropping for the Hole Making Toolpaths	
Multiaxis Toolpath Enhancements	
Consolidating Multiaxis Toolpaths	
New Feed Rate Control Page	
Replacing Rapid Motion with Feed Rate	
Setting Feed Rates as a Percentage or Direct Value	
Controlling the Guide Curve Across Machining Geometry	
Specifying Connection Moves Between Cuts	
Controlling Rapid and Linking Moves	
Controlling the Tool Contact Point	
Driving the Tool Axis Control from Another Surface	
Turning Enhancements	
New B-Axis Contour Turning Toolpath	
Automatically Create the Toolpath Motion	
Manually Control the B-Axis Angles	
Adjusting the Toolpath Motion for Both Modes	
Supporting Multiple Steady Rests	
Using Multiple Standalone Steady Rests	
Using Tandem Steady Rests	
Using Multiple Tandem Steady Rests	
Programming Multiple Steady Rests	
Incorporating Multiple Steady Rests in Operation Strategies	
Setting a Base Spindle for Steady Rest Operations	
Supporting Transform Subprograms in Mill-Turn	
Creating Part Handling Operations That Use the Active Tool Plane	
Displaying the Stock Preview Directly from the Ribbon	
Designing Enhancements	
Organizing Your Solid Models into Groups	
Improving the Selection Workflow	
Creating Additional Geometry Types	51

Creating a Cylinder with Unknown Base Points	
Mesh Enhancements	55
New Offset Mesh Function	55
Automatically Check and Fix Mesh Results	
Check Mesh Enhancements	
Deleting Problematic Facets	
Detecting Folded Facets	
New Trim to Chain Function	
Improving Your Facet Selection	
Automatically Split Meshes	61
Selecting Edges to Fill Holes	
Wireframe Enhancements	
Spline Enhancements	
Converting Complex Spline Shapes	
Modifying Spline Tangencies	
Creating Planes and Points for Each Curve Slice	64
Automatically Determine Z Depth	
Linear Extension to Splines	
2D/3D Construction Mode Support	
Model Prep Enhancements	
Detecting Holes in Solid Bodies	
Positioning Solid Bodies	
Improving the Workflow for Split Solid Face	
Providing Consistent Controls and Greater Flexibility	
Creating Chamfers with Push-Pull	71
Surfaces Enhancements	
Creating Consistent Flowlines with Surface Ruled/Lofted	
Supporting Multiple Flow Surfaces in Overflow UV	
Improving Selection for Trim to Surface and Trim to Curves	
Drafting Enhancements	77

Top-Level Editing of Drafting, Label, and Note Entities	77
Modifying Your Drafting Font Directly	
Remembering Previous Note Text	
Improving Visibility for Drafting Dimensions	
Including Dimensions When Aligning	79
Transform Enhancements	
Projecting Dimension Entities	
Easily Enter X, Y, and Z Values While Manipulating Geometry	80
Improving the Plane Selection for Translating to Plane	81
Remembering the Last Dynamic Transform Operation	
Simulation Enhancements	
Appending Operations to a Current Mastercam Simulator Session	83
Improving the Overall Processing Speed	
Setting Your Collision Checking Options	85
Viewing Rotational Axis Jumps During Simulation	
Toolpath Utility Enhancements	
Displaying Dirty Toolpath Motion in the Graphics Window	
Copying and Moving Operations with the Left Mouse Button	
Importing Toolpath Operations from Recent Files	
Viewing Plane Name Tooltips	
System Enhancements	
Dynamic Plane Enhancements	91
Creating Associative Dynamic Planes	91
Creating Dynamic Planes with Normal Orientation	
Grouping Viewsheets in Mastercam	
Configuration Enhancements	
Configuring Your Planes Settings	
Adjusting the Degree of Rotation for the Graphics View	96
Customizing the Geometry Display	96
Including Comments with Defaults from a Previous Operation	

Selecting a Language for the Mastercam Installation	
Auto Scaling of Temporary Planes During Functions	
Importing Viewsheets for Merged Files	
Customizing the Right-Click Menu	
Displaying Tooltips Longer	
Machine Systems Enhancements	
New Workflow and Interface for Managing Your Machine Groups	
Encrypting Your Posts	
Improving the Experience of CD Compare and MD CD PST Rename	
File Management Enhancements	
Compressing and Extracting Mastercam Files	
Better Quality of Imported and Exported CAD Files	
Improved Quality for Importing and Exporting of 3D Annotations	110
Exporting Drafting Entities to STEP Files	
Quickly Find a 3D Annotation Associated with an Entity	
Enhancements When Importing Solid Models	
Healing Stitched Models	111
Automatically Detect Hole Features	
Improved Management of Assemblies, Sub-assemblies, and Component Parts	111
Newly Supported File Formats	111
Adding Migrated Machine Definition Files to the Machine List	112
Exporting a Stock Model Into Different File Formats	
General Enhancements	
New Mastercam Learning Edition	
Improving Efficiency When Using Analyze	
Combining Duplicate Error and Warning Messages	116
Working with the NET-Script Editor	116
Turning a NET-Script into a Mastercam Function	117
Debugging Mastercam .NET-Scripts in Visual Studio Code	
Storing Third-Party Add-ins Outside of the Mastercam Installation	

Previewing AutoCursor Positions		
Improved and Renamed Mastercam Do	ocumentation	
Mastercam Resolved Issues		121
Mastercam Basics Tutorial		121
Geometry and Toolpath Nesting Enhan	icements	
Nesting Multiple Corners		
Improved Error Handling and File Su	ipport	
Toggling the Display of Mesh Facet Edg	ges	123
Automatically Adjust the Opacity of the	on-screen Ruler	
Posts and Machine Environments		
New Posts and Machines for Mastercar	m 2022	125

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.



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.

2D Toolpaths - Slot mill				×
Y 🔒 🌃 🥗				
Toolpath Type Tool Holder Cut Parameters Rough/Finish Depth Cuts Break Through Linking Parameters Home / Ref. Points	Compensation type Compensation direction Tip compensation	Computer V Left V Tip V	Entry/exit arc sweep	180.0
Cplane / Tpl Front Axis Combi Default (1) ✓ = edited ⊘ = disabled			Generate toolpath	

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, userfriendly layout.

Model Geometry	Stat	tus Tool Number	Assembly N	Tool Name	Holder Name	Diameter	Corner Radius	Length	Flutes
Toolpath Control	7 🗸	271		END MILL	B2C3-0020	12.0	1.0	19.0	4
Tool									
Holder									
Stock									
Cut Parameters									
Linking Doromotors									
LINKING Parameters									
Arc Filter / Tolerance									>
Planes				Filter active					
Coolant	Colo			Filter					
Canned Text	Sele	Lt library tool		Filter					
Misc Values									
Axis Control		12.0		10					
Axis Combination	I ool diame	ter: 12.0	Corner radi	us: 1.0			Spindle	direction:	cw 🗸
Rotary Axis Cont	<u> </u>				 Eacd rate	. 296	5 9136 Coindle	conced.	8912
Quick View Settings	Technome	END MTU		12 / P1 0	reeurate	: 200	Spindle	speeu:	0712
Tool END MILL WI	Toorname		WITT RADIOS -	127 K1.0	EPT:	0.0	832 CS		335.9849
Tool Diameter 12	 Tool #:	271	Length offs	set: 271					
Corner Radius 1					Plunge rat	te: 100	0.0 Retrac	t rate:	2000.0
Feed Rate 2965.91	Head #:	0	Diameter o	ffset: 271	Force 1	tool change	Rap	oid retract	
Spindle Speed 8912	Tool I	Inspection / Change				-			
Coolant Off	Eorce re	tract every	10000.0	Millimeters	Comment				
Tool Length 60	, ordere	state or of yr							~
Length Offset 271			40.0	Minutes					
Diameter O 271									
Cplane / Tpl Front									· ·
· · •					To bat	ch			

Toolpat	h paran	neters Surface	e parameters 🛛 Fi	nish parallel para	ameters					
St	atus	Tool Number	Assembly Na	Tool Name	Holder Name	Diameter	Comer Radi	us Length	Flutes	
		241		BALL-NOSE		12.0	6.0	26.0	4	
<										>
Sele	ct librar	y tool		Tool filter		Coo	lant			
		10.0					S	pindle directio	n: CW	\sim
Tool diar	meter:	12.0	Corner radius	6.0	Feed rate	e: 🚺	6207.24 S	pindle speed:	7958	
Tool nam	ne:	BALL-NOS	SE END MILL - 1	2	FPT:		0.195 C	S	300.0189	•
Tool #:		241	Length offset	241	Plunge ra	ite:	1000.0 F	etract rate:	2000.0	
Head #:		0	Diameter offs	et: 241	Force	tool change	• E	Rapid retrac	t .	
					Comment					
										· ·
J	Axis Cor	mbo's (Default (1)) M	isc values		\sim	Tool displa	ay	Ref point	
🗌 To I	batch		H	lome pos	Rotary as	xis	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.

Tool List Filter		>	×
Tool Types	Tool Diamet	ter	
	Radius Type	e 🗹 Full 🗹 Corner	
	Tool Materia	ial S 🔽 Ceramic	
		arbide	
	All	None Copy job setup mat	
All None			
Operation masking Unit masking	9		
No operation masking \lor No unit mas	sking \checkmark		
Reset all		22	

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.



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**.

I 3D High Speed Toolpaths - A	rea Roughing		×
🎙 🗊 🔒 🖬 🗐	*		
Toolpath Type Model Geometry Toolpath Control Tool Holder Good Cut Parameters Orchoidal motion Steep / Shallow Linking Parameters	Rest material Compute remaining stock from: Previous operations One other operation Detect undercut stock Roughing Tool Diameter: 25.0	Image: State of the state	_

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.

3D High Speed Toolpaths - Dynamic OptiRough					
🎙 🗊 🔒 🜃 🗏 G	*				
Toolpath Type Model Geometry Toolpath Control Tool Holder Stock Cut Parameters Steep / Shallow ∎ Linking Parameters	Containment boundary Boundary chains: Automatic boundary: Include stock	(1) <table-cell> 🛞 Silhouette 🗸</table-cell>			

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.

3D High Speed Toolpaths - Area Roughing						
Y 🗊 🔒 👪 🗐 🗐	₽					
Toolpath Type Model Geometry Toolpath Control Tool Holder	Cut style Closed contour direction Open contour direction	Climb ~ One Way ~				
Stock Cut Parameters Cut Parameters Cut Parameters Steep / Shallow	Tip compensation	Tip ~				

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.



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.

Toolpat	h Type 🔺	Retract		Leads		
Model G	Seometry	Classing alanas	50 0 N	Mathad		
Toolpati	h Control	Clearance plane:	50.0	Method:	Same lead in and o	ut Lood Out
100l		Position:	Absolute		Lead In	Lead Out
Holder			 Incremental 	Type:	Horizontal \sim	Horizontal 🗸 🗸
Stock		Туре:	Minimum Vertical Retract	Distance:	0.0	0.0
Cut Para	ameters			Current and a	00.0	00.0
V Troo	choidal motion	Part clearance:	4.0	Sweep angle:	90.0	90.0
i Stee	ep / Snallow			Radius:	1.0	1.0
LINKING	Parameters	Arc fit radius:	2.0			
Arc Filte	er / Tolerance	Feed rate:	2500.0	Ramp angle:	0.0	0.0
Planes				Secondary L	eade	
Coolant	t I	Linear entry/exit:	0.5		2003	
Canned	Text			Method:	Same lead in and o	ut
Misc Val	lues	Keep tool down within			Lead In	Lead Out
Axis Co	ntrol 🗸	Percent of tool diameter:	100.0	Type:	Vertical V	Vertical V
c	>	OBistanau	12.0			
viel: View Cattin		O Distance:	12.0	Distance;	0.0	0.0
uick view Seturi	iys	Transition		_	00.0	00.0
Tool	END MILL WI	Tansidon		Sweep angle;	90.0	90.0
Tool Diameter	12	Micro lift retract		Radius:	2.0	2.0
Corner Radius	1	Never				
Feed Rate	2965.91	Miero lift distance :	0.25	Ramp angle:	0.0	0.0
Spindle Speed	8912	Micro In Custance:	0.23			
Coolant	Off	Back feed rate:	2500.0	Fitting		
Fool Length	83			Type:		Minimize Trimming
Length Offset	271	Keep tool down between Z de	pths			Ĭ
Diameter O	271	Eeed rate:	110.0	Manager		1.0
Cplane / Tpl	Тор	e l'éculate.		Maximum trimmii	ng distance:	1.0
Formula File	Default.Formula	Apply leads		Fit transition	s	
Axis Combi	Default (1)					

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.

···· ✓ Toolpat	n Type	<u>^</u>	Retract	
Model G	eometry			
Toolpat	n Control		Clearance plane:	50.0 K3
Tool			Position:	Absolute
Holder				 Incremental
			Turner	Minimum Vertical
Stock			Type:	Retract ~
Cut Para	ameters			
- O Troc	hoidal motion		Part dearance:	2.0
Stee	ep / Shallow			
Emme Linking P	Parameters		Arc fit radius:	4.0
				3500.0
Arc Filte	r / Tolerance		Feed rate:	2500.0
Planes			Linear entry/exit:	0.5
Coolant				
Canned	lext		Keep tool down within	
Misc Val	ues			
Axis Cor	100	×	Percent of tool diameter:	100.0
<	>		O Distance:	12.0
Ouick View Setting	ns		O bistance.	1210
	90 10 0 1 1 1 1 1		Transition	
100	18 Spherical /			
Tool Diameter	18		Micro lift retract	
Corner Radius	9		When exceeding a distance	
Feed Rate	120		Micro lift distance	0.0
Spindle Speed	1000		Micro Int distance:	0.0
Coolant	Off		Back feed rate:	240.0
Tool Length	100			
Length Offset	1		Keep tool down between Z dep	ths
Diameter O	1			110.0
Cplane / Tpl	Тор		✓ reed rate:	110.0
Formula File	Default.Formu	ila 📗	Apply leads	
Axis Combi	Default (1)			

Keep tool down off



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.





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.

3D High Speed Toolpaths - Waterlin	e
🎙 🗊 📙 👪 🗐 🖷	
Toolpath Type Model Geometry Toolpath Control Tool Holder Stock	Angle From 0.0 To 90.0
Arc Filter / Tolerance Planes Coolant Canned Text Misc Values Axis Control Axis Combination Rotary Axis Control Quick View Settings Tool BALL-NOSE E	Z depth Minimum depth 0.0 Detect limit Automatically detect depth Include stock Adjust for stock to leave Maximum depth -45.62959 Detect limit Automatically detect depth Include stock Adjust for stock to leave

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.

3D High Speed Toolpaths - Spire			×
🕴 🗊 🔚 🔛 🗏 🖻			
Toolpath Type Model Geometry Toolpath Control Tool Holder	Leads Method:	Same lead in and o Lead In	ut Lead Out
Stock	Distance:	0.0	0.0
Steep / Shallow Linking Parameters	Sweep angle: Radius:	90.0	90.0 2.0
Arc Filter / Tolerance Planes	Ramp angle:	10.0	10.0
Coolant Canned Text	Secondary Le	ads	
Misc Values	Method:	Same lead in and o	ut
Axis Combination Rotary Axis Control	Туре:	Lead In Horizontal V	Horizontal
Quick View Settings	Distance:	0.0	0.0
Tool 20. FLAT E	Sweep angle:	90.0	90.0
Corner Radius 0	Radius:	2.0	2.0
Feed Rate 7.1625 Spindle Speed 3500	Ramp angle:	10.0	10.0

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.

3D High Speed Toolpaths - Area Roughing						
Y 🗊 🔒 👪 🛙		*				
Toolpath Type Model Geometry Toolpath Control Tool Holder Stock Cut Parameters Cut Parameters Cut Parameters Steep / Shallow Linking Parameters Entry Motion Home / Ref. Points Arc Filter / Tolerance Planes Coolant	^	(Default Holder) B2C3-0016 B2C3-0020 B2C3-0020 B2C3-0032 B2C4-0011 B2C4-0016 B2C4-0020 B2C4-0032 B2C4-0032 B2C4-1000 B2E4-0125 B2E4-0187 B2E4-0250 B2E4-0312	^	Collision checking Trim to avoid gouge Tilt to avoid gouge Additional avoidance (0) Max. tilt angle 1.0 Shank clearance 0.0 Holder clearance 1.0 Clearance on holder bottom		
Canned Text	v	B2E4-0312	×			

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.



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.

loolpath Hole De	finition # ×
Selection Advanced	o 😒 📀
Features	۲
*/ Туре	Diameter
Simple Ø 10.00 1	10.0
Simple Ø 10.00 2	10.0
Simple Ø 10.00 3	10.0
Simple Ø 10.00 4	10.0
Simple Ø 10.00 2	10.0
Simple Ø 10.00 6	10.0
Simple Ø 10.00 7	10.0
Simple Ø 10.00 8	10.0
•	•

_

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:

📧 Multiaxis Toolpath - Unified					
🎙 🔚 💵 😁					
🛛 🕐 🔨 Toolpath Type	Pattern				
Tool	Name	Style		Entities	Action
Holder	Surface	Morph	\sim	1	×
Stock	Surface	In Morph	\sim	1	×
Cut Pattern					
Advanced Options					
Margins Darameters For St					
Tool Axis Control					
Collision Control	I I I I I I I I I I I I I I I I I I I			6	8

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.

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

🖉 🖉 Toolpath Type	Cut feed rate control				
Tool Holder	First cut feed rate %	100.0 % 2.984375			
Stock	\checkmark Tool contact based feed rate optimization				
+ Cut Pattern	Minimum feed rate %	10.0 % 0.298438			
Tool Axis Control					
Feed Rate Control	Feed control zone				
Roughing					
Utility	Geometry	45			
- Inde	Offset	0.0			
Additional Settings	Inside feed rate	100.0 % 2.984375			
	Outside feed rate	100.0 % 2.984375			
Quick View Settings	Surface radius based feed optimization				
Tool 12. BALL EN A	R= • Radius flat	Feed rate % 100.0	Linking feed rate control		
Tool Diameter 12 Corner Rad 6			Custom feed rate for clearance blend spline	3000.0	
Feed Rate 2.98438	R. Radius 10.0	Feed rate % 10.0	Replace Rapid with Feed	9999.0	
Coolant Off			Area links	1000 mm/mi	in
Tool Length 24	R Radius 5.0	Feed rate % 2.0		1000	
Diameter Q., 246	× – – – – – – – – – – – – – – – – – – –		Links between slices	1000 mm/mi	ara.
Colane / To Top	R=0		Links between regions	1000 mm/mi	rin
	Radius 0	Feed rate % 1.0			
= edited					

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

E.

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.

📧 Multiaxis Toolpath - Unified	💽 Multiaxis Toolpath - Unified							
🎙 🔚 🌃 🥗								
🛛 🕐 Toolpath Type 🔥	Cut feed rate control							
Tool Holder	First cut feed rate %	100.0 % 2.984375						
	Tool contact based feed rate optimization							
Stock								
Cut Pattern	Minimum feed rate %	10.0 % 0.298438						
Advanced Options								
Margins								
Parameters For Su								
	Feed control zone							
E. Collision Control		N						
Einking	Geometry	13						
Feed Rate Control	Offerst	0.0						
Roughing	Offset							
Utility	Inside feed rate	100.0 % 2.984375						
Misc 🗸								
< >	Outside feed rate	100.0 % 2.984375						

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.

👘 🔗 Toolpath Type 🔺	Projection direction	
Tool	N	
Holder	No projection	~
Stock	Margin	
Cut Pattern		
Advanced Options	Start type	Start on guide \sim
Containment		
Guide Curve - Adv	Offset from curves	0
Machining Geomet		
Tool Axis Control		
Collision Control		
Einking	Guide-curve to containment tolerar	0
Feed Rate Control		
Roughing	Add internal tool radius	
Utility		
< >	Input curve	
Quick View Settings	Extend open input curves	
Tool 12, BALL EN A		
Tool Diameter 12		
Corner Rad 6	Propagation direction relative to qui	
Feed Rate 2,98438	repugador an eccorrelative to gain	
Spindle Speed 0	Propagation direction	Both ~
Coolant Off		
Coolant On		

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.

Multiaxis Toolpath - 3+2 Automatic Roughing						
🎙 🔚 🌃 🥶						
Toolpath Type Model Geometry	Entry/Exit First entry	Approach from clearance area	~	Use ramp	~	
Tool Holder	Last exit	Retract to clearance area	~			
Stock		Start from home position		Home position	$[] \begin{tabular}{ c c c c } \hline \\ \hline $	
Tool Avis Control	Default links					
Collision Control	Within group	Direct	~	Don't use ramp	~	
Enking Feed Rate Control	Between groups	Retract to feed distance	~	Use ramp	~	
Roughing	Small gap size	0 In % of tool diameter		0 O as val	ue	

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**.

O Toolpath Type	Clearance plane		
Tool			
Holder	Туре	Cylinder	\sim
	Direction	Line	~ 13
Tool Axis Control	Radius	Uses defined	200
Linking	Raulus	User defined	200
Feed Rate Control	Through point	User defined	~ N
Misc	moughpoint	osci denned	- 43
	Rapid distance		20
Additional Settings	Rapid distance		
	Feed distance		10
	r ced distance		
	Air move safety distance		10
	· · ·		
	Smooth corners		0
	Links		
Quick View Settings	Links	Petract to dearance area	~
Quick View Settings	Links Type	Retract to clearance area	~
Quick View Settings	Links Type	Retract to dearance area	~
Quick View Settings Tool 12. BALL EN A Tool Diameter 12	Links Type	Retract to dearance area	~
Quick View Settings Tool 12. BALL EN A Tool Diameter 12 Corner Rad 6	Links Type	Retract to dearance area	~
Quick View Settings Tool 12. BALL EN A Tool Diameter 12 Corner Rad 6 Feed Rate 6207.24	Links Type Lead in/out	Retract to clearance area	~
Quick View Settings Tool 12. BALL EN. ▲ Tool Diameter 12 Corner Rad 6 Feed Rate 6207.24 Spindle Speed 7958	Links Type Lead in/out Radius	Retract to dearance area	~ 3
Quick View Settings Tool 12. BALL EN. ▲ Tool Diameter 12 Corner Rad 6 Feed Rate 6207.24 Spindle Speed 7958 Coolant Off	Links Type Lead in/out Radius	Retract to dearance area	~ 3
Quick View Settings Tool 12. BALL EN A Tool Diameter 12 Corner Rad 6 Feed Rate 6207.24 Spindle Speed 7958 Coolant Off Tool Length 24	Links Type Lead in/out Radius Minimum radius	Retract to clearance area	× 3 1
Quick View Settings Tool 12. BALL EN A Tool Diameter 12 Corner Rad 6 Feed Rate 6207.24 Spindle Speed 7958 Coolant Off Tool Length 24 Length Offset 246	Links Type Lead in/out Radius Minimum radius	Retract to dearance area	~ 3 1
Quick View Settings Tool 12. BALL EN A Tool Diameter 12 Corner Rad 6 Feed Rate 6207.24 Spindle Speed 7958 Coolant Off Tool Length 24 Length Offset 246 Diameter O 246	Links Type Lead in/out Radius Minimum radius	Retract to dearance area	× 3 1
Quick View Settings Tool 12. BALL EN A Tool Diameter 12 Corner Rad 6 Feed Rate 6207.24 Spindle Speed 7958 Coolant Off Tool Length 24 Length Offset 246 Diameter O 246 Colane / To Too	Links Type Lead in/out Radius Minimum radius Advanced options for clear	Retract to dearance area	× 3 1
Quick View Settings Tool 12. BALL EN Tool Diameter 12 Corner Rad 6 Feed Rate 6207.24 Spindle Speed 7958 Coolant Off Tool Length 24 Length Offset 246 Diameter O 246 Colane / To Top	Links Type Lead in/out Radius Minimum radius Advanced options for clear Angle step for rapid moves	Retract to dearance area	× 3 1 5
Quick View Settings Tool 12. BALL EN Tool Diameter 12 Corner Rad 6 Feed Rate 6207.24 Spindle Speed 7958 Coolant Off Tool Length 24 Length Offset 246 Diameter O 246 Colane / To Top	Links Type Lead in/out Radius Minimum radius Advanced options for clear Angle step for rapid moves	Retract to dearance area	× 3 1 5 5
Quick View Settings Tool 12. BALL EN A Tool Diameter 12 Corner Rad 6 Feed Rate 6207.24 Spindle Speed 7958 Coolant Off Tool Length 24 Length Offset 246 Diameter O 246 Colane / To Top < = edited	Links Type Lead in/out Radius Minimum radius Advanced options for clear Angle step for rapid moves Angle step for feed moves	Retract to dearance area	 ✓ 3 1 5 5

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.

💎 🔗 Toolpath Type	Tilting		
Tool			
Holder	Machining type	5 axis (simultaneous)	~
Cut Pattern	Strategy	Fixed to main axis	\sim
Tool Axis Control			
Linking			
Feed Rate Control			
Misc	Tilt range		
Additional Settings			
	Maximum angle step		3.0
Quick View Settings	Clearances		
Tool 12, BALL EN.	Clearances		
Tool Diameter 12	Holder		2.0
Corper Rad 6			
Feed Rate 6207.24	Shank		0.5
Spindle Speed 7958	chard days		0.2
Coolant Off	shoulder		0.2
Teel ength 24			
Length Officet 246	Tool contact		
Discustor O 246		sist on some (adiadae (C/)	50.0
Diameter O 246	/ Fixed tool contact p	oint on cone/cylinder (%)	30.0
Colone / To Too			

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.

Multiaxis Toolpath - Unifie	d		
🕴 📙 👪 🗃			
Toolpath Type Tool Holder	Output format	5 Axis	~
Stock	Tool axis control	3 Tilt relative to reference surface	~
Cut Pattern Tool Axis Control Collision Control	Reference surface		(1) 🞝 🚱
Linking Feed Rate Control Roughing	Tilt angle	0	

TURNING ENHANCEMENTS

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

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**.

111

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.

(2)	la 🙆	📀 🙆
	Basic Motion Control	
ļ Ī	Tool Axis Control Automatic: Manual:	٢
111	Tool Angle Limits	۲
	Minimum: 0.0	• ‡
	Maximum: 90.0	- ‡
Ĕſ	Selection	۲
6	Tool Offset Angle	$\overline{\bullet}$

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.

(2)		📀 🙆
	Basic Motion Control	
,	Tool Axis Control	۲
Ī	Manual:	
111	Tool Angle Limits	\odot
М	Selection	٢
Ĕſ	Line 2	
4		
111	Tool Offset Angle	۲
nc.	0.0	- ‡



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.

(2)	le 💿	📀 😰
	Advanced Motion Control	
7	Cut Tolerance	۲
Ŧ	0.02	• ‡
	Maximum Distance Between Points	۲
	2.5	• ‡
	Maximum Angle Step	۲
Ĕ	1.0	• ‡
,	Holder Clearance	۲
\mathcal{Q}	0.0	• ‡
111	Maximum Insert Depth	۲
1/10	0.0	• \$

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.

Steady Rest		×
Driven Component Steady Rest Steady Rest	Left Right	
Move		
Move type:	Follow	` ('_')
Spindle origin:	Machine Group-1.Turning.Left.U	pper 1 🗸 📥 🛶

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.

Linked	Components	
	Steady Rest - Right	
Comment:		

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:

Pickoff			×
Pickoff Setup Operations Strategy: Pickoff with Steady Rest Support Operations	∨ Name:	Pickoff with support	
 Undamp Steady Rest Left Position Steady Rest Left Clamp Steady Rest Left Undamp Steady Rest Right Position Steady Rest Right Clamp Steady Rest Right Clamp Steady Rest Right Align Spindles Spindle Sync Pickoff spindle - Unclamp and eject part Pickoff spindle - Move to clearance distance Pickoff spindle - Clamp and transfer stock Stock spindle - Unclamp Pickoff spindle - Clamp and transfer stock Stock spindle - Unclamp Pickoff spindle - Retract Linked Steady Rest - Follow Part Cancel Spindle Sync 	Comment: Linked Steady Rest - Fo Driven Component: Steady Rest - Lef Steady Rest - Rig Data stream: Feedrate: Linked Components Steady Rest - Righ	llow Part t ght Lower 300.0 inch/min	

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

Trim to Curves	д ×
?	o 😒 💿
Basic	
Surfaces	۲
Surface 1	
Chains	۲
Chain 1	
	O. O

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.

Spline Automatic	Р ×
(2)	🐼 🔗 😢
Basic	<u></u>
Manual Points	٢
Start point	
Direction point	
End point	
	8
Automatic Points	۲
Point 1	
Point 2	

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.

Primit	ive Cylinder		ч×
2		o 😔	
Basic A	dvanced		
Entity			۲
Type: 🖲	Solid		
0	Surface		
0	Mesh		
Base Poir	nt Definition		٢
Method:	Manual		
	○ Tangent		
Arc:	(Undefined)		
		0000	

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.

Wirefi	rame Su	irfaces	Solids	Model Pr	ер	Mesh
Sphere Cone Torus	Meshes from Entit	Offset es Mesh	Smooth Free Edge	Fill s Holes	Smooth Area	∆ ‡ Refine
	Crea	ate				

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.

Offset Mesh	Ψ×
3	🐼 📀 😒
Basic Advanced	
Entity	\odot
Method: Copy Modify	
Meshes	۲
Mesh 1	
Offset Distance	۲
0.1	•\$
Tolerance	۲
0.001	- ‡

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.



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.

Trim to Chain	Ψ×
(?)	
Operation Split mesh Create cap	۲
Target Meshes Mesh 1	۲
Chain Chain	 Ø

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.

Explode Mesh	Ψ ×
Basic Advanced	s S S
Entity Method: Copy Modify	۲
Selection Reselect	۲
Calculation Flatness: Edge angle tolerance: Merge percentage: Split disjointed mesh	2.0 * * 28.0 * * 0.05 * *
Results Mesh bodies: 0	۲

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.

Find Holes	Ŧ×
(2)	G 📀 😒
Basic	
Operation	۲
Type: Hole operations Create a single operation Plugs Suppress fillets Include split holes	
Selection	۲
Method: Bodies Faces	
Entities:	
Solid 1	
Radius	\checkmark
Associative Points	€
Plugs	\checkmark
Color	\checkmark

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.

Solid Position	Ψ ×
Basic	☞ ⊘ 😢
Operation	۲
Method: Move Copy	
Vector:	3
 ✓ Stop on contact bodies ✓ Rotate body 	
Body to Move	۲
Defined	3
Additional Geometry	۲
Contact Bodies	۲
Solid 1	

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.

Split Solid Face	Ψ×
3	國 📀 😰
Basic	
Operation	۲
Method: Wireframe	
Project using construction plane	
Extend to edge	
O Flowlines	
(i) U	
\bigcirc V	
OBoth	
Target: Defined	
Splitting Geometry	۲
Wireframe Entity 1	

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 onedistance 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.

Surface Ruled/Lofted	Ψ×
Basic	G 🕑 😢
Entity Method: O Ruled © Lofted Type: O Surface O Sheet solid O Mesh	۲
Chains Chain 1 Chain 2 Chain 3	۲
	64
Dynamic Sync O Average C Entity: O Minimum distance	۲
Flowline Analysis	٢
U flowlines: 0	\$
✓ V flowlines: 21	\$
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.

Trim to Curves	Т ×
Basic	☞ ⊘ 😒
Surfaces	۲
Surface 1	
Chains	۲
Chain 1	
	\mathcal{E}

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.

Trim to Surfaces	 7 ×
2	i 📀 📀 😢
Basic	
Entity Trim: Both sets First set Second set	۲
Surface Set 1	۲
Surface 1	
Surface Set 2	۲
Surface 1	

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.

Drafting			Р >	ζ
?			 Image: Second sec)
Basic Advanc	ed			
Entity			٢)
Method:	Auto			
	○ Horizontal			
	○ Vertical			
	O Parallel			
ion ap				
○ Left				
Leaders			•)
Format			•	١
Туре:	OLF SimpleSansCJK OC		~ T	l
Height:	5.0		- ‡ 🕞	l
Apply to dimensions and notes				
Centered text				
Decimal places:	2		÷)
v				

Drafting	τ, ×
Basic Advanced	I I I I I I I I I I I I I I I I I I I
Text	۲
Dimension: Ø 21.23	C
Tolerances:	
Reset	

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.



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.

Selection	
Reselect	E
Instances	
Number: 1	
Distance: Between	
○ Total	
Delta 🕥	
X: 0.0	
Y: 0.0 - + +	
Z: 0.0 • \$	
Verter Frem /Te	
Vector Prom/16	
Reselect	E
Polar	
Length: 0.0 -	-2
Angle: 0.0 -	

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.

Translate To Pla	ane 🛛 🕹 ×
Basic Advanced	😪 📀 😰
Entity Method: Copy Move	۲
Selection Reselect	۲
Instances Number: 1	 Image: A start of the start of
Source Plane	۲
	X Y Z 🖏 / ‡+ 🗟 🖄 🛅

Translate To Plane	
?	
Basic > Plane Selection	📀 📀
Named Planes	۲
Тор	A
Front	
Back	
Bottom	
Right	
Left	
Isometric	
Isometric reverse	
Trimetric	
Тор	
Front	~

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.

Chaining		
E Colors	Curve tolerance: 0.001	
Communications		
Converters	STL tolerance: 0.001	
Default Machines		
Dimensions and Notes	Workpiece tolerance: 0.002	
Files		
On-Screen Controls	Rotary arc smoothing: 10	
Planes		
Post Dialog Defaults		
Printing	lool starts from home position after tool change	
Reports	Go to home position on tool plane change	
±		
Selection	Skip drill cycle pecking	
Shading		
Element Simulation	Simulate disabled posting	
Solids		
Spin Controls		
Start / Exit		
Tolerances	Derault machine: 5_5AXGEN_VMCTTAB	/
r orer on reco		

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.

💥 📚 🏂 📮	▼ Verify	
File Home	View Verify	
Color Material Loop + Cutting + Playback Co	Collision hecking	strict Stop Restrict Drawing Display
	Coll	ision Detection Options X
 □ Tool vs. Fixture □ Tool vs. Workpiece □ Tool vs. Stock □ Mill tool holder □ Mill tool shank □ Mill tool shoulder □ Mill tool cutting length □ Lathe tool holder □ Lathe tool insert □ Stock vs. Fixture 		
		OK Cancel

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.

Simulator Options	×
Components Data Simulation	
Machine 5_5AXGEN_VMCTTAB Post Settings Use external post n n n n 	
Arc curve tolerance: 0.025 Tool Center Point Control 0.04 Maximum distance 0.04 Maximum angle change 3.0	

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 dropdown menu listing recently imported operations libraries. Improve your workflow by selecting an operations library from the list, instead of navigating with the file explorer.

🔳 Import Toolpath Operations 🛛 🗙
Source library: C:\Users\Public\Docume\MILL_MM.mcam-operations
Calculate speeds and feeds Assign current system tool and construction planes Import operation's geometry Disable duplicate tool checking Import operation groups
Machine Group-1 Properties D Toolpaths D Toolpaths D Toolpaths Art Toolpaths Art Toolpaths Surface Rough Toolpaths Surface Rough Toolpaths Multiaxis Toolpaths Multiaxis Toolpaths Art - Curve 5 Axis - [WCS: Top] - [Tplane: Top] At - Curve 5 Axis - [WCS: Top] -

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.

Toolpath Coordinate System		×
Toolpath Coordinate System Working Coordinate System Operation 1 - Fixture Plate Pla Operation 1 - Fixture Origin X -0.639752 Y -0.440397 Z 0.0 IIII &	Tool Plane Top re Plate Plane Origin X 0.0 Y 0.0 Y 0.0 ↓ X 0.0 ↓ Y 0.0 ↓ Y 0.0 ↓ Mork Offset Manual ● Automatic	Comp / Construction Plane Top Origin X 0.0 Y 0.0 Z 0.0 IIII
		⊘ 😢 🚱

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**.

Edit Common Parameters (Mill/Router)		×
Propagate Operations	Clearance 0.0	Rapid Retract
1: Contour (2D)	When No change 🗸 🗸	Force Tool Change
	Abs/Inc No change 🗸 🗸	~
	Retract 0.0	Batch
	Use No change 🗸 🗸	
	Abs/Inc No change 🗸 🗸	Coolant Cannad taxt
	Feed plane 0.0	
	Abs/Inc No change 🗸 🗸	Misc. values Home pos
	Top of stock 0.0	Planes NC Name
	Abs/Inc No change 🗸 🗸	

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**.

	New viewsheet		
	Group	>	New Group
	Ungroup		Group-1
	Сору	Г	
	Rename		
	Delete		
Y	Save viewsheet as bookmark		
1	Delete viewsheet bookmark		
X	Restore from viewsheet bookmark		
Тор	Settings		
•	Viewsheets Help	-	
Group-1 Viewsheet-1 Viewsheet-2 V	ewsheers viewancer		

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

	Group Ungroup	>
Ť	Copy Delete	
×	Delete viewsheet bookmarks	
Тор	Settings Viewsheets Help	
Group-1 Viewsheet-1 Viewsheet-1	CCC2 VIEWSNECC-5 VIEWSNECC-5 VIE	heet-5 +

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**.

Create Vie	wsheet Group
Name:	Group-2
Color:	10

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.

System Configuration	
Analyze CAD Chaining Colors Converters Default Machines Default Machines Files On-Screen Controls	Planes work offset conflicts Update the plane and all operations that use the plane Create a copy of the plane and update just the changed operation Update just the changed operation without changing the plane Suppress planes work offset warning
Post Dialog Defaults Printing Reports Screen Selection Shading Simulation Solids	Plane association New planes Image: Always associate to selected geometry Existing planes Image: Always associate to selected geometry
Spin Controls Start / Exit Tolerances	 Always associate to selected geometry Keep existing associativity Prompt to associate to selected geometry

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.

Analyze	^		
CAD		Number of optition for dynamic rotation	0
Chaining		Number of endies for dynamic rotation	·
Colors		View transitions	
Communications			
Converters		Use 'free' mode in dynamic rotation	
Default Machines		Spap to views in dynamic rotation	
Dimensions and Not	es 🛛		
Files		Span tolerance (degrees)	25.0
On-Screen Controls			
Planes		Motion controller velocity	0.3
Post Dialog Defaults	s I		
Printing		Animate view transitions	
Reports			
Screen		Animate zoom fits	
Grid		Name I and an Kar	
Viewsheet		Normal animation	
View		Potation factor (degrees)	5.0
Selection		Rotation factor (degrees)	
Shading			

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.

Analyze 🔨		
CAD	Color: 93	<u>C</u> ustomize
🕂 🕀 Chaining		
Colors	Arc Center Points	
Advanced Toolpath Display	Endpoints Default groups	
Tools and holders	Draft dirty	
Simulation Display	Draft phantom	
Geometry Display	Group	
Communications	Mesh backside	
Converters	Mesh edge	
Default Machines	Remainder	
Dimensions and Notes	Result	
🕂 🕂 Files		
🗄 On-Screen Controls	Use Group and Result color in Transform	
Planes	Group colors	
Post Dialog Defaults		
Printing	 Use entity colors 	
Reports	O Use group's color	
Screen		
Grid	Default attributes	
Viewsheet	Circu	
View	Size:	
Selection	Small Large	
Shading		
Simulation ✓		

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.

Merge Pattern	Ψ ×
Basic	
Position Select Align Dynamic Mirror Scale	۲
 Merged file levels Active level 	۲
O Offset by: 0	▼
Viewsheets Image Viewsheets	۲

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.

Options		×
Quick Access Toolbar	Choose commands from: Context Menu:	
Customize Ribbon	Commands Not in the Ribbon + Import	
Customize Ribbon Context Menu Options	Import Import	ienu perties for more options. ate Submenu Rename
	Customizations: Res	set
	ОК	Cancel Help

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.

Mach	/lachine Group Setup 🔹 🤅		
?		📀 😒	
		Machine	
<u>.</u>	Group	۲	
	Name:	Machine Group-1	
Ŷ	Output folde	r: C:\Users\smg\Documents\My Master	
Ŷ	Comment:		
Ϋ́π.			
Ī			
	Machine Defi	inition 📀	
9	Description:	Mill Default	
	File:	Mill Default.mcam-mmd	
	Control:	DEFAULT.mcam-control	
	Post:	C:\Users\Public\Documents\Shared Masterca	
	Configure m	achine	
	Preview Setti	ngs	
	✓ Housing		

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.

	Machine Group Setup			т ×
	?			📀 😒
			Machine	
0	Ī	Machine	Group	
6			Name:	Machine Group-1
9	¥	Master Model	Output folder	C:\Users\smg\Document
3	Ŷ	Stock Setup	Comment:	
4	Ń	Workholding		
6	ī	Tools	Mashina Dafi	-141
6	10	Simulation	Description:	Mill Default
			File:	Mill Default.mcam-mmd

- 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.

Input files: "C:\Users\Public\Documents\Shared Mastercam 2023\Mill\Post
Target directory:
C:\Users\Public\Documents\Shared Mastercam 2023\Mill\Posts'
Use the path of the input file as the target directory Do not append license number to output file Options
 Encrypted file for any license of the specified type Single encrypted file for all license numbers Separate encrypted file for each license number
Always allow post to run on Reseller license

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.

MPBin License Entry									
License numbers									
		Туре	ID						
	1	License ID	777777						
	2	License ID 🗸	0						
		License ID Customer ID							

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.

MD CD PST	[Rename			- 🗆 ×		
Folder:	C:\Users\	Public\Docun	nents\Share	d Masterca	am 2023\CNC_M	ACHINES 🔹
Mill) Lathe	Router	Wire	Log	Ç Configuration	
						Rename to:
Machine:	MILL DEFAULT MM.MCAM-MMD					MILL DEFAULT MM
Control:	DEFAULT					DEFAULT
Post Files:	C:\Users\Public\Documents\Shared Mastercam 2\MPFAN.PS					T MPFAN
						O O O
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.

STEP File Parameters	×
Import Solids Attempt to heal solids Use surface stitching to import solids	
Create hole operations	

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.

🛼 Migration Wizard	
Advanced: File Locations	File Types Select one or more file types to migrate.
File Types Versions Finish	 ✓ File types ✓ Part files ✓ Operation libraries ✓ Control definitions ✓ Default files ✓ Tool libraries ✓ Machine definitions ✓ Machine simulation

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

💥 Script Linker	- 🗆 X
Function name:	
Screen tip:	
16 x 16 pixel icon:	*
32 x 32 pixel icon:	*
Output location:	Shared Add-Ins folder 🔹
	Copy script to output location
	Export Cancel Help

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.

Nesting 2023 V2018R1 - untitled	×
Sheets Parts	
Sheet List: 1 of 1	
Sheet #1	
Parameters Size: 2400.0 ∨ ≜ 1200.0 ∨ ≜	Common Parameters (all sheets)
Create necessary quantity Priority: Quantity: 1 1	Nesting Comer: Lower Left Multiple Comers: Off Fill Direction: Flipped
Origin: X: 0.0 ✓ Y: 0.0 ✓	Automatic sheet Four Sheet-Sheet Distance. 1.0
Material	Sheet Margin: 0.0 V

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.





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.

≞) ı

ΝΟΤΕ

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		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	Fanuc 30i 31i 32i		
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	
Doosan DVF 5000		 Head-head AC 	
Doosan DVF 6500		Head-head	The Siemens post now
Doosan DVF 8000		• Table-table	adds the Doosan machines to its portfolio.
Doosan VC 630/5AX		AC	
Generic Siemens 5X Mill		• Table-table BC	
Haas 5X Mill			
Haas UMC-1000			New generation Haas 5-
Haas UMC-1000SS			axis Mill post, replacing the previous UMC-750.
Haas UMC-1250			This post supports:
Haas UMC-1250SS			• The newer
Haas UMC-1500-DUO			generation of the Haas control
Haas UMC-1500-DUO	Haas NGC	Table-table	• DWO (G254)
Haas UMC-1600-H			• TCPC (G234)
Haas UMC-500			• TWP (G268)
Haas UMC-500SS			 High speed machining modes
Haas UMC-750			• G10 list (tools and
Haas UMC-750SS			work datums)

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		Single Spindle / Single Turret / Tailstock / Steady Rest	
TD45 Y Z3200_12st BOT	Fanuc 31i-A		
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		Dual Spindlo / Single Turret	
ST-15_S_Hybrid_v2	Hads CNC	Dual Spindle / Single Turret	
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			
230BL YSMC_BMT65X24	Fanuc 0i-MF	Dual Spindle / Single Turret	
230CL YSMC_BMT65X24			
DMG Mori Seiki NLX			
NLX3000MC 3000TSY_R_12st BOT		Single Spindle / Single Turret / Tailstock / Steady Rest	
NLX1500MC 500SY_12st BOT	Mitsubishi (CELOS)	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	Famue 21; DE	Dual Spindle / Tool Spindle	
TMX-4000ST	Falluc STI-BS	Dual Spindle / Tool Spindle / Lower Turret	
Mazak Integrex j-series			
j-200 500U		Single Spindle / Tool Spindle / Tailstock	
J-200S	Smooth C	Dual Spindle / Tool Spindle	
j-300	Shiooth G	Single Spindle / Tool Spindle / Tailstock	
j-400		Single Spindle / Tool Spindle / Tailstock	
DMG Mori Seiki CTX			
CTX beta 1250 TC 4A – Gen 2		Dual Spindle / Tool spindle / Lower Turret	
CTX gamma 3000 – Gen 1		Single Spindle / Single Turret / Tailstock	
CTX gamma 1250 TC 4A – Gen 2	Siemens 840D (DMG Structure)	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	Sigmons 840D (DMC		
NTX2500 1500SZM – Gen 2_ BMT60x10	Structure)	Dual Spindle / Tool Spindle / Lower Turret	
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		Single Spindle / Dual Turret / Tailstock	
LU35-II 2SC x1500	03P-P300L	Single Spindle / Dual Turret / Tailstock	
Okuma Multus U			
U4000 2SW x1500_Compact-H1		Dual Spindle / Tool Spindle / Lower Turret	
U4000 2SW x2000_Compact-H1		Dual Spindle / Tool Spindle / Lower Turret	
U4000 1SW x1500_Compact-H1	03P-P300	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/productdownloads for the latest downloads.

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





Mastercam[®] is a registered trademark of CNC Software, LLC. All other trademarks are property of their respective owners. ©1983-2022. All rights reserved.

www.mastercam.com