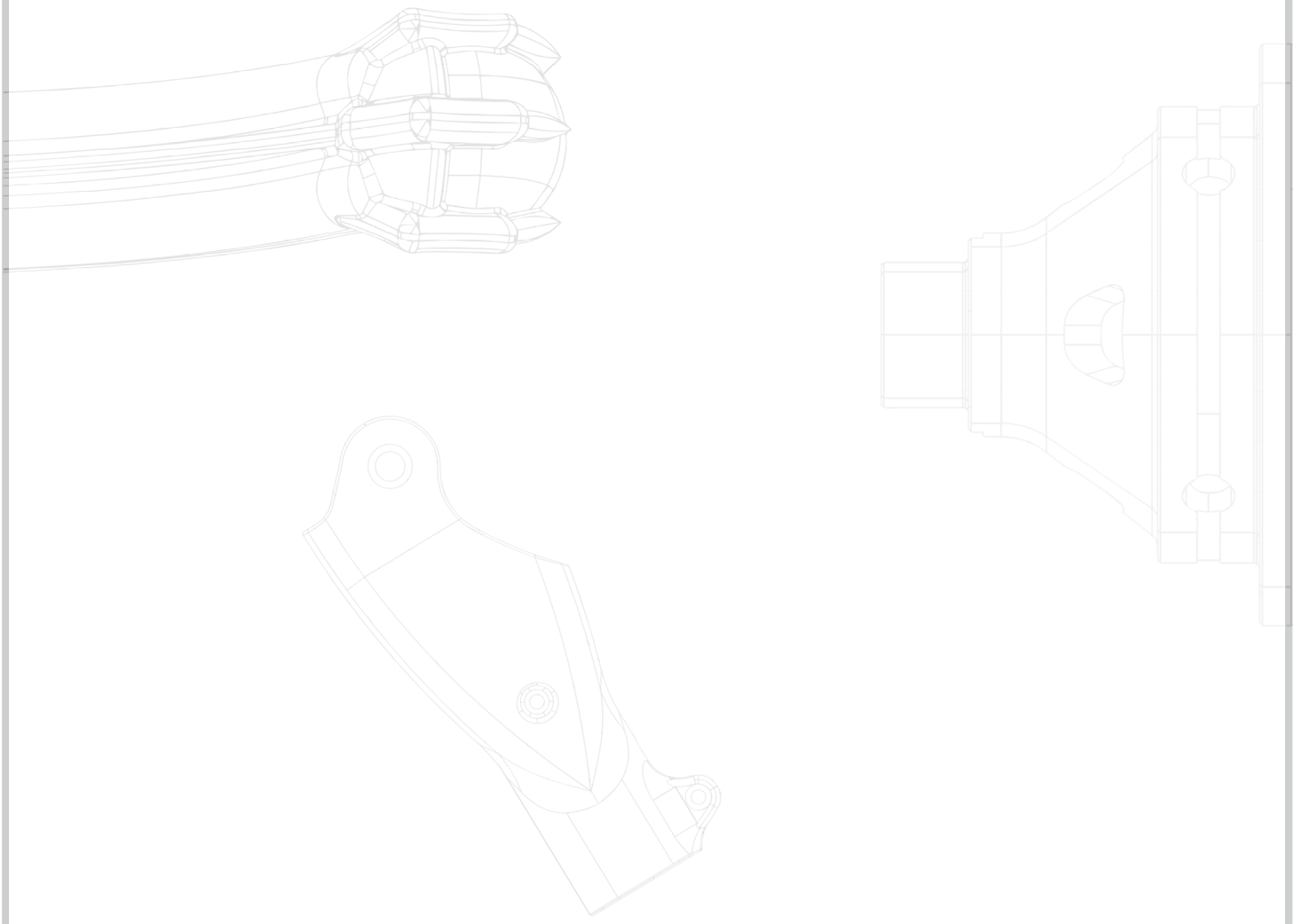
# WHAT'S NEW IN MASTERCAM 2019

April 2018



POWERED BY MASTERCAM'S DYNAMIC MOTION™ TECHNOLOGY

Mastercam

# WHAT'S NEW IN MASTERCAM 2019

April 2018
© 2018 CNC Software, Inc. – All rights reserved.
Software: Mastercam 2019

## Terms of Use

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

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

## Be sure you have the latest information!

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

# TABLE OF CONTENTS


**Introduction** ..... **9**

Release Highlights ..... 9

Mastercam Resources ..... 9

Contact Us ..... 10

**General Enhancements** ..... **11**

Advanced Toolpath Display ..... 11

Analyze Enhancements ..... 13

Analyze Distance ..... 13

Analyze Toolpath ..... 14

Block Drill Support ..... 14

Bounding Box ..... 15

Capture Images ..... 15

Changes to Files, Options ..... 16

Dynamic Gnomon Settings ..... 17

File Support Enhancements ..... 17

AutoCAD Paper Spaces Support ..... 17

Export STEP Options ..... 17

Import ProE/Creo Files with B-Rep Data ..... 18

Model Based Definition Presentation Data Support ..... 19

Parasolid ..... 19

SolidEdge Support ..... 19

Unigraphics NX ..... 20

Files with Multiple Data Streams ..... 20

Guided Chaining for Partial Chaining ..... 21

Levels Enhancements ..... 21

Pasting Entities to a Level ..... 21

Turn All Levels On and Off ..... 22

Mastercam Simulator ..... 23

Axis Control ..... 23
Capture/Replay ..... 23
Color Loop ..... 24
Display or Hide Machine Housing ..... 25
Save Tool as Wireframe or Mesh ..... 26
Stop Conditions ..... 27
Timeline Zoom ..... 28
Toolpath Analysis ..... 29
Material Display ..... 29
Nesting Enhancements ..... 30
MCLink INI Files ..... 30
Order of Operations ..... 31
Save Color Images ..... 32
Planes Enhancements ..... 32
From Plane Displays in Color ..... 32
Gnomon's XY Pane Displays in Color ..... 33
Planes Axes and Grid Settings ..... 34
Quick Settings Saved as Defaults ..... 35
Section View ..... 36
Selection Enhancements ..... 38
Glow Highlighting ..... 38
Mesh Node Selection ..... 40
Toolpaths Manager Display Options ..... 40
Viewsheet Enhancements ..... 41
Wireframe Tool Display ..... 42
**Design Enhancements ..... 43**
Drafting Enhancements ..... 43
Cross Hatch ..... 43
Drafting Associativity ..... 43
Note Height ..... 45

On-screen Triggers ..... 45
Solids and Model Prep Enhancements ..... 46
Boolean ..... 46
Chamfer ..... 46
Disassemble ..... 46
Hole Axis ..... 47
New Align Functions ..... 47
New Solid Hole Function ..... 49
Push-Pull ..... 50
Solids Manager ..... 51
Surface Enhancements ..... 51
New Edit Surface Function ..... 51
Power Surface ..... 52
Surface From Solids ..... 54
Wireframe Enhancements ..... 54
Curve One Edge and Curve All Edges ..... 54
New Divide Function ..... 55
Join Entities ..... 56
Offset Chains ..... 56
On-screen Controls ..... 56
Point Nodes ..... 57
Spline Blended ..... 57
**Mill Enhancements ..... 58**
Check Holder ..... 58
Mastercam's Accelerated Finishing™ ..... 59
New Point Selection Workflow ..... 61
Skip Pocket Smaller Than ..... 61
2D Enhancements ..... 63
Miscellaneous Enhancements ..... 63
Maintain Sharp Corners ..... 63

New Model Chamfer Toolpath ..... 64
3D Enhancements ..... 66
Area Roughing Improvements ..... 66
New Equal Scallop Toolpath ..... 67
Smoothing Tolerance ..... 69
Transitions Feed Rate ..... 70
Multiaxis Enhancements ..... 70
Miscellaneous Enhancements ..... 70
Gradual Front Shift ..... 71
Multi-Threading ..... 72
New Deburr Toolpath ..... 73
**Lathe Enhancements ..... 75**
3D Tool Support ..... 75
Improved Support for Cross-Centerline Turning ..... 76
Mastercam Lathe for Swiss Machines ..... 77
Multiple Plunge ..... 78
PrimeTurning™ Toolpath ..... 80
Improved Tool Libraries for PrimeTurning ..... 80
3D Tool Libraries for PrimeTurning ..... 80
**Mill-Turn Enhancements ..... 82**
Adjust Tool Projection ..... 82
Bar Feed Enhancements ..... 83
Import and Export Operations ..... 83
Improved Tool Loading ..... 84
Improved Work Offset Support ..... 84
New Options for Managing Collision Detection Files ..... 84
Reference Positions ..... 85
Simulation Enhancements ..... 85
**Post Enhancements ..... 87**
Cascading Postblocks ..... 87

NCI Updates ..... 88
New MP Function for Regular Expressions ..... 89
New Stock Function ..... 90
UTF-8 Encoding for MP Posts ..... 91
UTF-8 Encoding for NC Output ..... 91
Main NC Output Stream ..... 91
Auxiliary Output Stream ..... 91
Buffers ..... 92
NCI Files ..... 92

# INTRODUCTION

Welcome to Mastercam 2019! Mastercam 2019 features new functionality focused on delivering speed and efficiency to your machining jobs.

## Release Highlights

Listed below are a few of the highlights of this release, including new toolpaths and enhanced functionality.


- "New Deburr Toolpath" on page 73
- "New Model Chamfer Toolpath" on page 64
- "PrimeTurning™ Toolpath" on page 80
- "New Solid Hole Function" on page 49
- "3D Tool Support" on page 75
- "Advanced Toolpath Display" on page 11


**WARNING:** Screen colors in the document pictures were modified to enhance image quality; they may not match your Mastercam settings or results. These color differences do not affect functionality.

## Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Documentation*—Mastercam installs a number of helpful documents for your version of software in the Documentation folder of your Mastercam 2019 installation.
- *Mastercam Help*—Access Mastercam Help by selecting **Help**, **Contents** from Mastercam's **File** tab or by pressing [**Alt+H**] on your keyboard.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—Our Technical Support department (+1 860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Mastercam Tutorials*—We offer a series of tutorials to help registered users become familiar with basic Mastercam features and functions. Visit our website, or select **Help**, **Tutorials** from Mastercam's **File** tab to see the latest publications.
- *Mastercam University*—Mastercam University, an affordable online learning platform, gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master skills at your own pace and help prepare for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit www.mastercamu.com, or email training@mastercam.com.
- *Online Communities*—You can find a wealth of information at www.mastercam.com. For tech tips and the latest Mastercam news, follow us on Facebook (www.facebook.com/mastercam), Twitter (www.twitter.com/mastercam), or Google+ (plus.google.com/+mastercam). Visit our YouTube channel to see Mastercam in action (www.youtube.com/user/MastercamCadCam)! Registered users can search for inform-

ation or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the knowledgebase at kb.mastercam.com.

## Contact Us

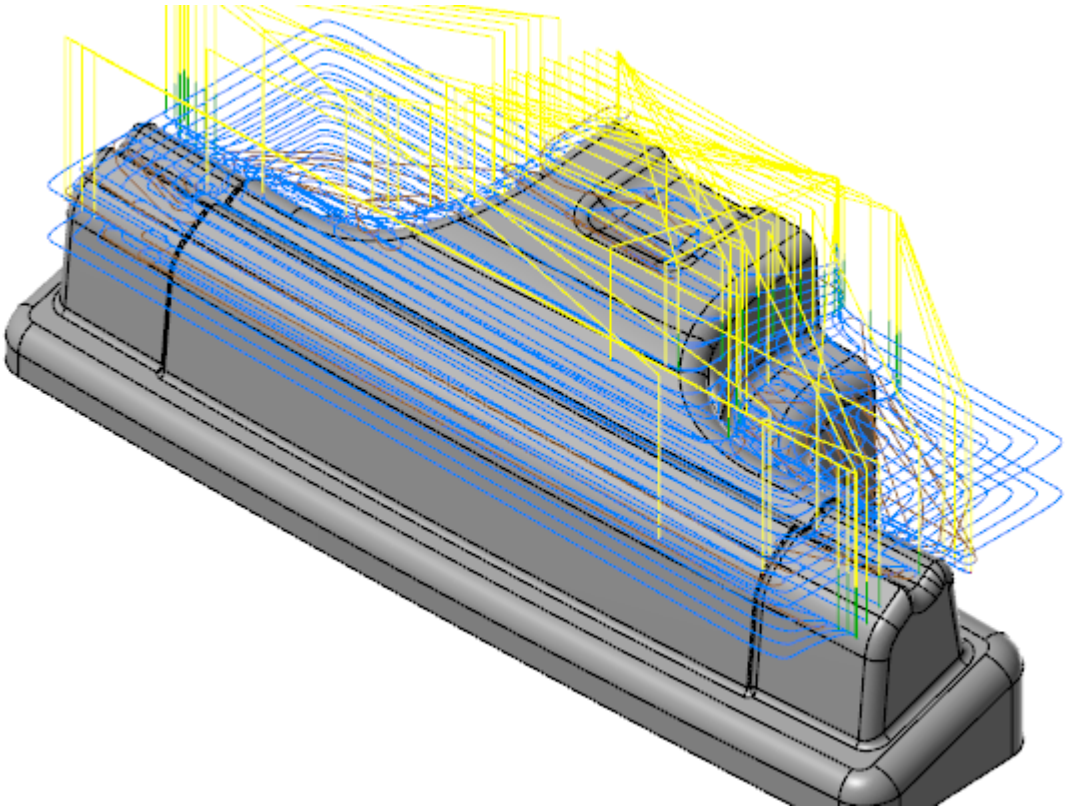
For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at techdocs@mastercam.com.

# GENERAL ENHANCEMENTS

Listed below are general enhancements made to Mastercam 2019. This includes improvements to selection, general toolpath enhancements, and other functions not specific to one product line.

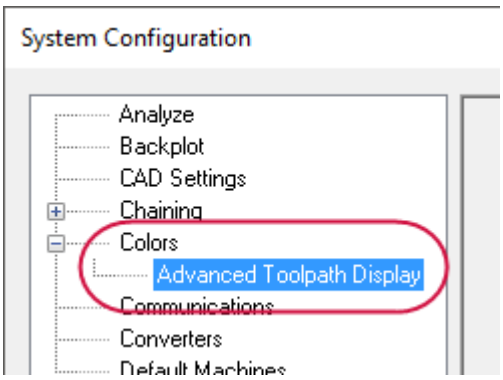
## Advanced Toolpath Display

Mastercam now displays toolpaths with different colors, based on the move type. The following image depicts a part with a Dynamic OptiRough toolpath, with **Advanced Display** toggled on. This is unavailable for Wire toolpaths.

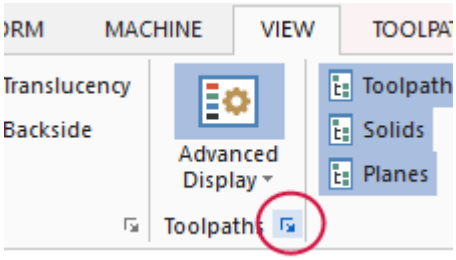


If **Advanced Display** is not toggled on, Mastercam displays only cutting motion and rapid motion in different colors.

Change the display state by selecting **Advanced Display** on the **View** tab. Set these colors in the **System Configuration** dialog box on the **Advanced Toolpath Display** page, which is located under **Colors**.

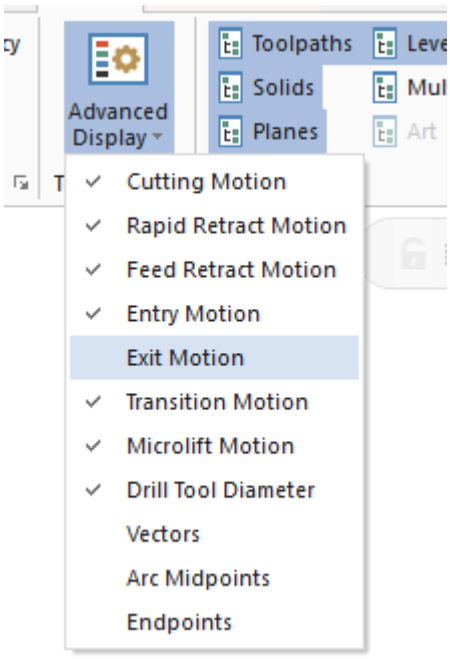


You can also set colors by selecting the dialog box-launcher for **Advanced Display**.

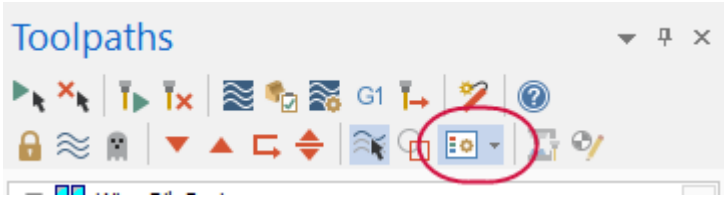



You can also change the line style, line width, and other attributes. For drill toolpaths, you can set the color, line style, and line width of the drill tool diameter.

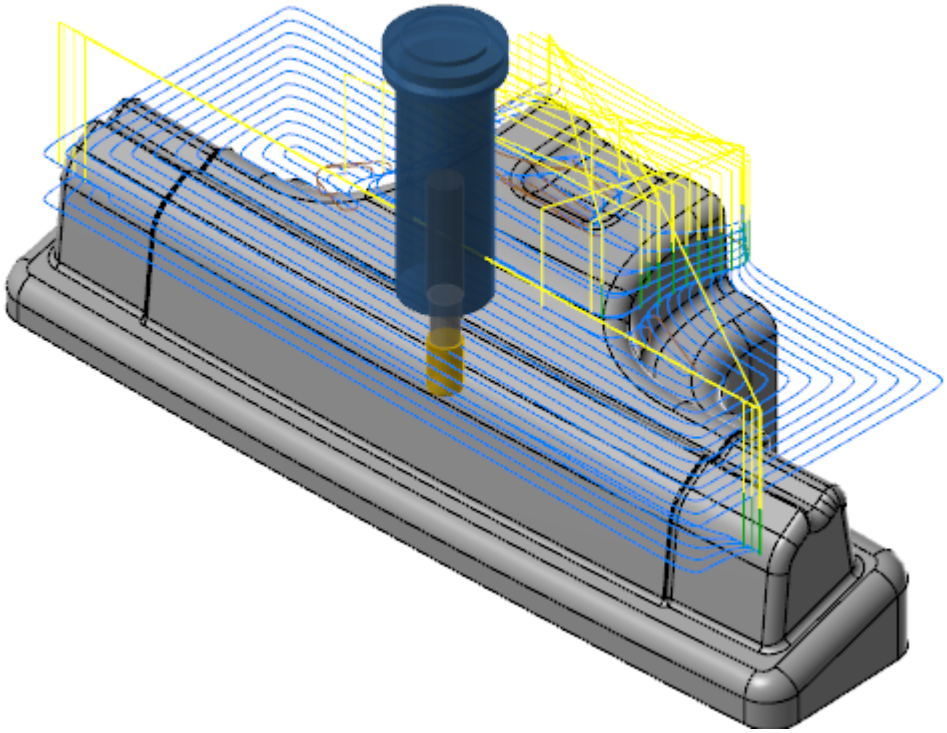
Use the drop-down menu to display different sections of the toolpath. For example, if you deselect **Exit Motion**, then any exit motion does not display in the graphics window.




You can also toggle the display from the Toolpaths Manager.



Classic Backplot and Mastercam Simulator display the customized toolpath colors if the **Advanced Display** toggle is on. If it is not, the default colors display.



## Analyze Enhancements

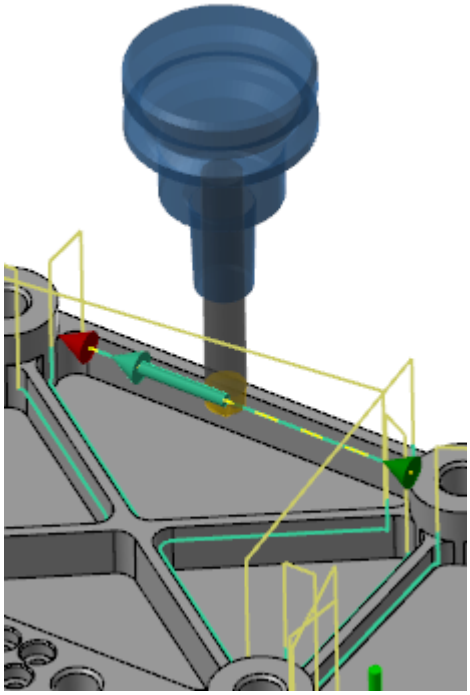
Listed below are enhancements made to **Analyze** functions, located on the **Home** tab.

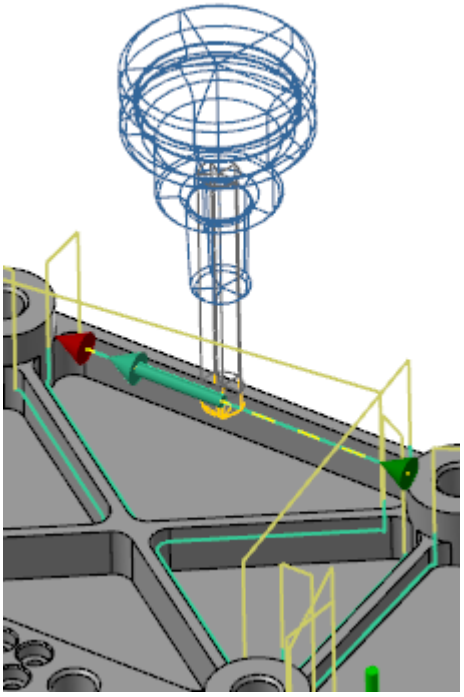
### Analyze Distance

After you select the first entity, the **Analyze Distance** dialog box dynamically updates the distance as you move your cursor. This small change allows you to sample different positions on the screen with approximate distances without requiring you to click a second point or restart the function.

## Analyze Toolpath

When using **Analyze Toolpath**, you can now choose to display the tool and tool holder as shaded or wireframe entities. You can also control the opacity of the tool and holder when they are displayed in shaded mode.

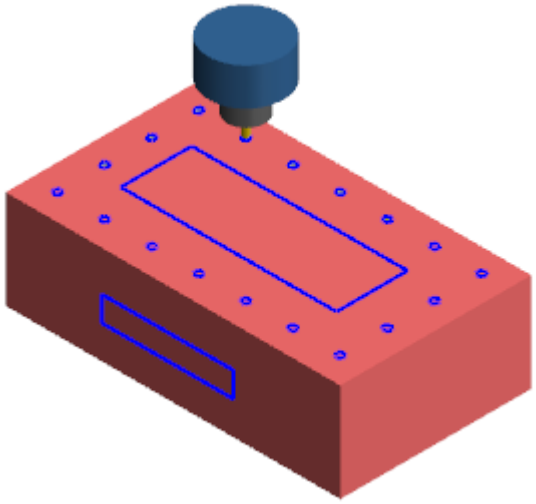
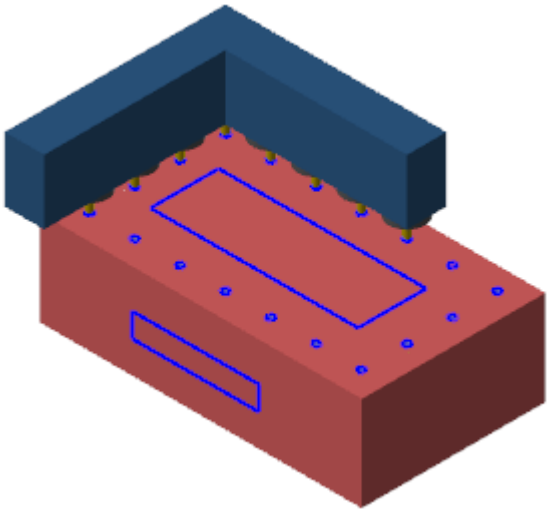




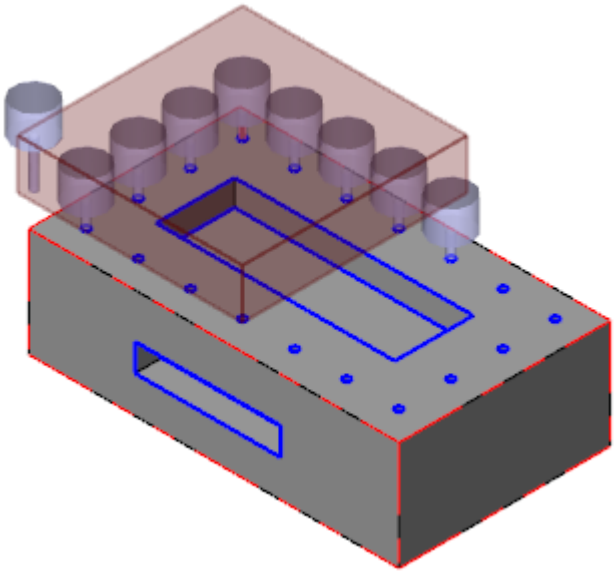
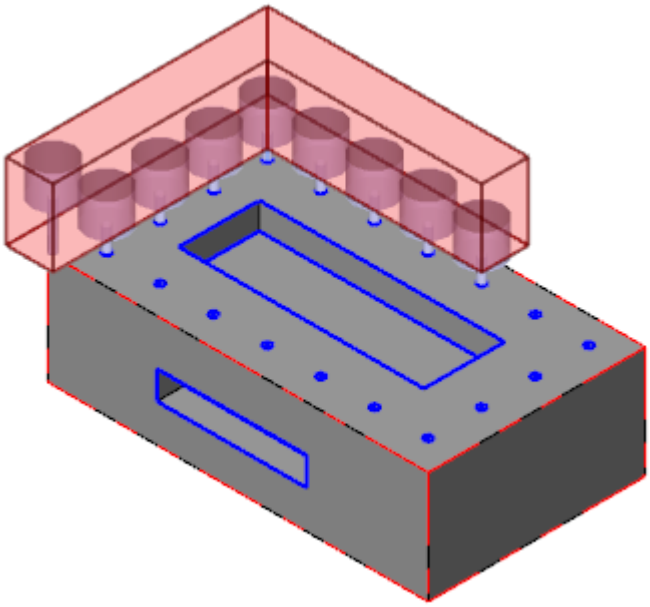
## Block Drill Support

Mastercam Simulator and Classic Backplot now properly support Block Drill toolpaths. This includes support for drilling multiple holes simultaneously.

**Mastercam Simulator, using Verify:**

| Mastercam 2018 | Mastercam 2019 |
| --- | --- |
|  |  |

**Classic Backplot:**

| Mastercam 2018 | Mastercam 2019 |
| --- | --- |
|  |  |

## Bounding Box

In previous versions of Mastercam, you could only create a bounding box in the current construction plane. In Mastercam 2019, four new options allow you to define a different bounding box orientation without having to change the construction plane. They are listed below:

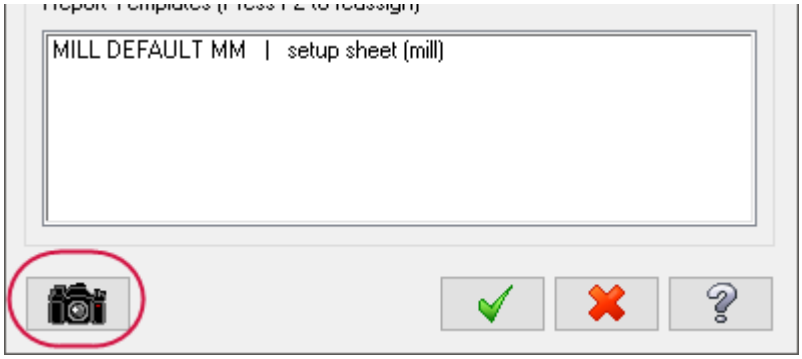
- **Construction plane**: Aligns the bounding box relative to the construction plane. (Default)
- **Auto**: Aligns the bounding box relative to the largest, flattest face of the selected solid body.
- **Face**: Aligns the bounding box relative to a selected solid face.
- **Manual**: Use a gnomon to orient the bounding box.

These options are available on the **Advanced** tab.

## Capture Images

You can now create additional graphics for your ActiveReports setup sheet from the graphics window view. Including these images lets you highlight areas that are complex or require special attention. Mastercam does not limit the number of custom images you can add to a setup sheet.

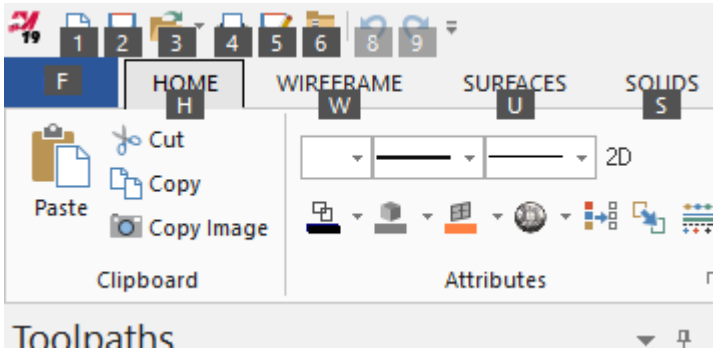
Select **Add Images** on the **Setup Sheet** dialog box to add custom images to your setup sheet. You can preview all of your custom images, edit captions, and add additional images from any location accessible by your system in the ActiveReports setup sheet.



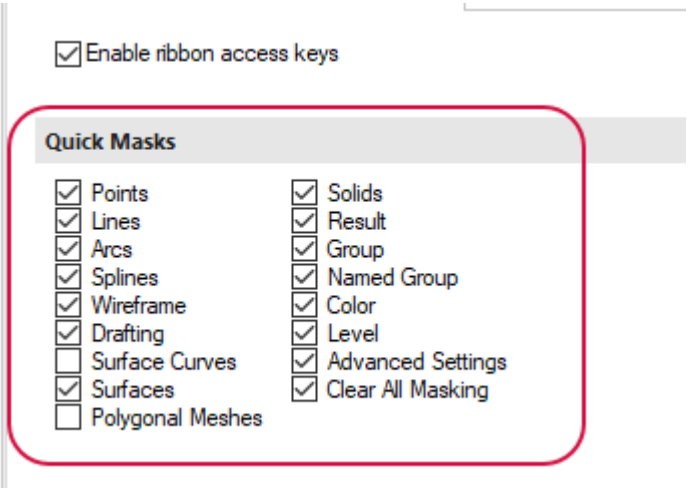

## Changes to Files, Options

The **Options** page in the **Options** dialog box, accessible from **File**, **Options**, has been updated to include several new features.

Deselecting **Enable ribbon access keys** hides the access keys that appear above a tab when you press the [**Alt**] key. This option is on by default.

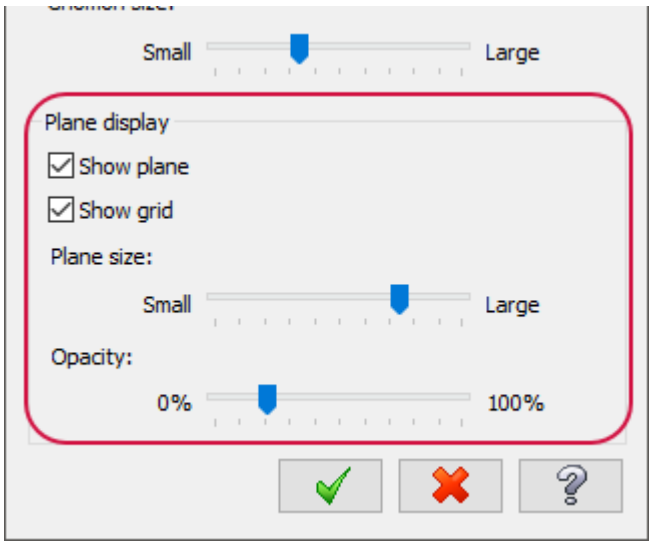



You can select which Quick Masks you want to appear in the graphics window. Deselect a Quick Mask to hide it.



## Dynamic Gnomon Settings

The **Gnomon Settings** dialog box controls the behavior of the gnomon in **Dynamic Planes** or **Transform Dynamic**. New options have been added to enable a translucent plane grid, as well control the size and opacity.




Access this dialog box by right-clicking the dynamic gnomon in the graphics window.

## File Support Enhancements

Listed below are enhancements made to Mastercam to support other CAD tools.

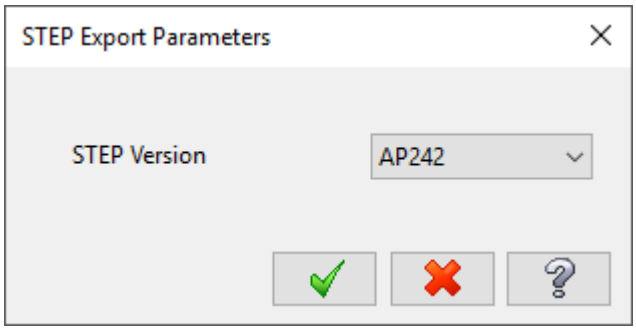
### AutoCAD Paper Spaces Support

In previous Mastercam versions, when you imported AutoCAD files (DWG, DXF) with multiple drawing layers (Paper Spaces) into Mastercam, the model space and only one Paper Space were imported. To see that Paper Space, you needed to unblank your part.

Mastercam 2019 now supports the import of all of your Paper Space layouts and automatically converts them into viewsheets. As a result, **Blank Paper Space Entities** has been removed from the **System Configuration** dialog box and the **DWG/DXF Read Parameters** dialog box.

### Export STEP Options

When saving a Mastercam file as a STEP file, you can specify a STEP application protocol. Select **Options** in the **Save As** dialog box to set the application protocol.

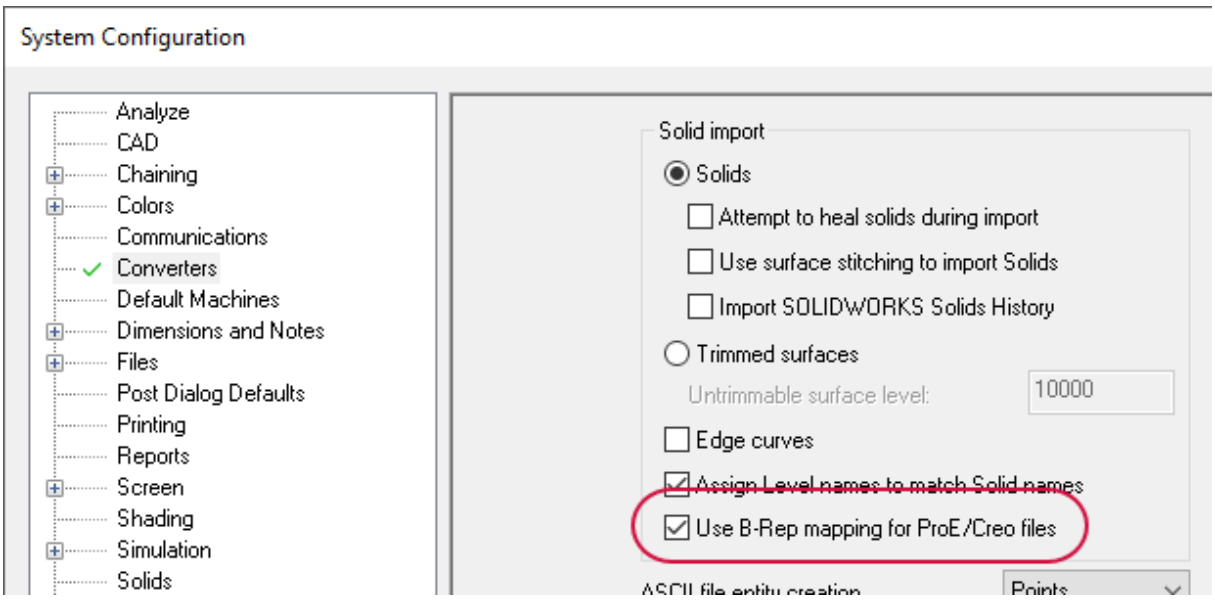


Files that are saved with the 242 application protocol (AP242) retain 3D annotation information. If you choose another application protocol, you lose this information.

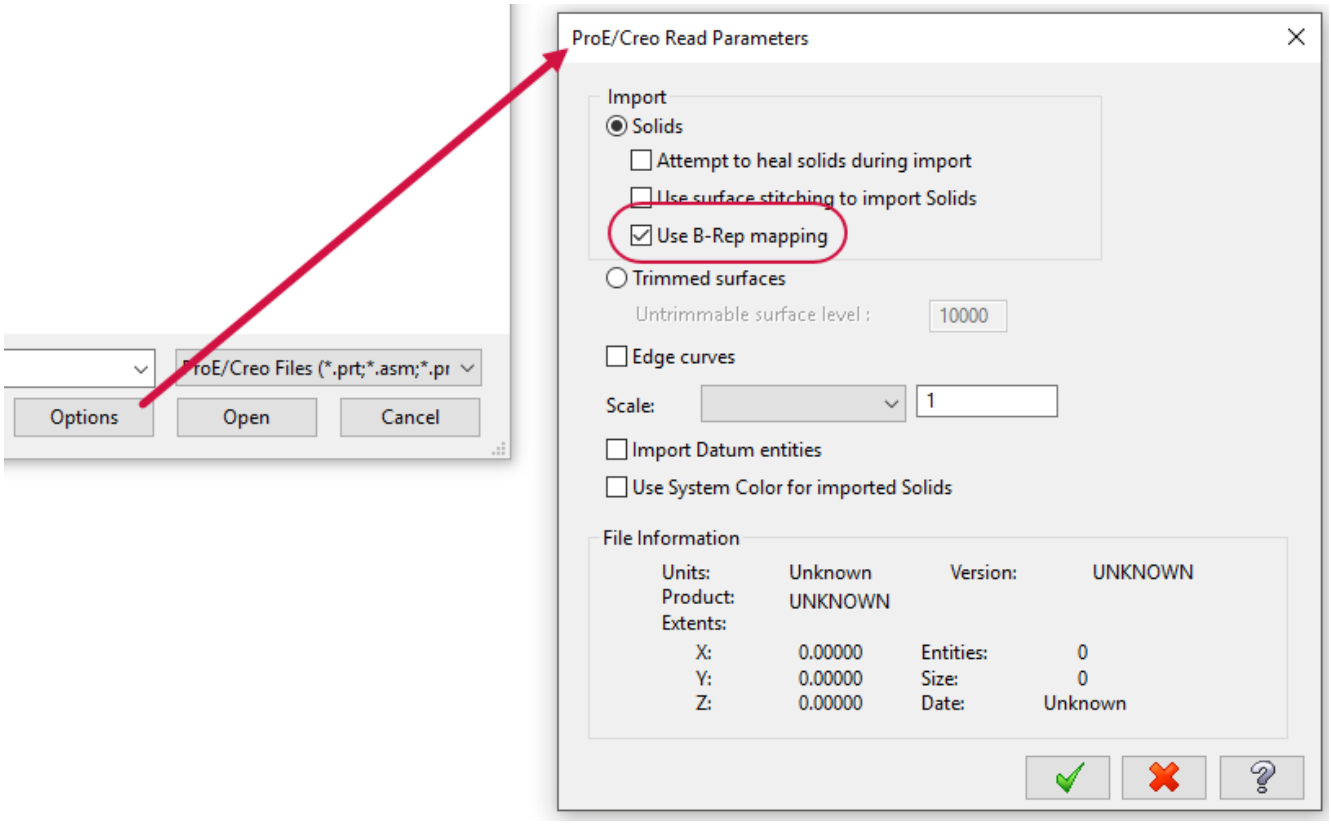
## Import ProE/Creo Files with B-Rep Data

When importing ProE/Creo files into Mastercam 2018, Mastercam would use the Parasolid data. Due to customer comments, Mastercam now provides an option to use the import method prior to Mastercam 2018, which uses the solid's Boundary Representation (B-Rep) data.

You can choose which of these methods to use when converting ProE/Creo files. Select the **Use B-Rep mapping for ProE/Creo files** option on the **Converters** page of the **System Configuration** dialog box to change the default behavior.



This option is also available in the **ProE/Creo Read Parameters** dialog box. Select **Options** when you open a ProE/Creo file. This allows you to override the default when converting individual files.




## Model Based Definition Presentation Data Support

Mastercam preserves Model Based Definition (MBD) data when you import solid models that have used the application protocol AP242. This helps to preserve design intent when you move your work from a CAD package (Creo, NX, CATIA V5, or AutoDesk Inventor[1] ) into Mastercam.

Upon import, Mastercam converts any user-defined views in the imported file into a viewsheet. All presentation data (such as annotations and dimensions) that belong to that view are preserved and displayed in that view.

When using **Analyze**, Mastercam classifies imported presentation data as 3D Annotation entities. They can be deleted, transformed, or included within a bounding box. Although they might resemble Mastercam's own dimensions and notes, you cannot edit or create 3D Annotation entities in Mastercam.

## Parasolid

Mastercam now supports Parasolid 30.0. Wireframe and surface entities are also now included when exporting files to Parasolid.

## SolidEdge Support

Mastercam now imports surfaces, solids, and multiple body types contained in SolidEdge parts.

[1] AutoDesk Inventor must be installed.

## Unigraphics NX

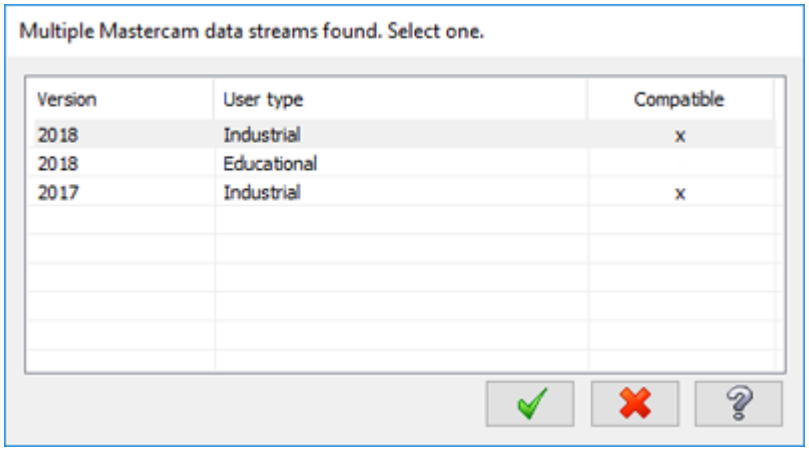
Mastercam has been updated to support the import of Unigraphics NX 12 files.

# Files with Multiple Data Streams

Mastercam now handles SOLIDWORKS files that contain data from multiple versions of Mastercam. When you open a file that contains data from multiple versions, you can choose a stream that is compatible with your version of Mastercam. Mastercam then saves that stream and purges other data streams in that file.

The **Multiple Data streams** dialog box also displays if the data was created by Industrial or Educational users, in addition to the different versions of Mastercam. Mastercam only saves the data to the version and user type that you are running. Data streams that differ only by user type are not purged from the file.

For example, if you are an Industrial user:



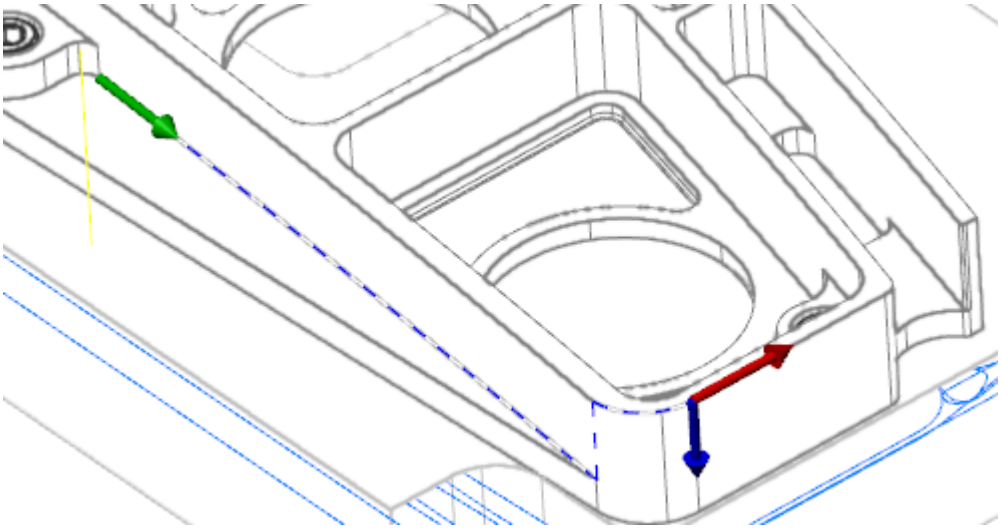

- You only have access to the Industrial data streams.
- If you select the 2018 stream, it is saved to the current version of Mastercam.
- The 2017 stream is purged.
- The Educational stream remains in your part, unchanged.

When you save the SOLIDWORKS file, the Mastercam data is saved to the current Mastercam version based on the current user type.

## Guided Chaining for Partial Chaining

Guided chaining, which was introduced in Mastercam 2018, is now available for most toolpath types when using **Partial chaining** on the **Chaining** dialog box.

When a branch point is reached, a red arrow displays pointing in the direction of the default next direction/entity. One or more blue arrows display, pointing in the direction of alternate branches, as shown in the following example.



Use the following controls with guided chaining:

Use **Next** to continue in the direction of the red arrow.

Use **Adjust** to select a different direction/entity.

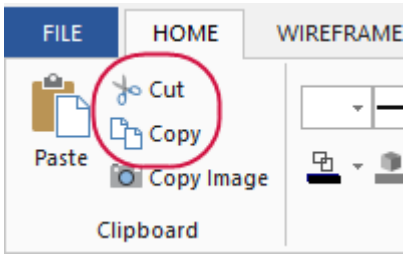
Alternately, click the red or blue arrow to advance the chain in that direction. Lathe toolpaths do not support partial chaining.

## Levels Enhancements

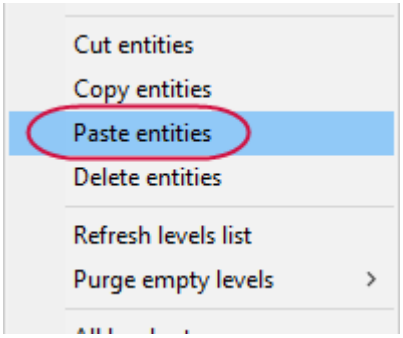
Listed below are enhancements made to levels and to the **Levels Manager**.

### Pasting Entities to a Level

You can now use the **Clipboard** commands, located on the **Home** tab, to cut or copy selected entities in the graphics window to a level.

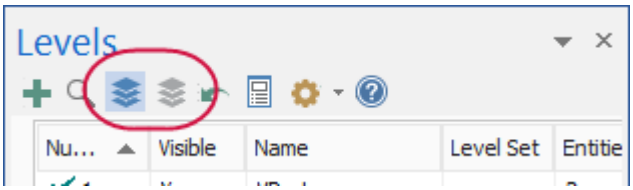


Use **Paste entities** in the **Levels Manager** right-click menu to paste the cut or copied entities to the selected level.




## Turn All Levels On and Off

The **Turn all levels on** and **Turn all levels off** buttons in the **Levels Manager** now toggle the functions as follows:



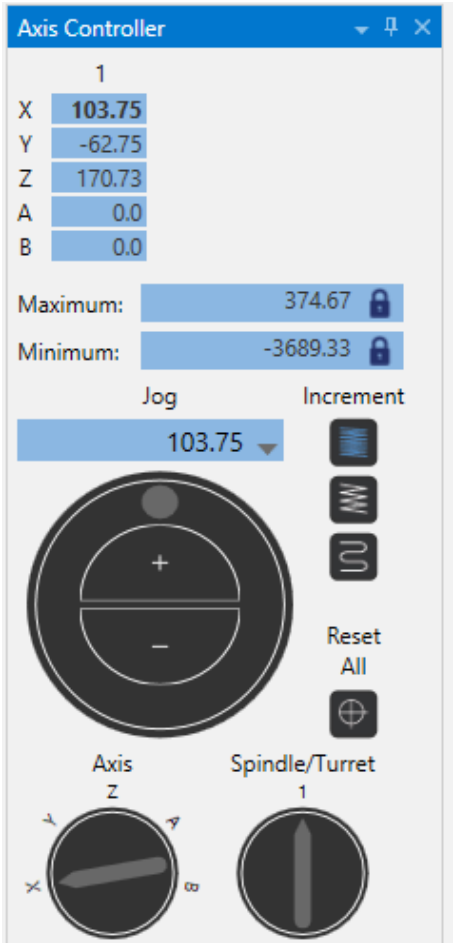

- Select **Turn all levels on** the first time to make all levels visible. The button remains toggled on and **Turn all levels off** toggles off.
- Select **Turn all levels on** a second time to turn all levels to their visible state prior to pressing the button. **Turn all levels on** toggles off.
- Select **Turn all levels off** the first time to turn off the visible state of all levels. The button remains toggled on and **Turn all levels on** toggles off.
- Select **Turn all levels off** a second time to return all levels to their visible state prior to pressing the button. **Turn all levels off** toggles off.

# Mastercam Simulator

Listed below are enhancements made to Mastercam Simulator, available from the Toolpaths Manager and **Machine** tab.

## Axis Control

**Axis Control**, which is available when using **Simulation** mode in Mastercam Simulator, has been enhanced. This panel can be toggled on by selecting **Axis Control** on the **View** tab.




The **Axis Controller** pane manually moves the axes. Use the sliders and buttons to jog the axes. Use this to check if the machine limits are set correctly or for collision checking. If necessary, use **Reset All** to return all options to their default setting and components to their default position.

## Capture/Replay

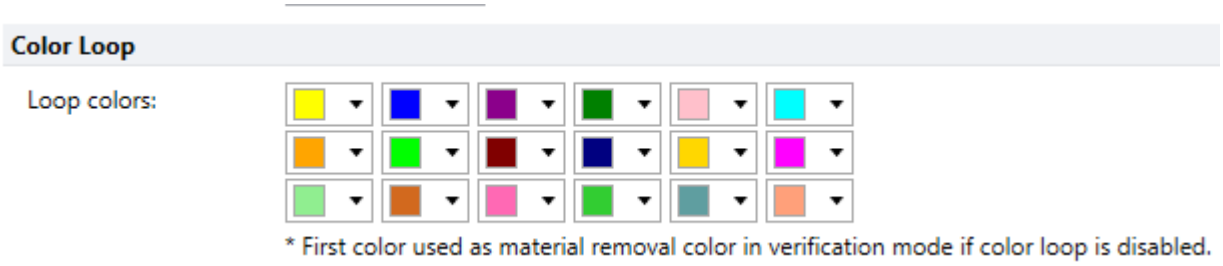
You can now stop your verification partway through processing, and rewind the progress to show material being added back onto the part. Capture/Replay is only available for **Verify** mode.

To use Capture/Replay, you must first turn on the option and then run through the simulation.

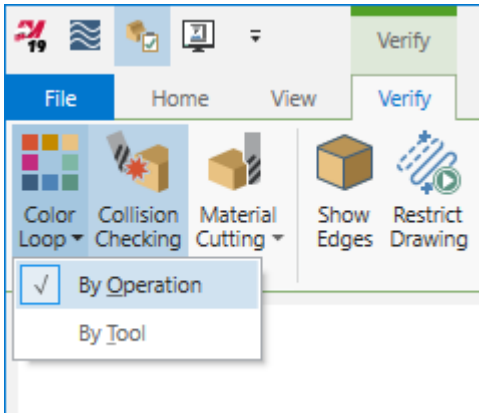
Once simulation is complete, you can use the **Playback bar** to move forward and backward to watch as material is removed or added.

## Color Loop

More colors have been added to **Color Loop** for **Mastercam Simulator**. In Mastercam 2018, there were only 12 available colors. In Mastercam 2019 there are now 18 colors.

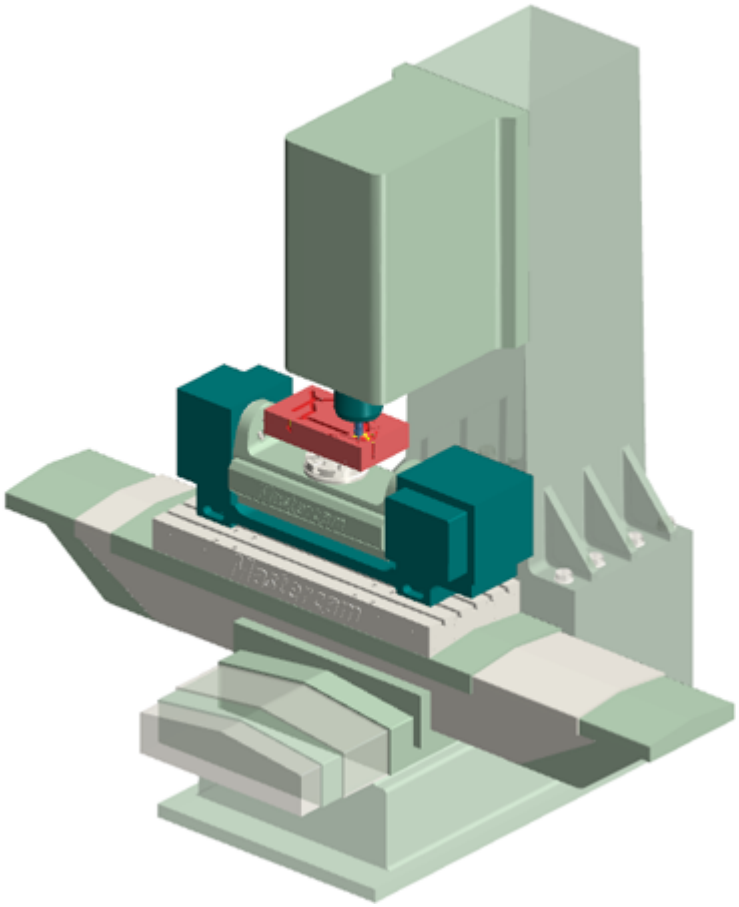



You can also now choose to loop **By Operation** or **By Tool** number from the **Color Loop** drop-down on the **Verify** tab. These options have been removed from the **Options** dialog box.

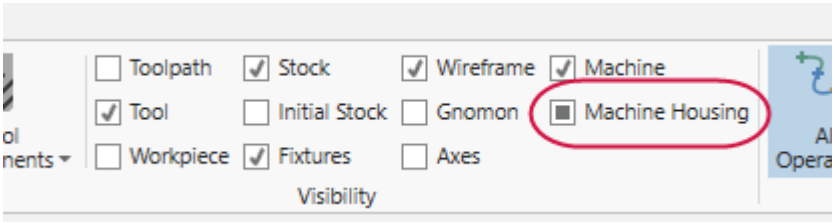


## Display or Hide Machine Housing

When in **Simulation** mode, you can now toggle the state of the **Machine Housing** to **On**, **Off**, or **Translucent**.

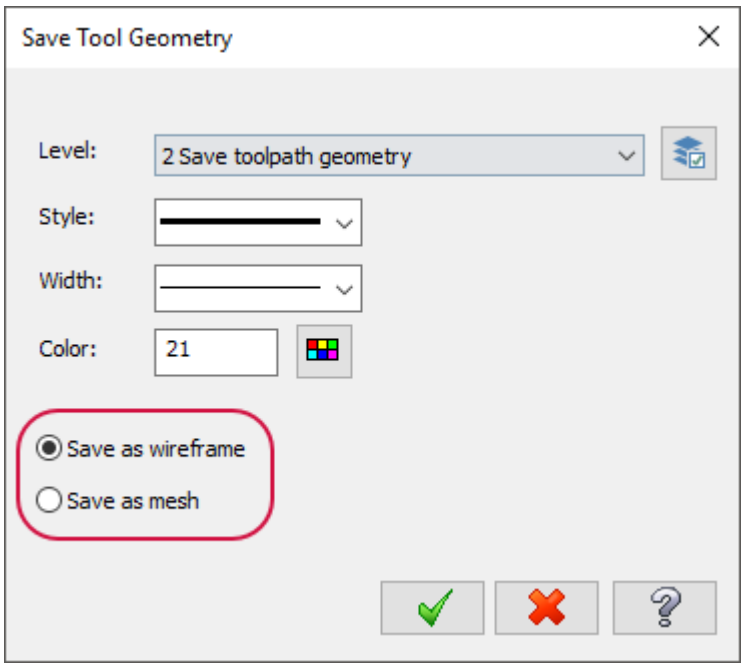


Set this option on the **Home** tab, in the **Visibility** group.

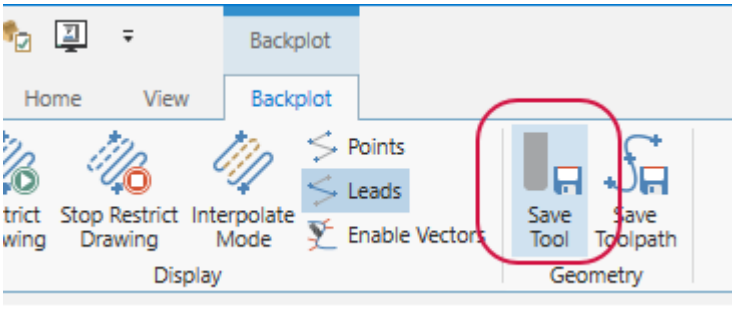


## Save Tool as Wireframe or Mesh

Mastercam Simulator, when in **Backplot** mode, allows you to save your tool as wireframe or as a mesh.

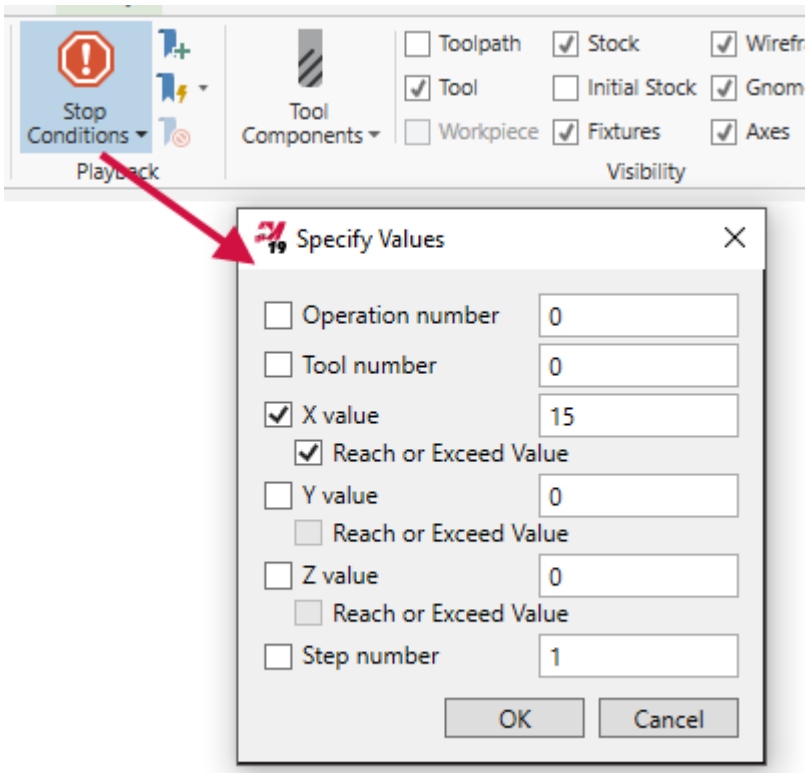



Select **Save Tool** on the **Backplot** contextual tab to set your options.



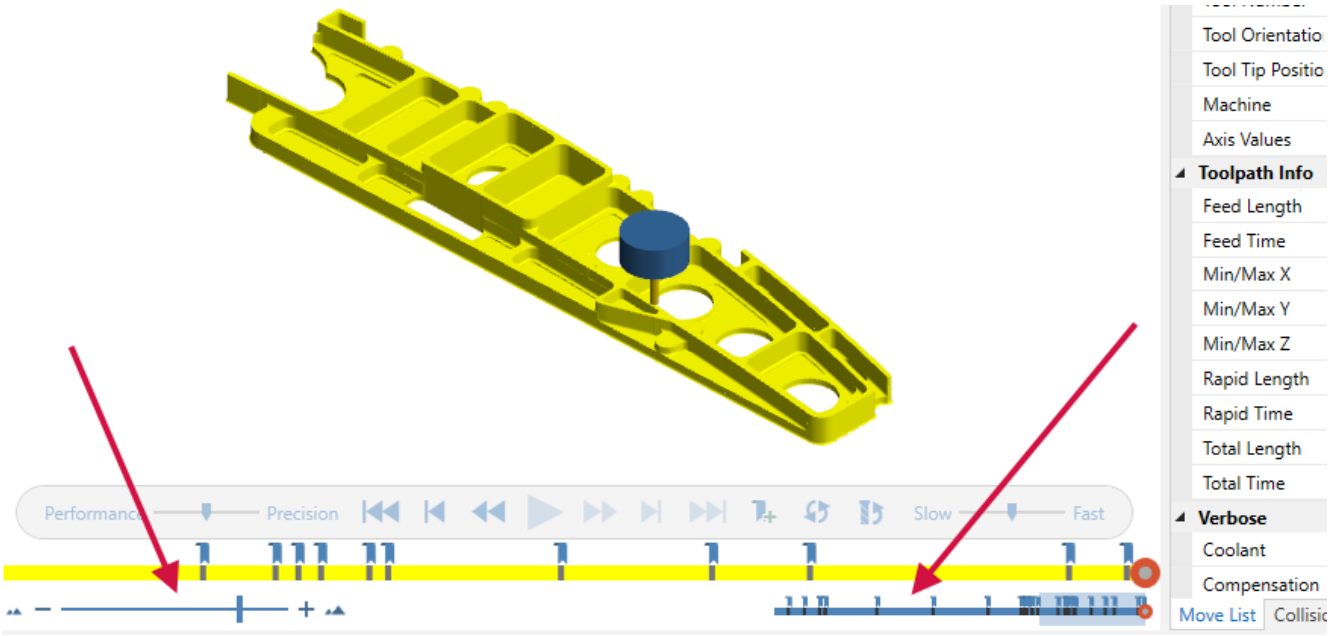
## Stop Conditions

When using **Specify Values** to set stop conditions, you can now select **Reach or Exceed Value** to stop Mastercam Simulator once you have reached or exceeded the entered **X**, **Y**, or **Z** value. You can set stop conditions on the **Home** tab.

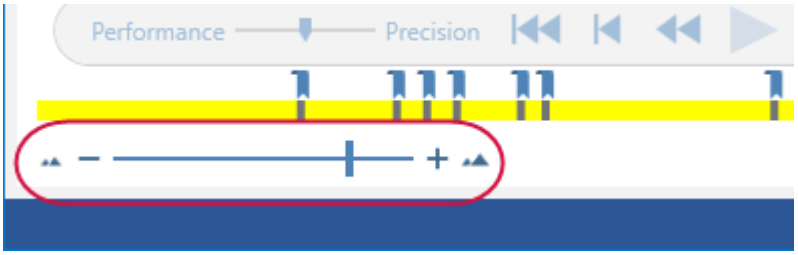


## Timeline Zoom

Timeline Zoom has been added to Mastercam Simulator, allowing you to zoom into the simulation timeline to see more or less detail. This is helpful when using **Bookmarks** that might be close together. Timeline Zoom is toggled on or off on the **View** tab.




Use the **Timeline Zoom** slider in the bottom left corner of Mastercam Simulator to zoom in and out of the timeline.

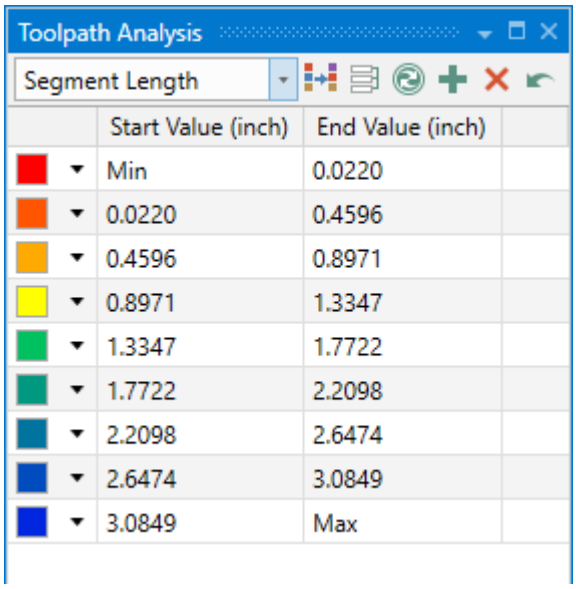



The **Timeline** in the bottom right corner of Mastercam Simulator displays the total timeline of the simulation. The highlighted section displays the zoomed section. You can also move the highlighted section by sliding it forward and backward.

If you toggle **Timeline Zoom** off, the simulation returns to its default setting. If you toggle **Timeline Zoom** back on, it remembers your zoom setting and applies it based on the location of the simulation. This only applies per each session of Mastercam Simulator.

## Toolpath Analysis

Toolpath Analysis is now available when using **Simulation** mode in Mastercam Simulator. Select **Toolpath Analysis** on the **View** tab to display the **Toolpath Analysis** panel.



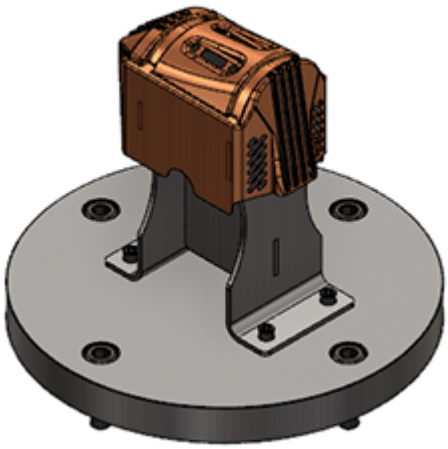

Toolpath Analysis analyzes changes to toolpaths by applying color schemes. Choose from the following options:

- **Operation**: Colors each toolpath differently.
- **Tool**: Colors toolpaths when different tools are being used.
- **Feed Rate**: Colors toolpaths by feed rate and rapid moves.
- **Segment Length**: Colors toolpaths based on the length of the segments.

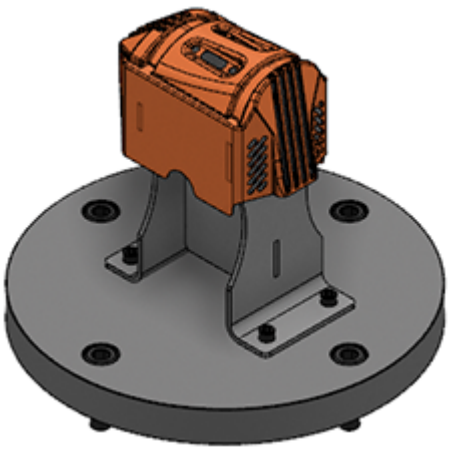
## Material Display

**Material** on the **View** tab controls the display of an assigned material on shaded entities. Toggling **Material** on shows materials that have been applied to entities. Toggling **Material** off returns the entities to their default appearance. **Material** display is not available in **Wireframe** mode.

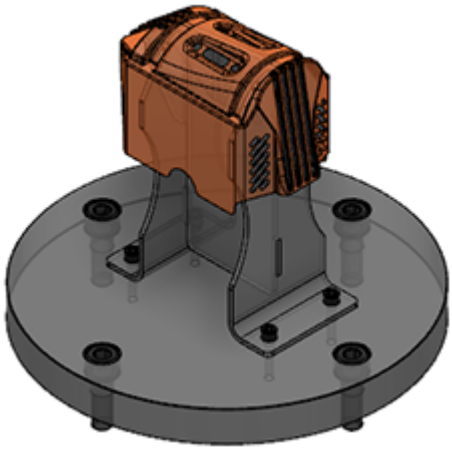
**Metal**



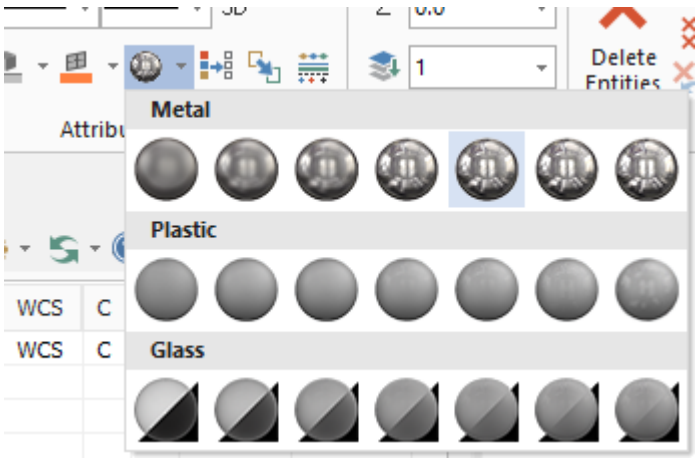
**Plastic**



**Glass**

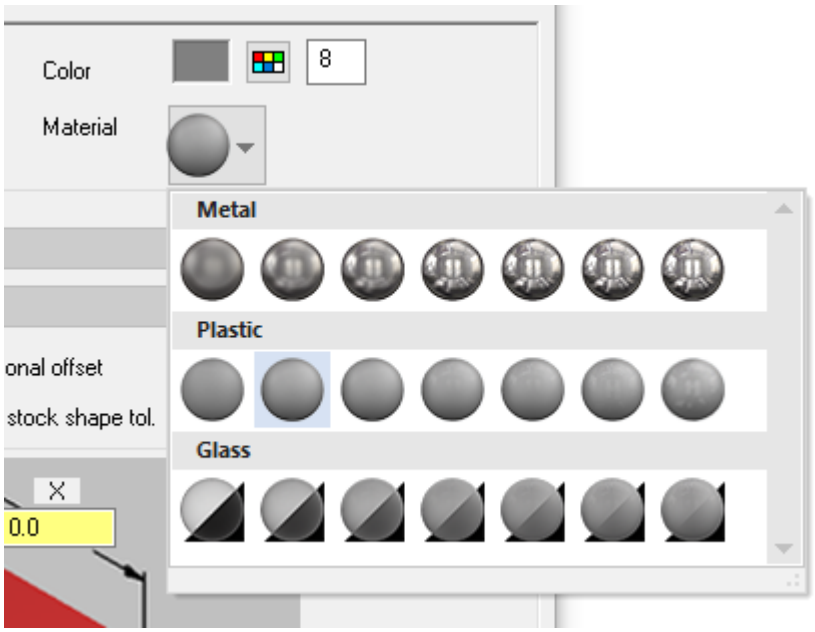


You can apply a metal, plastic, or glass material to an entity. Each material has seven levels ranging from most translucent to most opaque. Applying a material to an entity also toggles **Material** view for the file.




This is set on the **Home** tab in the **Attributes** group, or from the right-click menu in the graphics window.

You can also set the material when creating a stock model, on the **Stock Definition** page.




## Nesting Enhancements

Listed below are enhancements made to toolpath and geometry nesting.

### MCLink INI Files

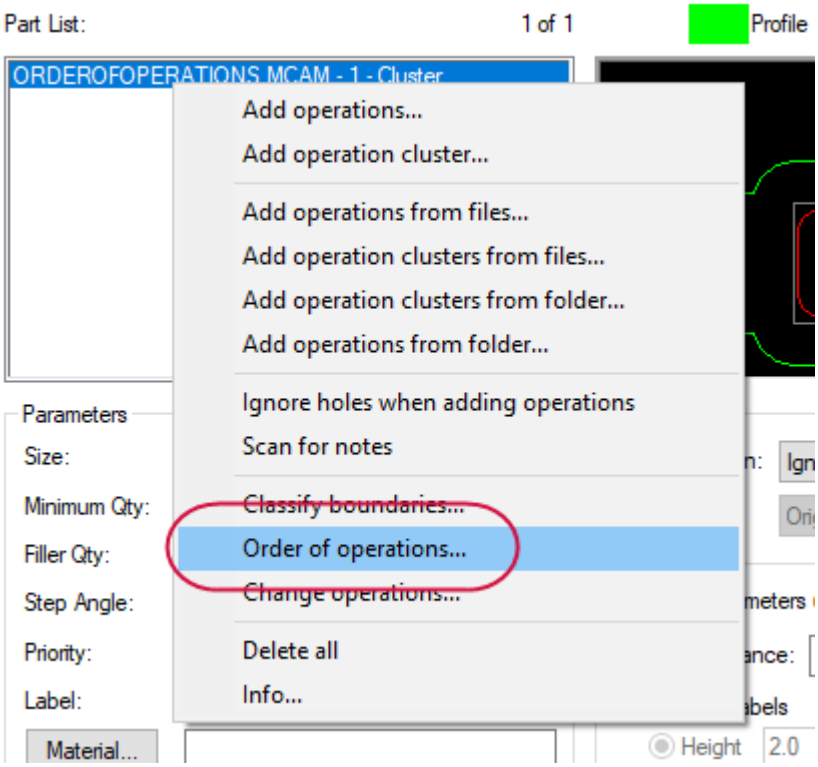
You can open and save MCLink's ATP-generated `.INI` files from the **Toolpath Nesting** dialog box. MCLink is a Mastercam Router utility. MCLink nesting is a method of running a set of nested parts and toolpaths that were output by the Automatic Toolpathing (ATP) add-on.

For a detailed procedure on how to open and save `.INI` files, please read *About Toolpath Nesting* in the Mastercam Help.

## Order of Operations

You can now reorder operations within an operation cluster when using Toolpath Nesting. This option is only available if you have selected **None** as your sorting method.

Right-click on the operation cluster and select **Order of operations**.

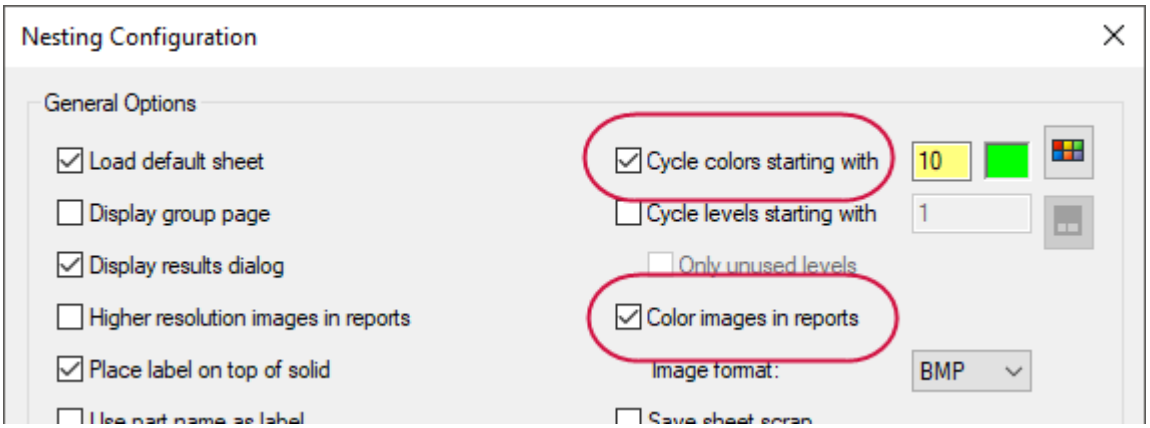



The **Order of operations** dialog box then displays, and lists all operations in the selected cluster. You can then use the dialog box to change the order of the operations.

After Mastercam has sorted the operations, it moves any operations assigned a non-zero machining index. The lower the machining index, the earlier the operation is machined. The higher the machining index, the later the operation is machined. For example, operations that are assigned a negative machining index (such as **-5** and **-1**) are machined before operations not assigned to an index (such as **0**). Operations assigned to a positive machining index (such as **10** and **25**) are machined after operations not assigned to an index. In that case, the machining order for six operations would be the following: **-5**, **-1**, **0**, **10**, **25**.

## Save Color Images

The **Nesting Configuration** dialog box allows you to include color images in your nesting reports by selecting **Color images in reports**. Toolpaths only display in color if you enable **Cycle colors starting with**.

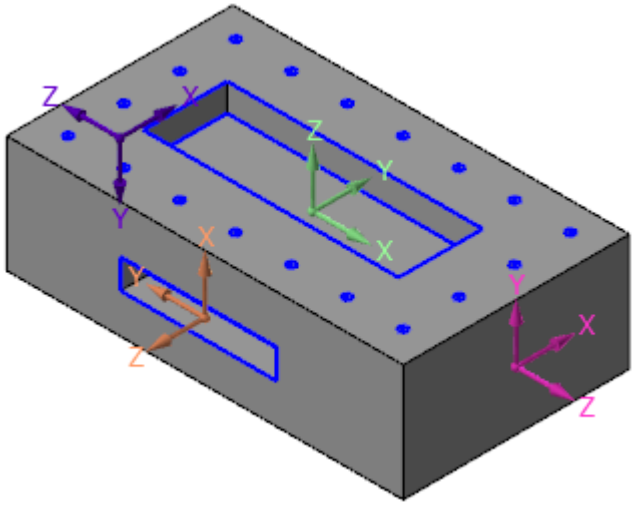


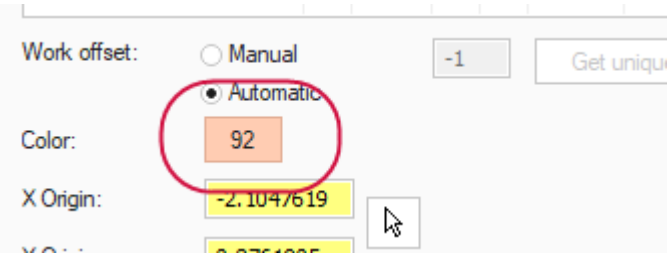

# Planes Enhancements

Listed below are enhancements made to planes and to the Planes Manager.

## From Plane Displays in Color

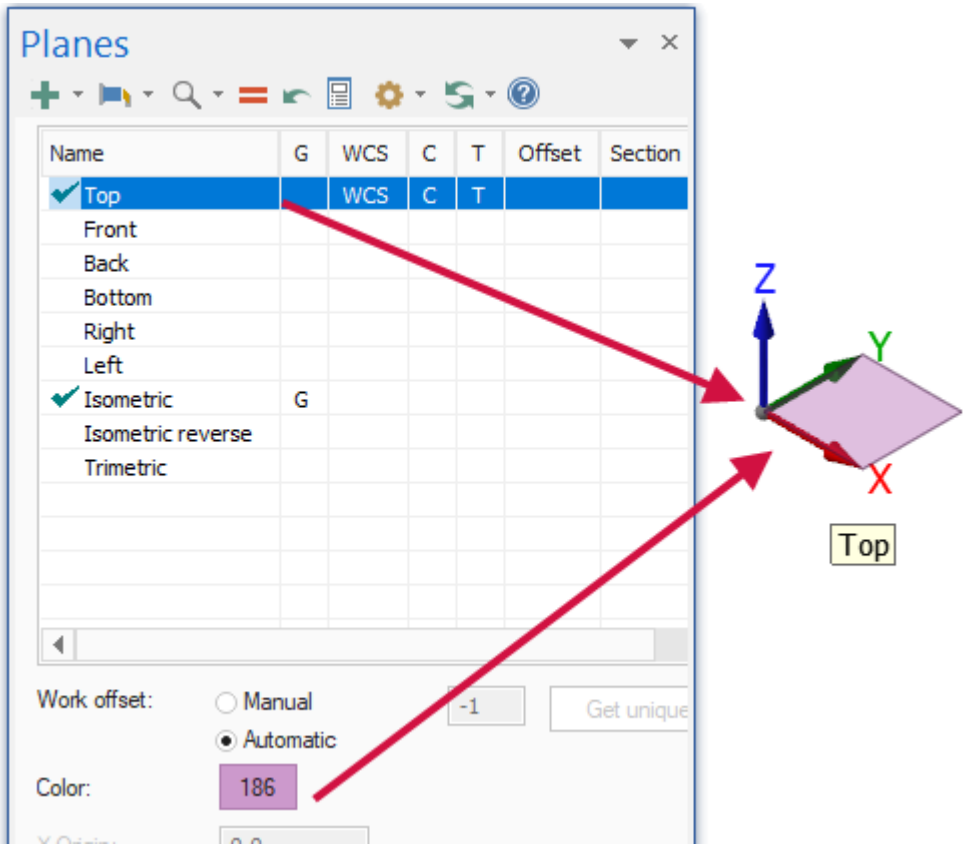
When you select **Find a plane**, **from plane**, the plane gnomons display in the graphics window in the colors configured for the planes in the Planes Manager, making it easier to identify which plane to select. To control which planes display, use the **Display** column in the Planes Manager.





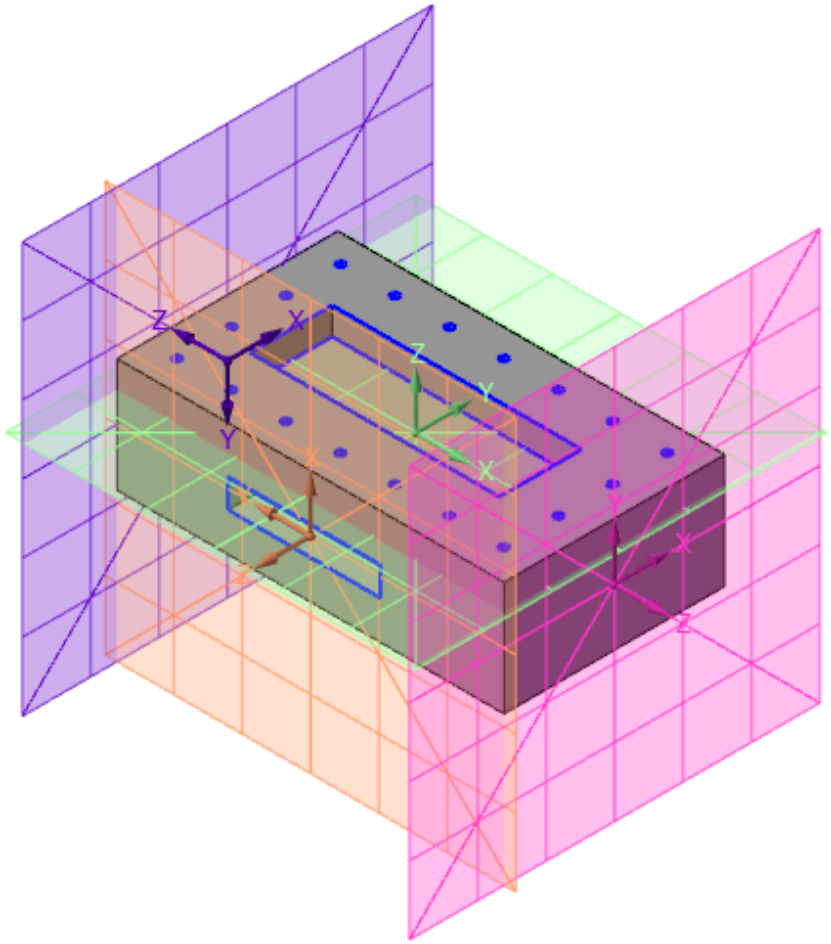
## Gnomon's XY Pane Displays in Color

The XY pane of the plane's gnomon can now be configured for each plane using **Color** in the Planes Manager. Displaying the XY pane in different colors makes it easier to identify the plane in the graphics window.

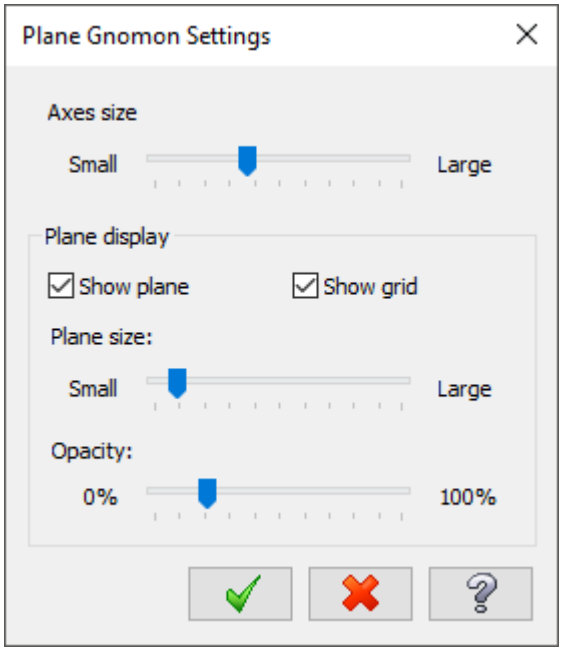


## Planes Axes and Grid Settings

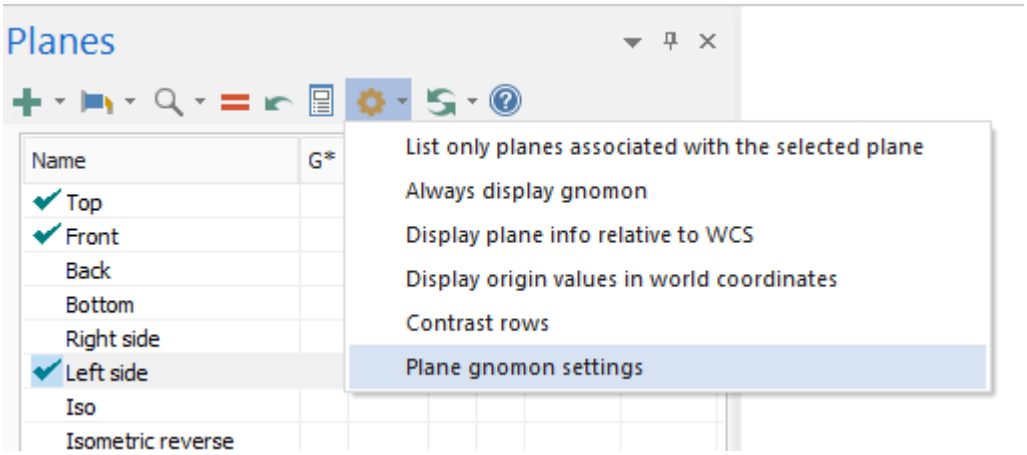
You now have control over the plane gnomon's size, as well as enabling a translucent plane grid in the **Plane Gnomon Settings** dialog box.



The plane grid's opacity and size can be configured, as well as the axes size.



Select **Plane gnomon settings**, under **Display options** in the Planes Manager to access these options.




## Quick Settings Saved as Defaults

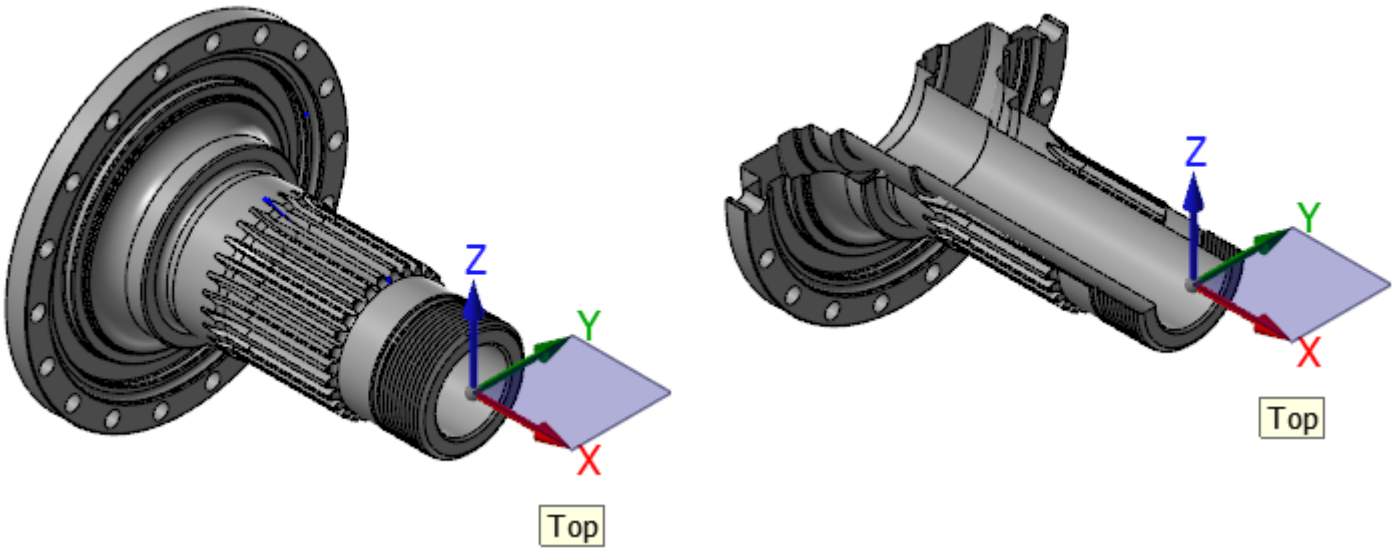
You can choose to save any settings you make in a dialog box launched from a tab as configuration defaults. Previously, you had to set the configuration defaults in the **System Configuration** pages.

The following quick settings dialog boxes will now display a prompt allowing you to save the settings to the configuration file, or use them for that session only.

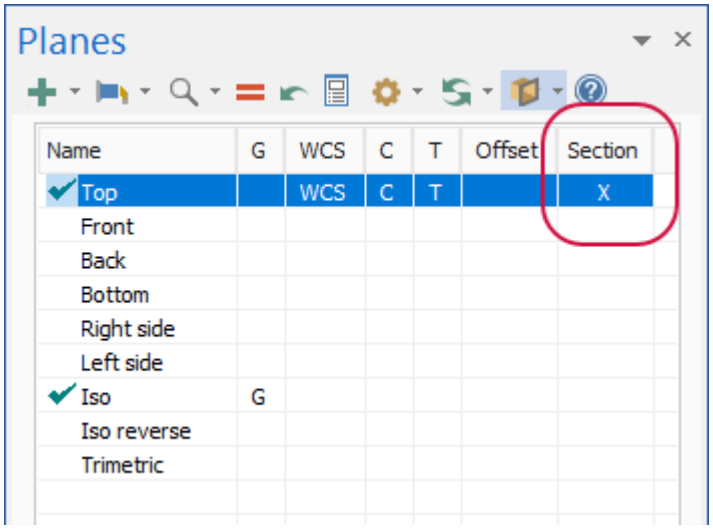
- **Home**, **Attributes**, **Entity Attributes Manager**
- **Drafting**, **Dimension**, **Drafting Options**
- **View**, **Appearance**, **Shading**
- **View**, **Toolpaths**, **Advanced Toolpath Display**
- **View**, **Grid**, **Grid**
- **View**, **Viewsheets**, **Viewsheets**

## Section View

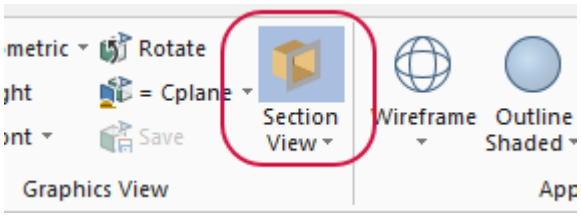
The Planes Manager allows you to create a section view by cutting a plane to see a section of the part.




To create a section view using an existing plane, click in the **Section** column of the Planes Manager on the plane you want to use.

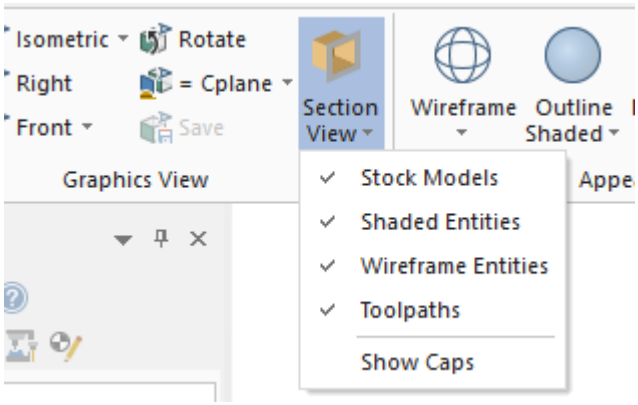



**Section View** toggle on the **View** tab, or **Section View** on the Planes Manager must be selected to show the section view in the graphics window.




When **Section View** is toggled off, the section view in the graphics window no longer displays, but the **X** is still displayed in the **Section** column. If **Section View** is toggled on, select the **X** in the **Section** column to remove that section view if you no longer need it.

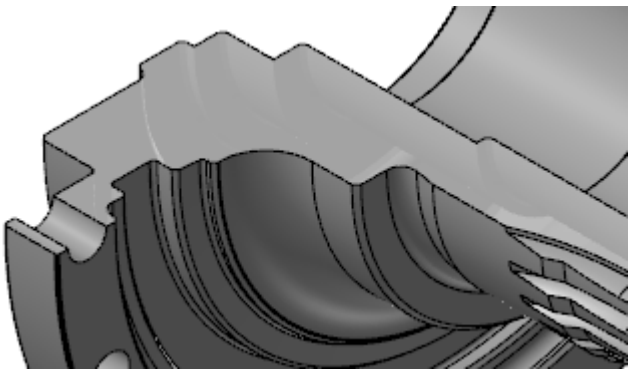
The **Section View** toggle has a list of entities that can be displayed. Choosing the entity from the list enables the section view for that entity type. The entities that can be enabled for section view include:



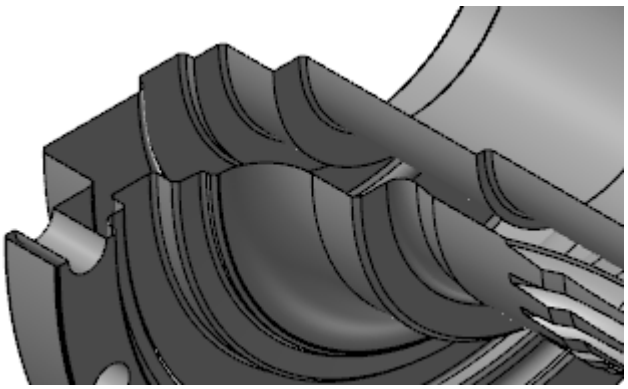

- **Stock Models**
- **Shaded Entities**
- **Wireframe Entities**
- **Toolpaths**
- **Show Caps**

**Show Caps** fills in the section of the part with a solid cap as shown in the following images:

**Show Caps on**

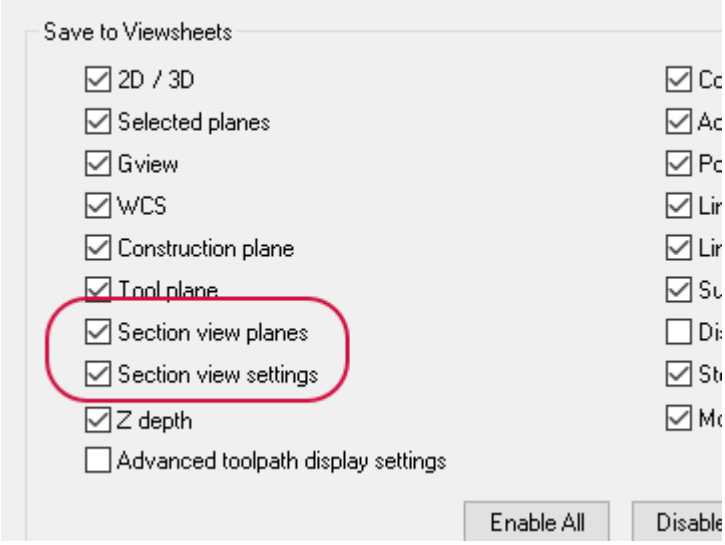


**Show Caps off**



You can also create a section view when creating a new plane in the **New Plane** function panel. **Section View** must be toggled on to be live during plane creation and editing. Select **Section** under **Set As** to set the new plane as the section view.

You can save a section view plane and its settings as part of a viewsheet. These options are set in the **Viewsheet** page, under **Screen** in the **System Configuration** dialog box. When you select this option, the section state for each new plane is saved within the viewsheet.

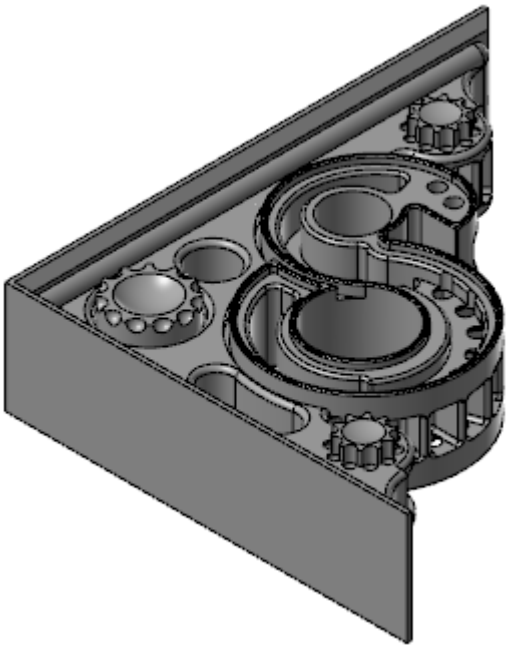


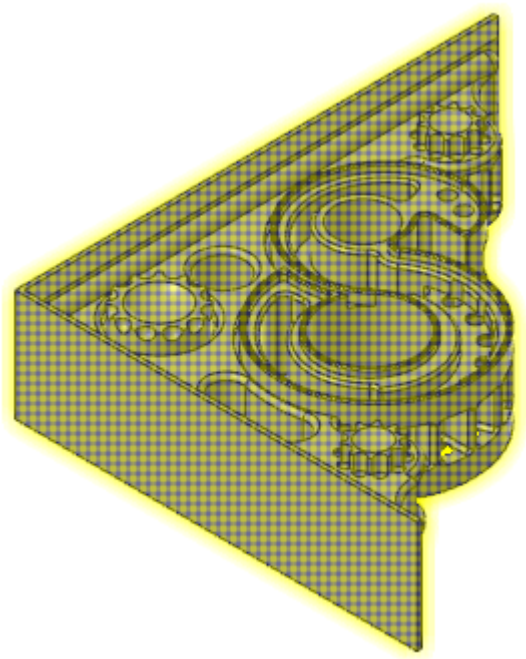

## Selection Enhancements

Listed below are enhancements for how you select entities in Mastercam 2019.

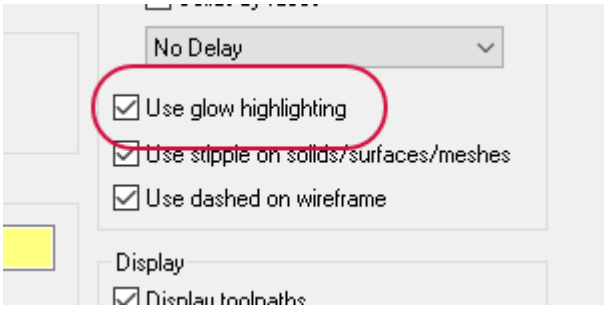
### Glow Highlighting

Entities have a glow highlight when you hover over or select them in the graphics window.





You can enable glow highlighting by selecting **Use glow highlighting** on the **Screen** page in the **System Configuration** dialog box.

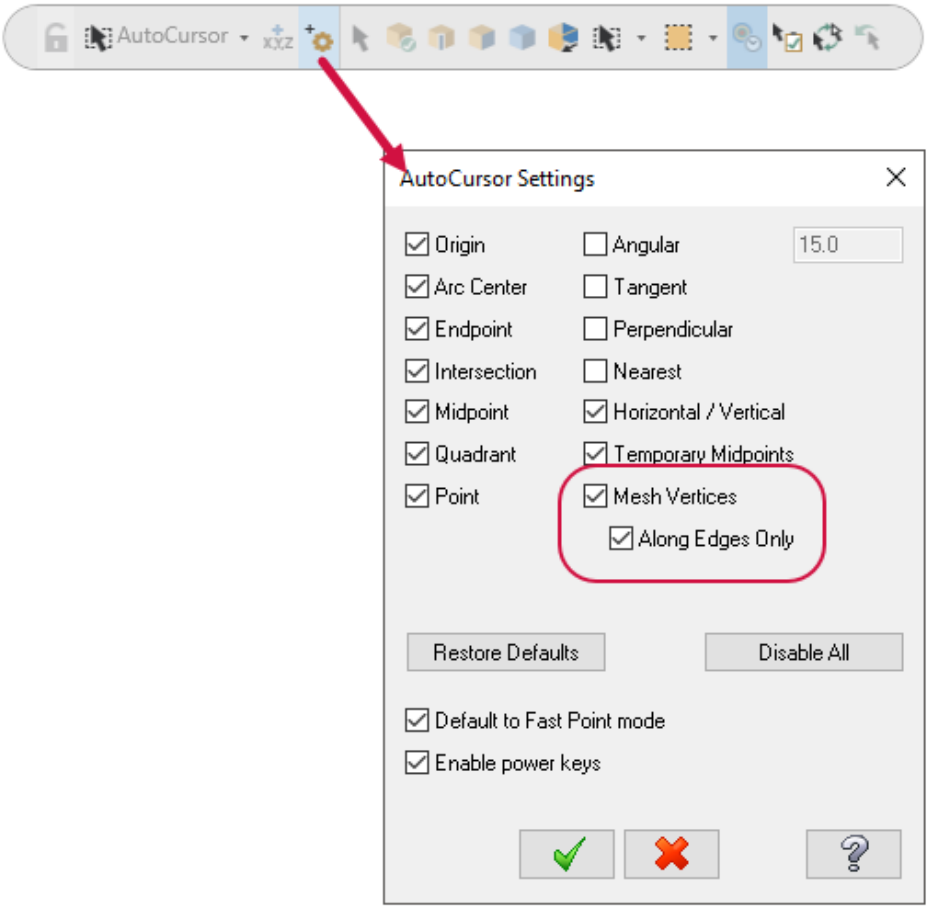
- 


Change the auto highlight glow colors on the **Colors** page of the **System Configuration** dialog box.

## Mesh Node Selection

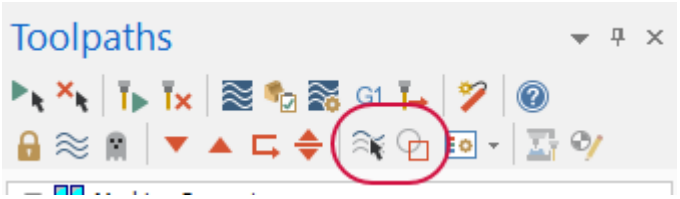
**AutoCursor** now recognizes the endpoints of Mastercam mesh vertices. Mastercam recognizes positions on all mesh vertices, or on vertices that reside along mesh edges. This increases flexibility during CAD creation and dimensioning.

You configure how **AutoCursor** snaps to mesh vertices in the **AutoCursor Settings** dialog box. Select **Mesh Vertices** only to enable snapping to all mesh vertices. Select **Along Edges Only** to only include mesh vertices that reside on the edges of the mesh.



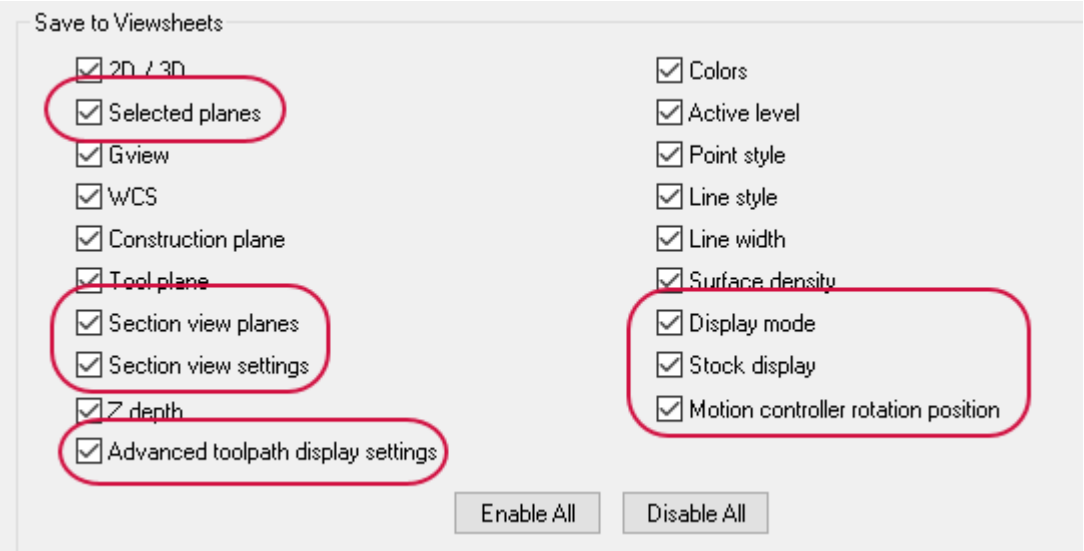

## Toolpaths Manager Display Options

The state of the **Only display selected toolpaths** now persist across Mastercam sessions. The state persists whether it is set from the Toolpaths Manager or the **Quick Access Toolbar**.



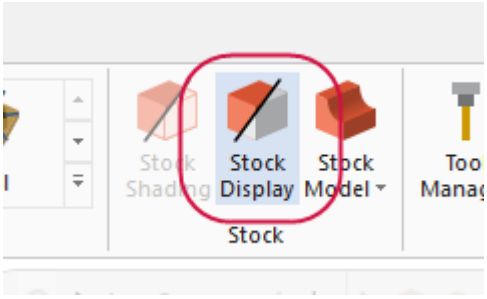
## Viewsheet Enhancements

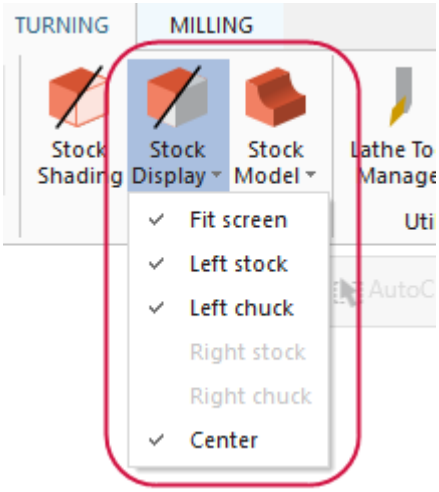
There are seven new options in the **Viewsheets** page in the **System Configuration** dialog box, or by right-clicking a viewsheet tab and selecting **Settings**.




You can save planes used to create section views by selecting **Section view planes**. Selecting **Section view settings** allows you to save the default options. Please see "Section View" on page 36 for more information on section views.

Viewsheet settings also include a new **Stock display** setting. Viewsheets can save multiple stock display settings for Lathe and Mill-Turn parts, but can only save a single stock display setting for Mill and Router parts.

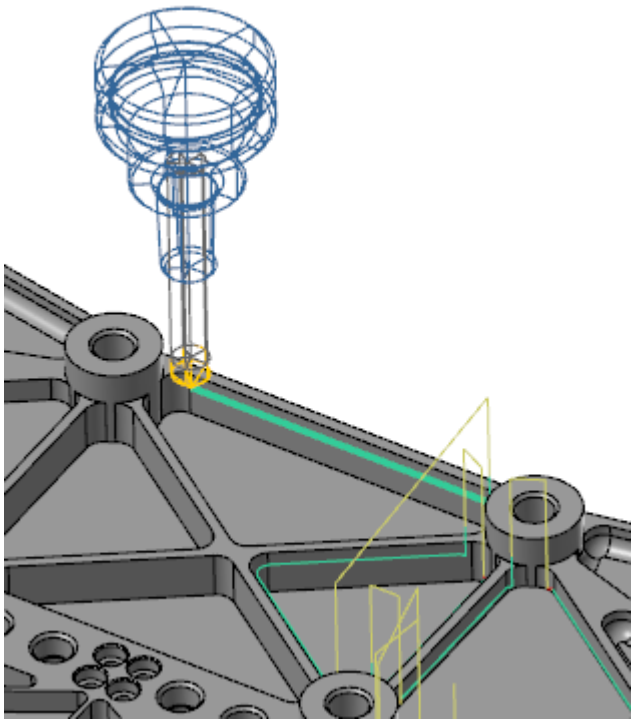


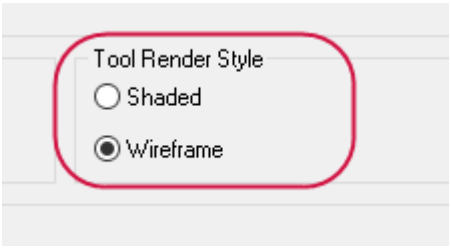

If you have a 3Dconnexions device, you can save the current rotation position in a viewsheet. Select the **Motion controller rotation position**.

## Wireframe Tool Display

When using Classic Backplot, you now have the option to display the tool as wireframe, or shaded.



You set this option in the **Backplot Options** dialog box, by selecting **Shaded** or **Wireframe** under **Tool Render Style**.



# DESIGN ENHANCEMENTS

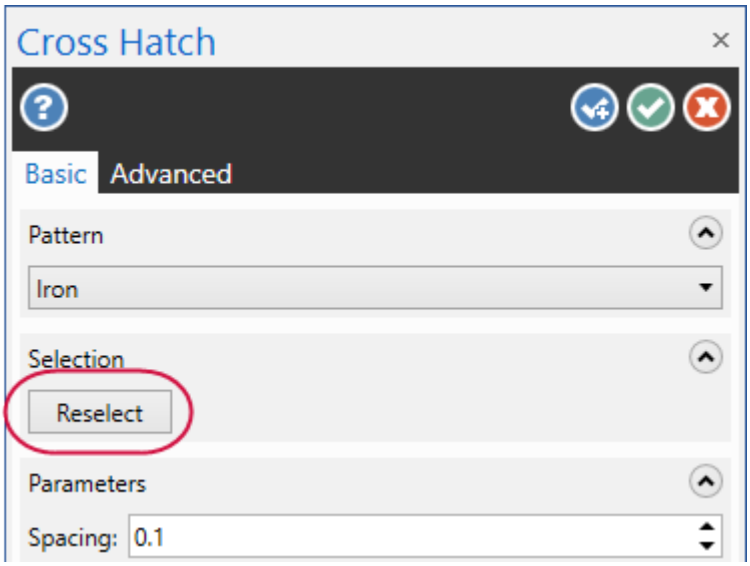
Listed below are enhancements made to design functions, such as **Cross Hatch**, **Push-Pull**, and **Point Nodes**.

## Drafting Enhancements

Listed below are enhancements made to functions on the **Drafting** tab, including **Cross Hatch** and **Note**.

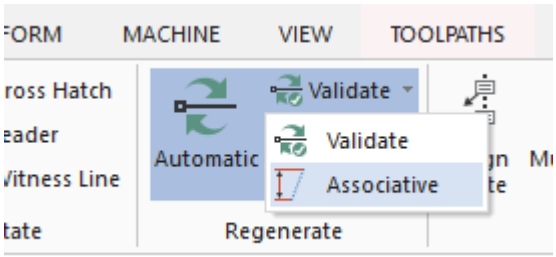
### Cross Hatch

In previous releases, you had to exit **Cross Hatch** if you wished to re-chain geometry. A **Reselect** button has been added to allow you to re-chain geometry without leaving the function.

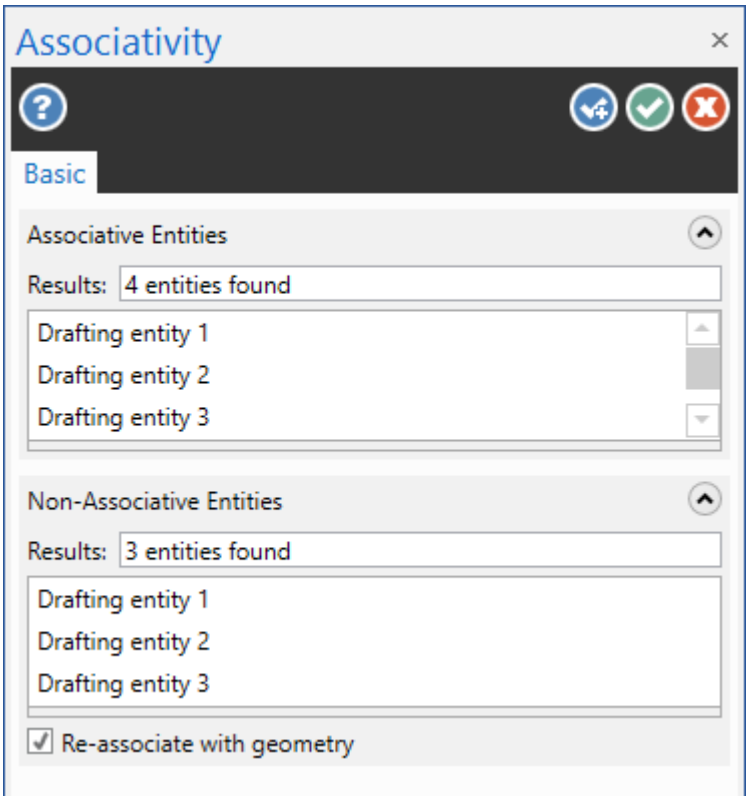



### Drafting Associativity

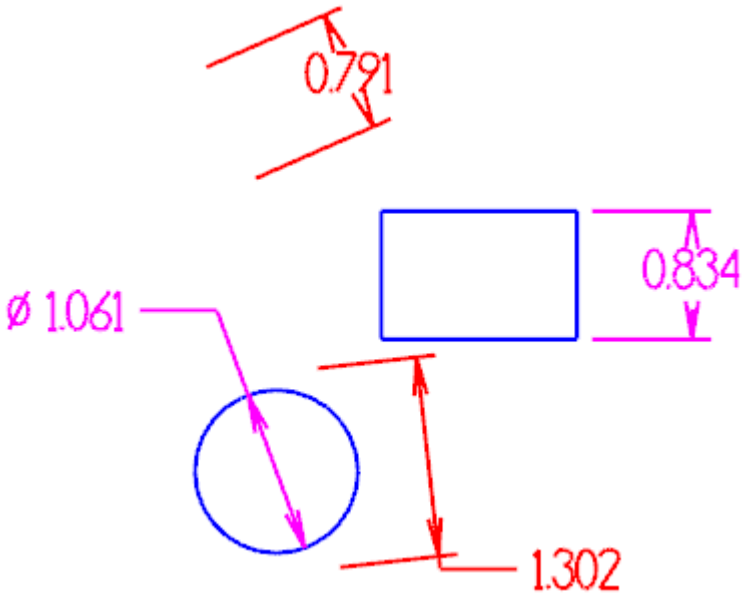
A new function, **Associativity**, has been added to Mastercam that shows which dimensions are associated to geometry. Access this function by selecting **Associative** under **Validate** on the **Drafting** tab.



Once you have entered the function, select the drafting entities you wish to check, and then press [**Enter**] or select **End Selection**.




Mastercam displays dimensions that are associative with the Result color (purple) and dimensions that are not associative with the Group color (red). Rolling your mouse over associative dimensions highlights the geometry it is associative to. Mastercam also displays how many selected dimensions are associative.



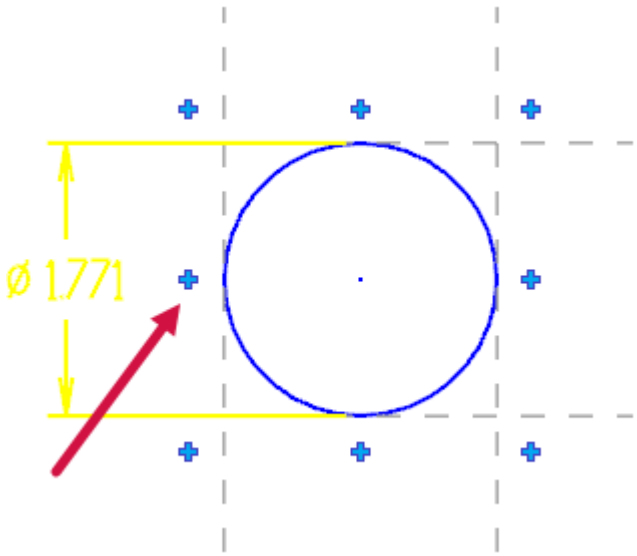
## Note Height

Use **Select Height** in the **Note** function panel to quickly change the height of your note text to be the same as another note. Select the button, and then select a note from the graphics window. The **Height** parameter updates to match the height of the selected note.

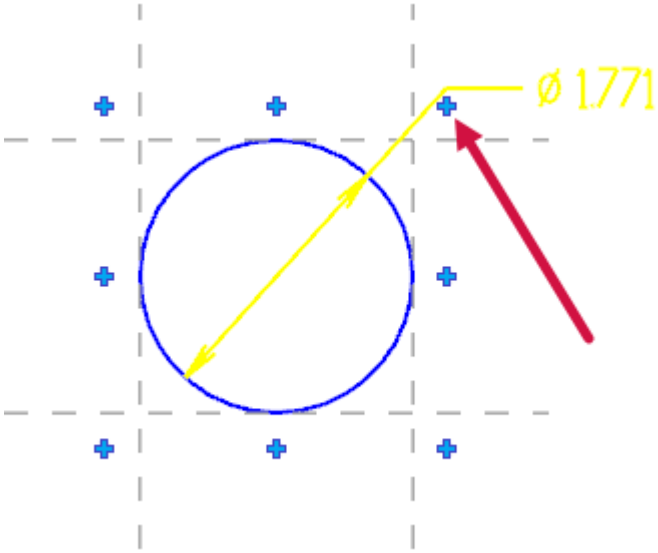
## On-screen Triggers

When you dimension an arc or a circle, small triggers appear in each quadrant of the arc or circle and offset from outside of the arc or circle in 45 degree increments. You can use these triggers to switch between creating tangent and circular dimensions.

Hover over a quadrant trigger to create a tangent dimension, as shown below:



Hover over an offset trigger to create a circular dimension, as shown below:




## Solids and Model Prep Enhancements

Listed below are enhancements made to the functions located on the **Solids** and **Model Prep** tabs, including **Push-Pull** and **Hole Axis**.

### Boolean

Since the introduction of the function panel interface, some users experienced delays with **Boolean** when they selected a complex part with multiple operations and/or tool bodies.

In Mastercam 2019, we have changed the workflow for **Boolean** to minimize processing time. You no longer need to select the target body and the tool body to display the function panel and edit the settings. The function panel now displays as soon as you enter the function.

### Chamfer

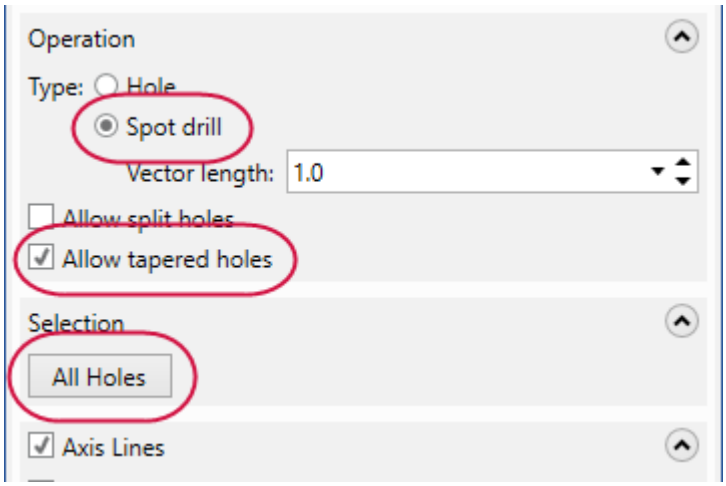
In Mastercam 2019, you can chamfer entities, such as holes in cylinders, where the angle of the newly created chamfer face will vary.

### Disassemble

**Disassemble** now supports the selection of cylinders, cones, and other solid models without flat faces.

## Hole Axis

**Hole Axis** has several enhancements for Mastercam 2019. You can now include tapered holes in your selection, as well as have the function draw vectors for holes that you will spot drill. Additionally, a new **All Holes** button selects of all the holes in your part with one click.

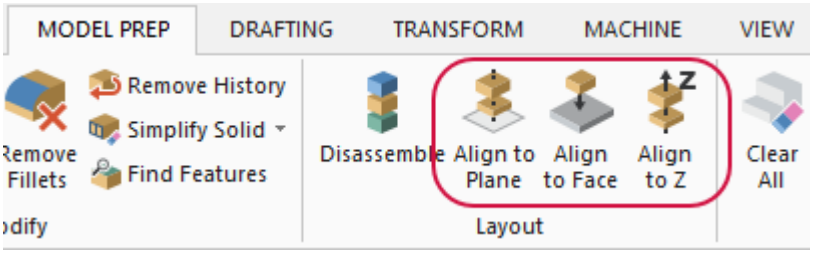



Once you create your axis lines, use the **Advanced** tab to change how the axis lines look and their level location.

Mastercam also now displays the diameter of the hole in the graphics window instead of its radius.

## New Align Functions

Two new align functions are available on the **Model Prep** tab, and **Position** has been renamed to **Align to Face**.



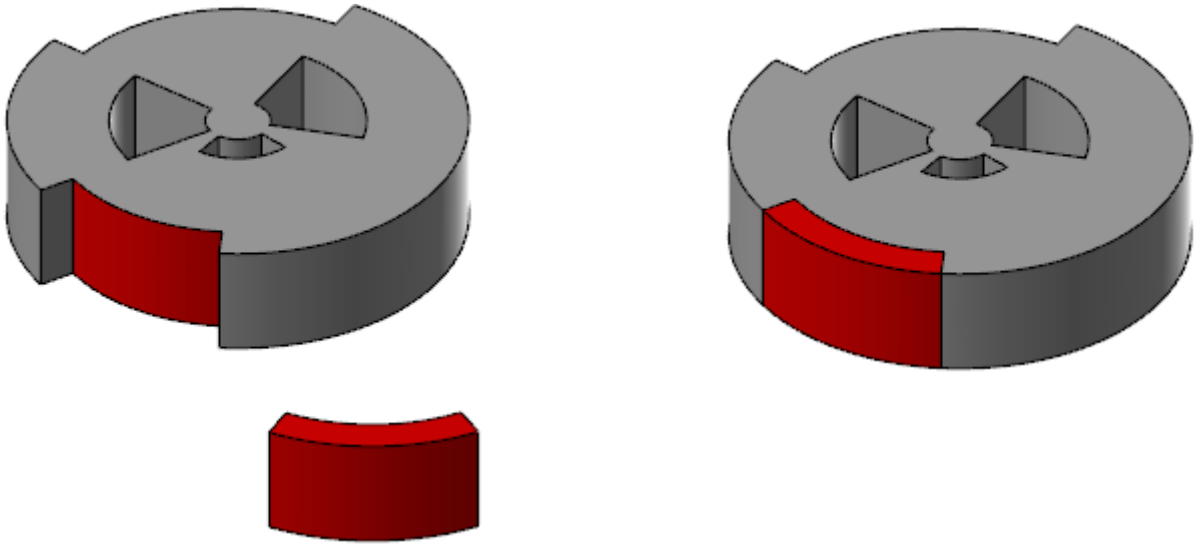

### Align to Face

**Align to Face** uses the dynamic gnomon to control orientation when mating solid models. This is useful for aligning inserts to holders when creating 3D tools. To learn more about 3D tools, please see "3D Tool Support" on page 75.

After placement, you can adjust it even further with the dynamic gnomon. **Align to Face** allows you to work in three modes: **Coincident**, **Perpendicular**, and **Parallel**.

**Coincident** allows you to copy or move a solid by mating a face on the solid to another solid's face. **Parallel** and **Perpendicular** require you to select a linear edge on the solid body that is being moved and on the target solid body. When you select a linear edge on the solid body that is being moved, Mastercam aligns the X axis of the gnomon to that edge. When you select an edge on the target body, Mastercam aligns the X axis of the solid being moved to be perpendicular or parallel to that edge.

**Align to Face** also supports mating cylindrical faces. After you select the faces, **Align to Face** rotates each body into position and mates each face at its center. It rotates the object into position, if the bodies are not in same plane or of the same radius.



### Align to Z

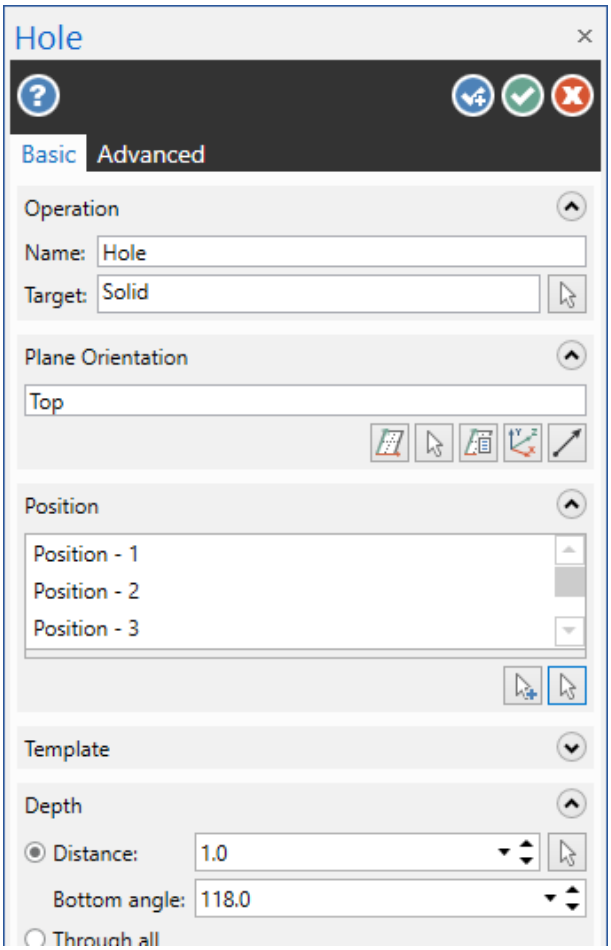
**Align to Z** aligns a solid body that is used for a turning operation to a new WCS or an existing construction plane along the Z axis. This lets you orient the body so that is ready to machine. This function is also located on the **Lathe Turning** contextual tab.

### Align to Plane

**Align to Plane** aligns a solid body to a new WCS or an existing construction plane. You can align an edge of the solid to the X axis of the plane or WCS, and a selected point on the solid to the origin of the WCS or plane.

## New Solid Hole Function

A new function called **Hole** is now available on the **Solids** tab. This function automates the punching of cylindrical holes in primitive solids, saving you from creating circles and extruding them.




Set the hole dimensions and the hole type. You can create a simple cylinder or select from the following hole types:

- Counterbore
- Countersink
- Counterdrill
- Taper

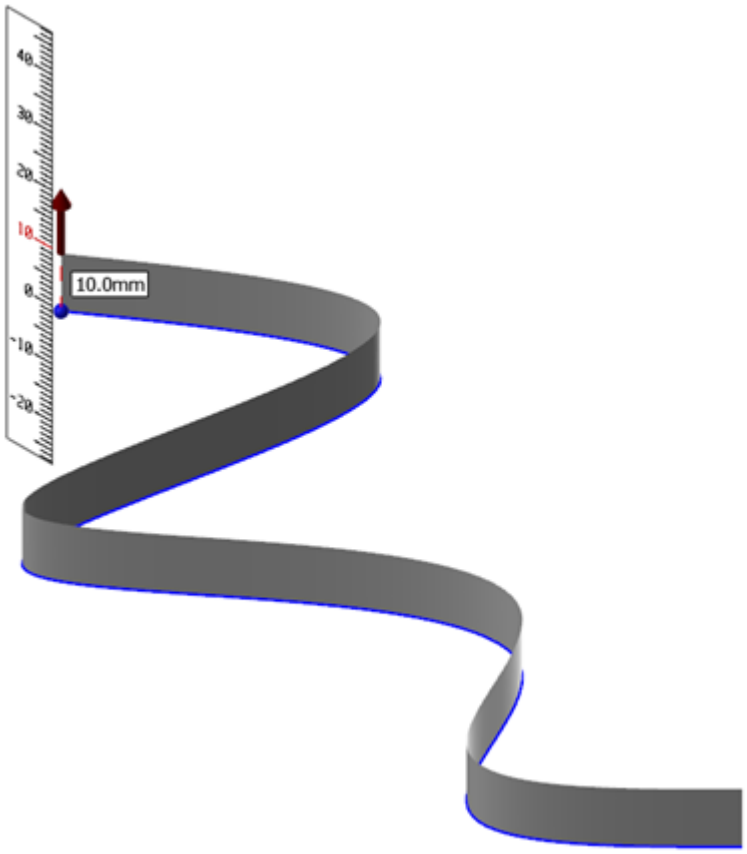
Click in the graphics window to place it on the solid. Press [**Enter**] to view the new hole. The hole remains live, and you can place copies of it on the part or modify any of its parameters until you exit.

Create and save commonly used holes in a preset library. You can use these presets in future sessions of Mastercam. Mastercam converts hole dimensions stored in a preset library to the current units (inch to metric, or metric to inch).

The Solids Manager displays this function's operations.

## Push-Pull

You can now use **Push-Pull** to create solid bodies from open, closed, and nested wireframe geometry, as well as sheet bodies.



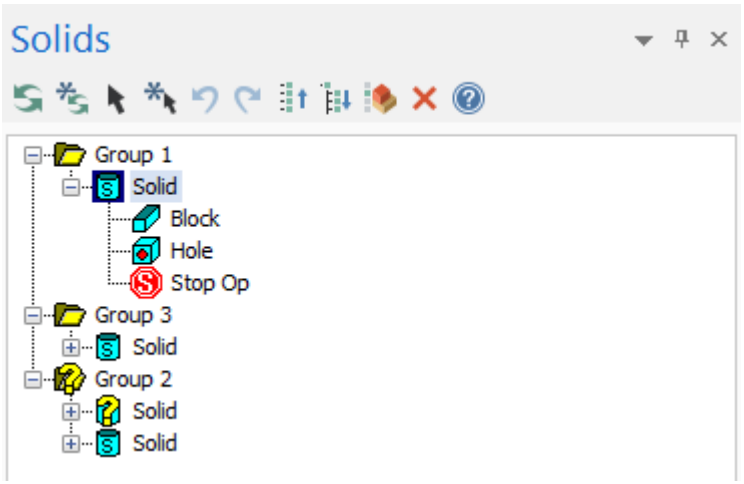

Once inside the **Push-Pull** function panel, click **Select** under **Chaining** to bring up the **Chaining** dialog box. Select the wireframe you wish to create a solid body from, and click **OK**. You can then use the control arrow in the graphics window to create a solid body.

**Push-Pull** also supports surfaces. You can select a single surface and use the arrow to create a solid body, or grab connected surfaces to move as a single sheet. A new option, **Maintain source surfaces**, prevents the original surfaces from being absorbed and deleted.

## Solids Manager

Solids Manager includes new organization features. These improvements include:

- You can now create multiple solid groups to hold bodies. Additionally, you can edit these groups and move them, reorder them, or delete the bodies inside them. Rename the groups with a slow double-click. Use [**Shift+click**] and [**Ctrl+click**] to move multiple solid bodies in and out of groups.
- Even when a group folder is minimized, the folder icon displays the state of the solid bodies within it.
- The toolbar now includes several options that were previously only available from the right-click menu.
- In the right-click menu, **Analyze Entity Properties** has replaced **Attributes** so you can quickly get more information on a selected body.

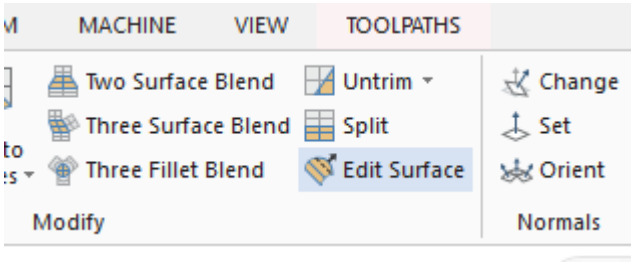



# Surface Enhancements

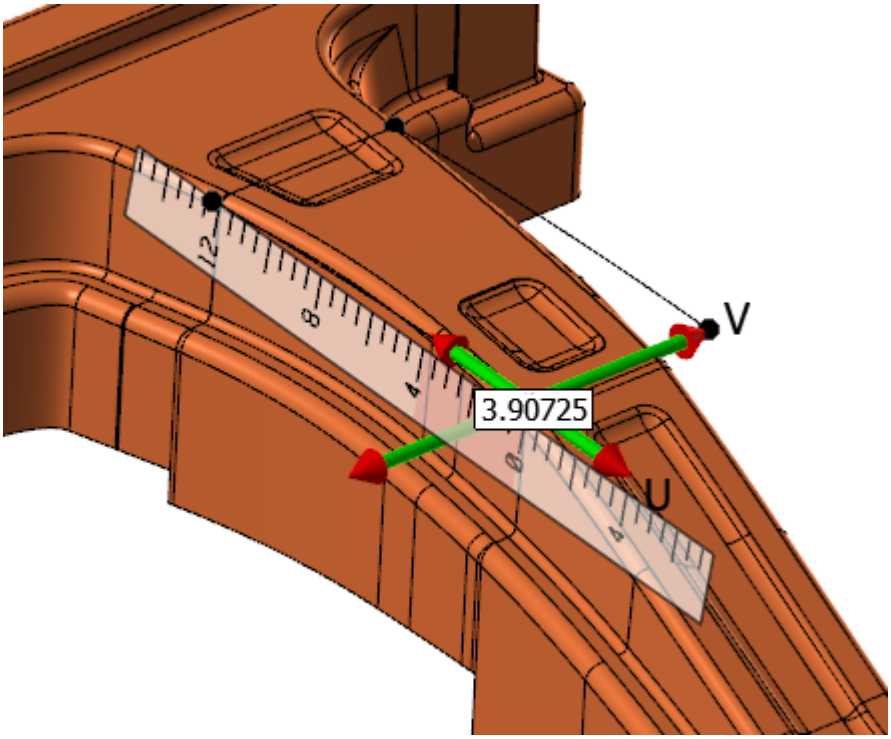
Listed below are enhancements made to functions on the **Surfaces** tab, including **Surface From Solid** and the new **Edit Surface** function.

## New Edit Surface Function

A new function, **Edit Surface**, is now available on the **Surfaces** tab.



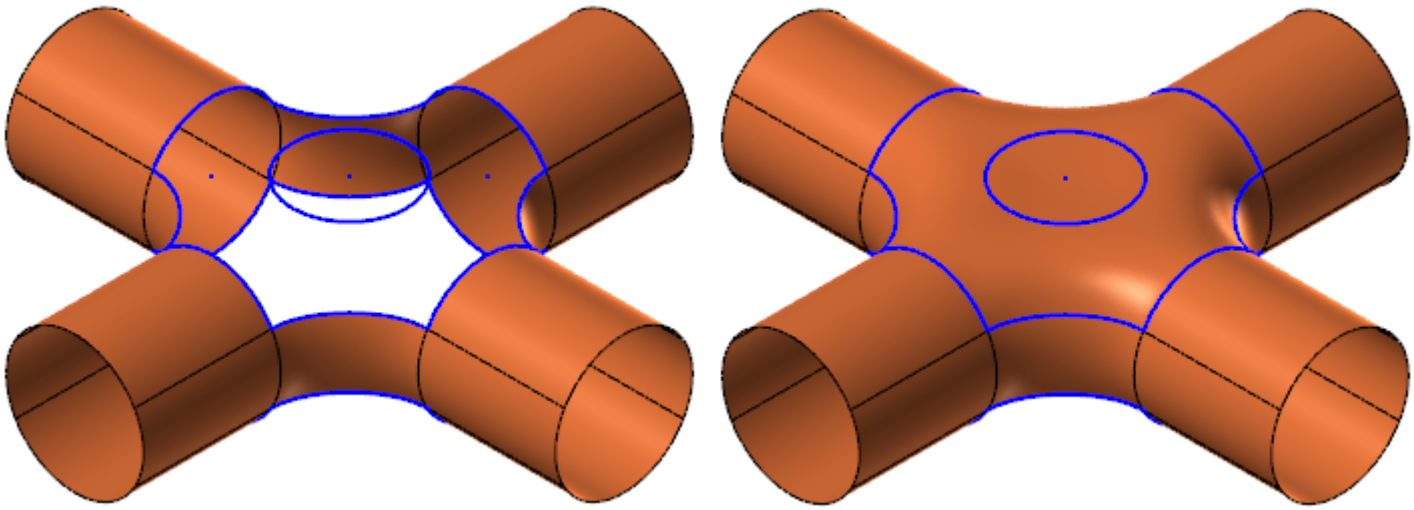
**Edit Surface** allows you to make edits to existing surfaces. It works much like the **Edit Spline** function that was introduced in Mastercam 2017. You can modify the tangenices and magnitudes of the curve at nodes points, or edit tangencies at the vectors of control points. You can also add or remove node points and isocurves to increase your editing control and to make finer adjustments.



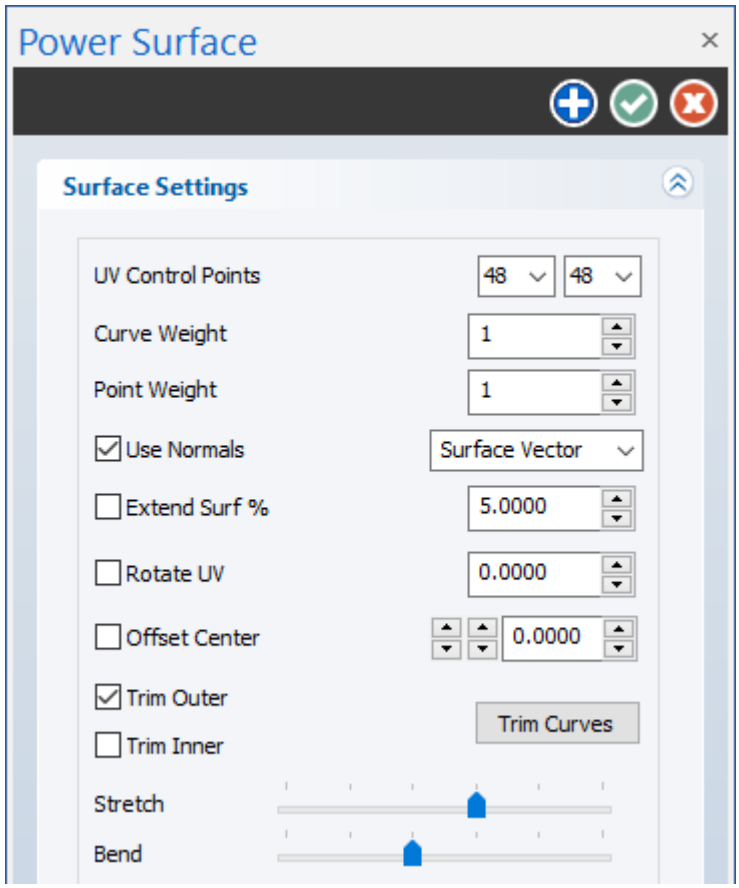

**Edit Surface** includes tools that allow you to assess and analyze the surface you are working on. You can add or remove the visibility of flowlines along the U or V axes. Curvature combs represent the curvature of the surface and display curvature continuity or discontinuity.

## Power Surface

A new function, **Power Surface**, is now available on the **Surfaces** tab in the **Create** group. Power Surface gives users the capability of easily creating surfaces from curves and influenced by adjoining surfaces.

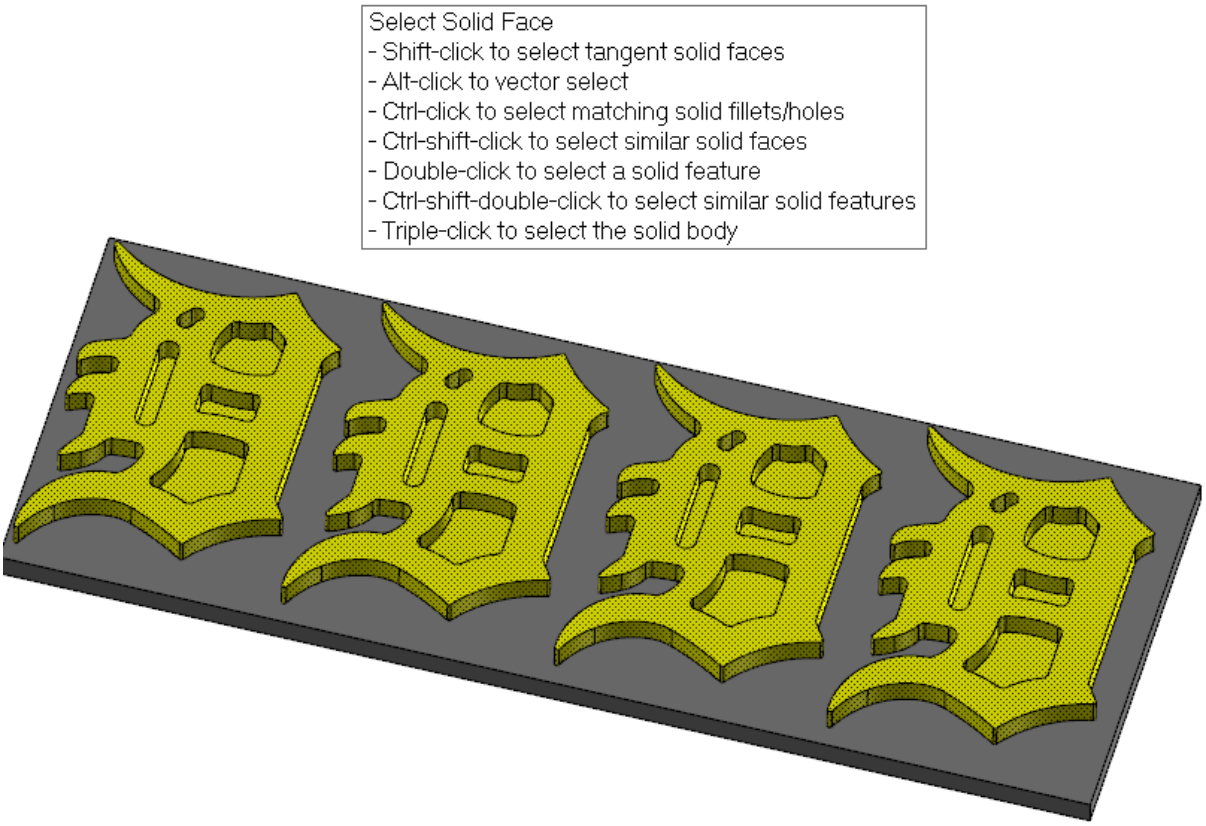


After you have selected the curves needed to crate a new surface, the **Power Surface** function panel displays allowing you to edit the create surface.



### Surface From Solids

The selection options that were available in Mastercam 2018 are now available when using **Surface from Solids** on the **Surfaces** tab. You can now use the following short-cut keys:




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

## Wireframe Enhancements

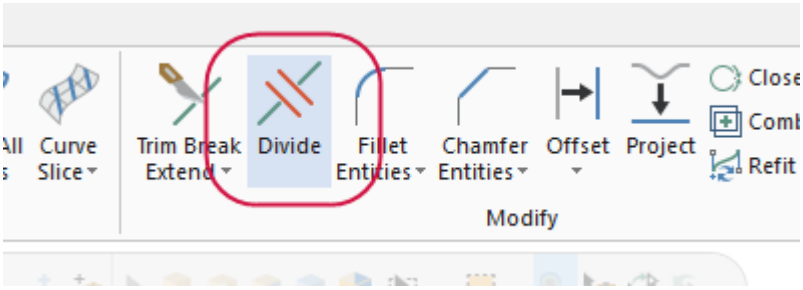
Listed below are enhancements made to functions on the **Wireframe** tab, including **Point Nodes** and **Join Entities**.

### Curve One Edge and Curve All Edges

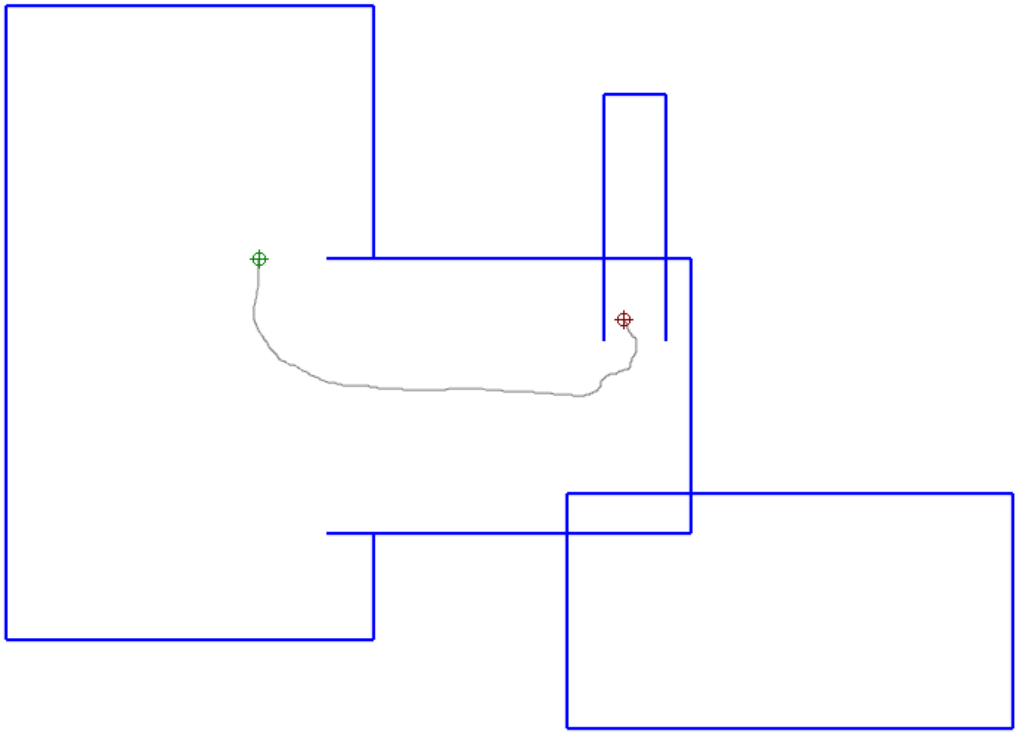
**Curve One Edge** and **Curve All Edges** support creating edge curves on mesh files.

## New Divide Function

**Divide** has been broken out from the **Trim Break Extend** function into its own function, and with more functionality. **Divide** is located on the **Wireframe** tab.

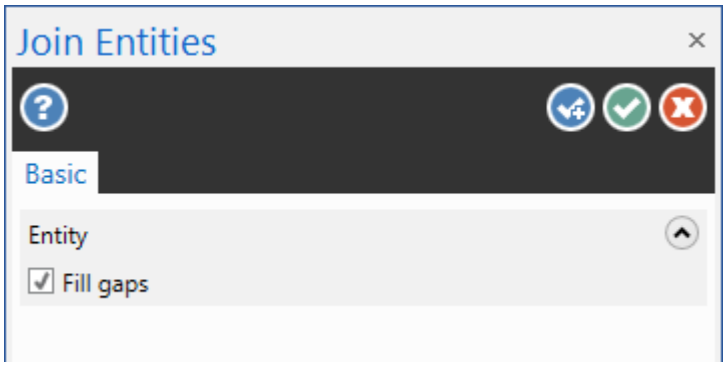



As with previous Mastercam versions, you have the option to use **Divide** to trim or break entities. You can also now hold down the left mouse button and use it to trim or break entities as your mouse encounters them.



## Join Entities

**Join Entities** (located under **Trim Break Extend** on the **Wireframe** tab) now displays as a function panel. This allows you to do multiple operations without having to reenter the function.

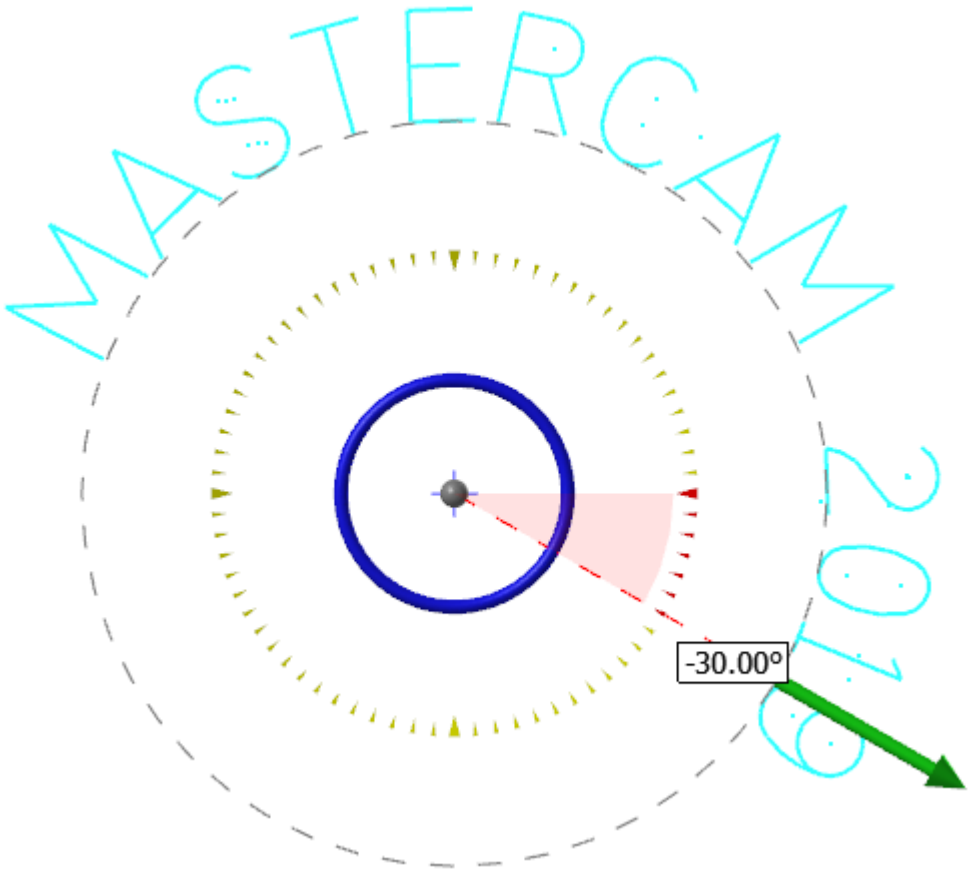



## Offset Chains

The workflow for **Offset Chains** has also been modified. In previous versions, the chaining direction determined the offset. Now, you are prompted to set the side and distance of the offset by clicking in the graphics window or by selecting an **AutoCursor** position.

## On-screen Controls

**Create Letters** and **Bolt Circle** use the origin, polar, and linear on-screen controls that were previously only available when using **Stretch** or translation functions.

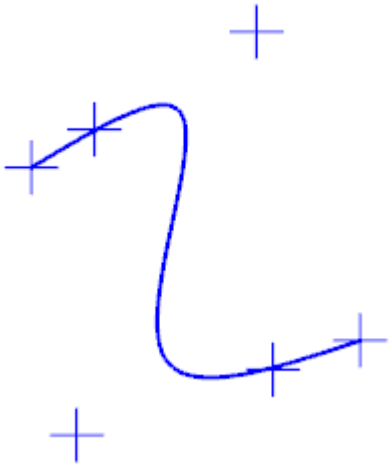


You can change the origin by clicking on the center ball and moving it. Click on the arrow and drag it in or out to edit the radius. Click on the blue circle to change the start or rotation angle.

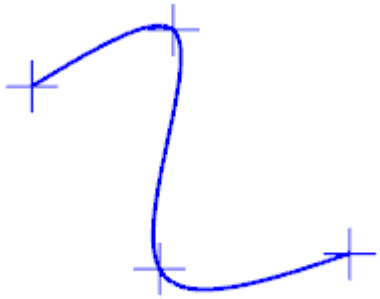
## Point Nodes

**Point Nodes** no longer creates points on the control points of a spline.

**Mastercam 2018**

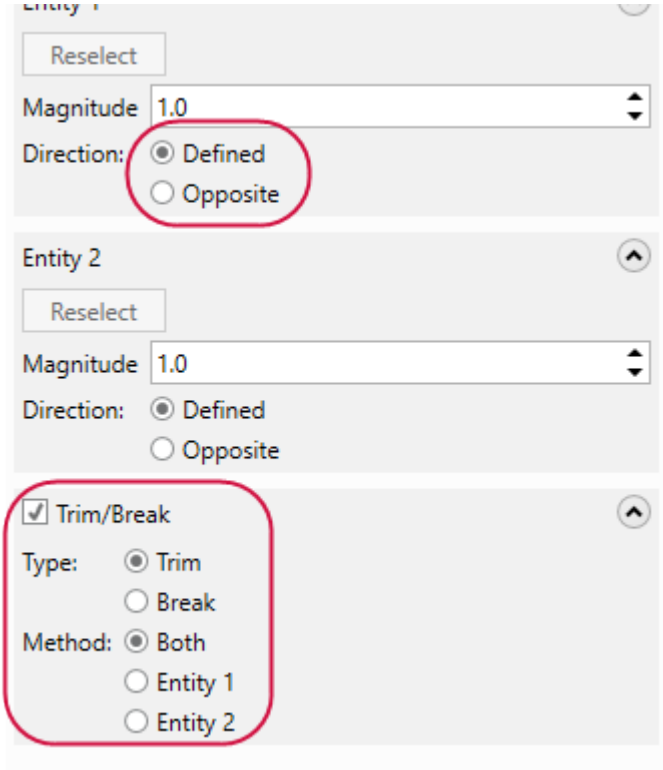


**Mastercam 2019**



## Spline Blended

**Spline Blended** now includes new direction controls that allow you to change the magnitude of each spline. Additionally, you can now trim or break either or both splines.

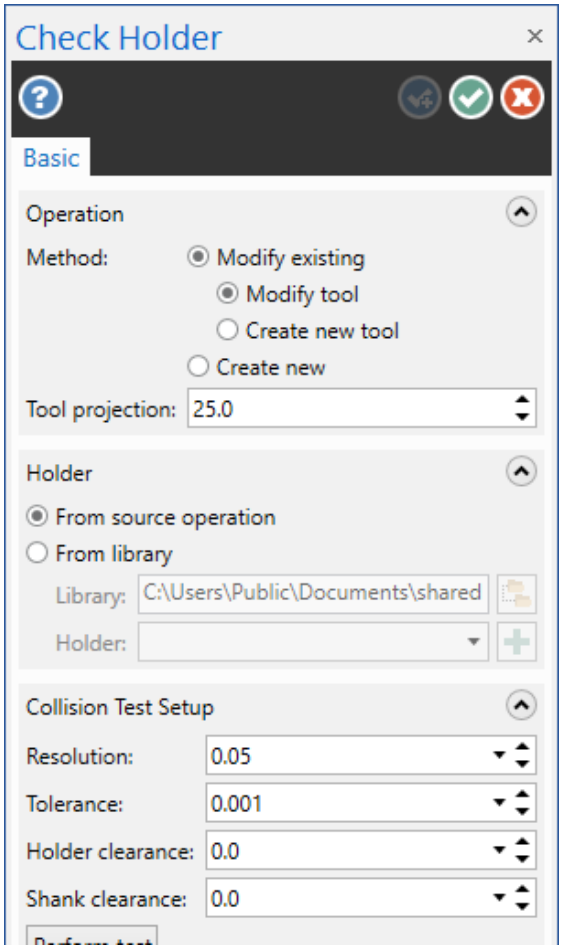


# MILL ENHANCEMENTS

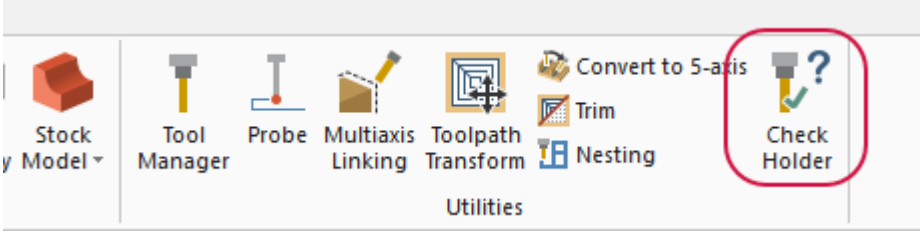
Listed below are enhancements made to the Mill product, including improvements to the 2D, 3D, and Multiaxis toolpath types.

## Check Holder

**Check Holder**, which was previously an Add-In in earlier Mastercam releases, is now integrated with Mastercam and has been improved. Use **Check Holder** to check an operation's tool holder for interference with the part. It calculates areas where there are interferences between the holder and the part, and tells you the minimum tool length required to avoid it. Before starting **Check Holder**, you should select the desired operation in the Toolpaths Manager.

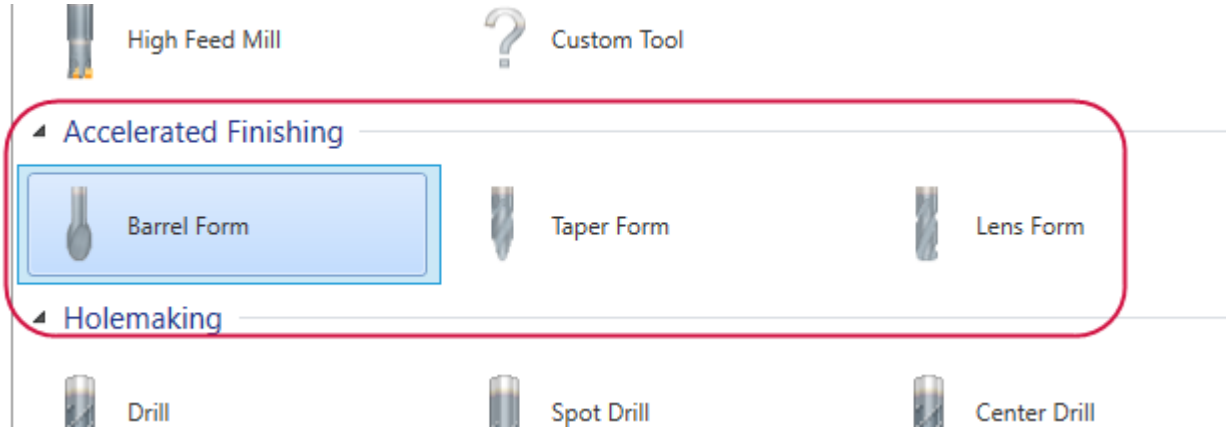



**Check Holder** is located on the **Mill Toolpaths** contextual tab.

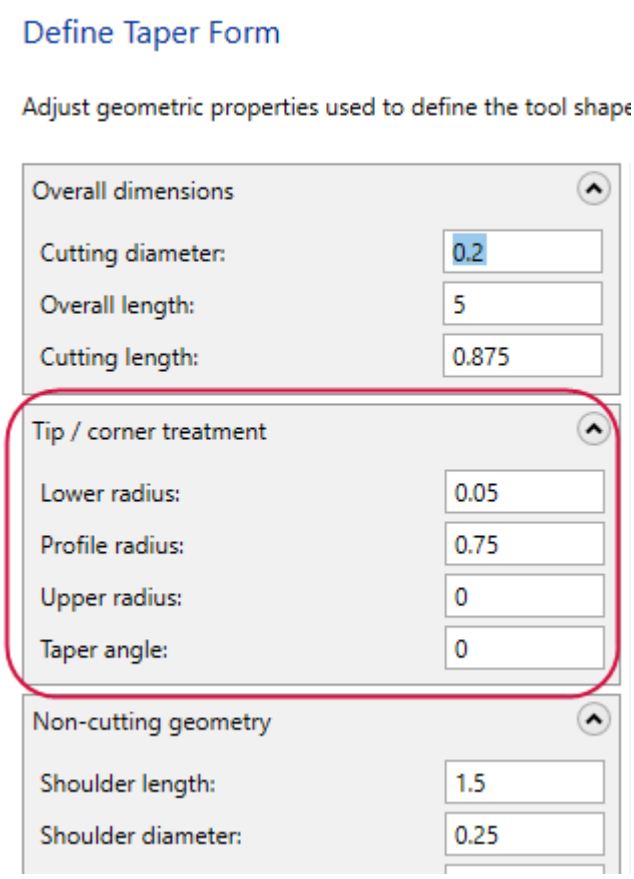


## Mastercam's Accelerated Finishing™

Mastercam's Accelerated Finishing technology addresses today's finishing tool geometries and processes, resulting in greater efficiency and higher machining productivity when using Mill. To support this, Mastercam has added two new tool types, Lens Form and Taper Form. Barrel Mill has been moved to the Accelerated Finishing group, and additional variations of this shape are now available.

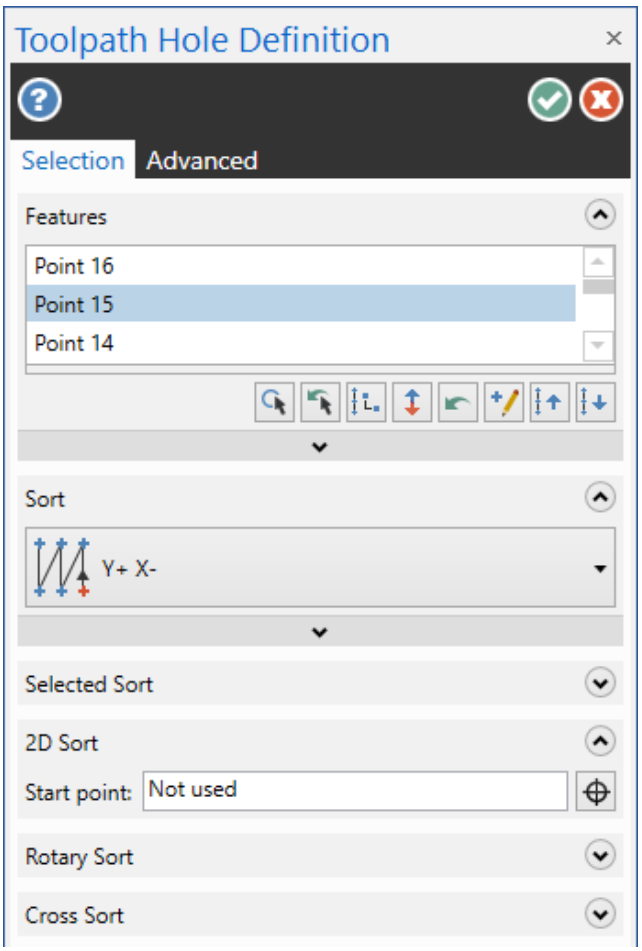


Lens Form and Taper Form are based on Barrel Mill tools, with new tip and profile dimensions.



## New Point Selection Workflow

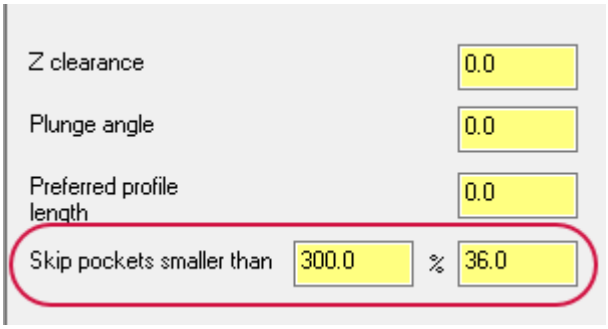
When you select point geometry for Mill, Router, and Canned Wire toolpaths, the new **Toolpath Hole Definition** function panel displays, in place of the **Drill Point Selection** dialog box.




Use the options on the **Selection** tab to pick your points, and make any changes to the sort order. Once you are satisfied with your selections and options, click **OK** to open the toolpaths dialog box.

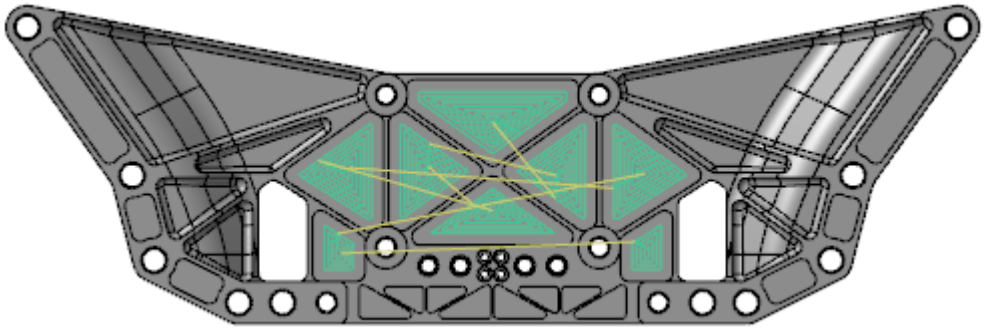
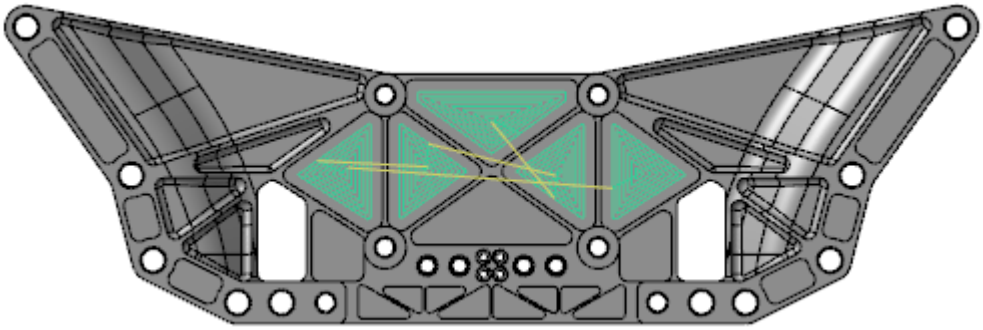
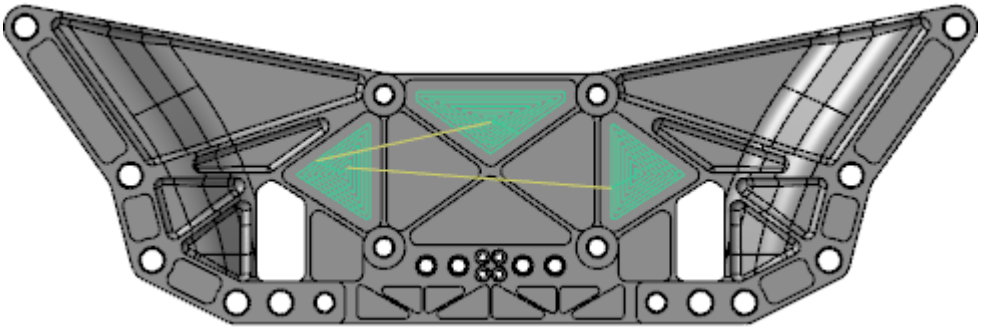
## Skip Pocket Smaller Than

A new parameter is now available on the **Transitions** page for Area Roughing, Dynamic OptiRough, Waterline, Horizontal Area, and 2D Area Mill toolpaths. This new parameter allows you to skip pockets based on the tool diameter percent, rather than by entering a dedicated size of the pocket. When you enter a value for either the **Tool diameter percentage** (on the left) or for the **Minimum pocket size** (on the right), the other parameter updates.

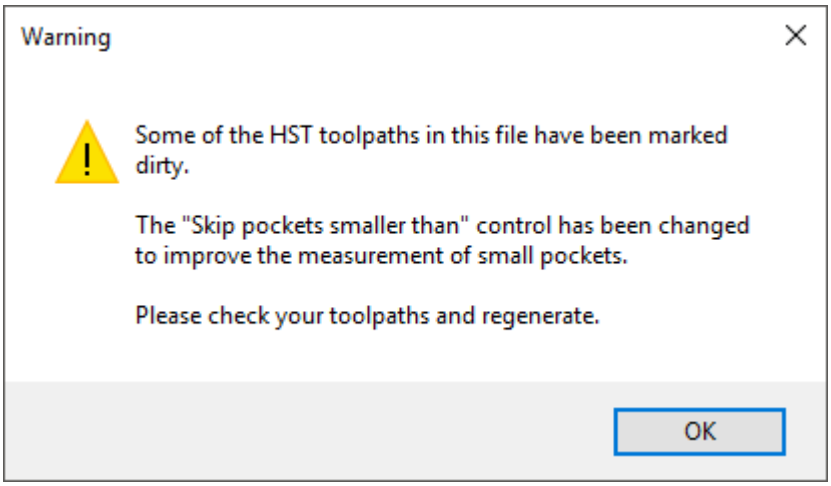


This is helpful when Mastercam thinks that a pocket is large enough to accommodate the tool, but the entry move is so compressed that the tool is effectively plunging into the part. If you want to ensure that the entire surface is machined, you can set these values to **0**. However, if the cutting area is too small for the programmed entry moves, the tool will plunge straight down into the pocket.

The images below show a 2D Area Mill toolpath using a flat end mill tool with a tool diameter of **12.0** (mm) with different **Skip pockets smaller than** percentages.

| Percentage amount | Resulting toolpath |
|---|---|
| 0% |  |
| 300% |  |
| 350% |  |

When opening a file from an older version of Mastercam, your operations might be marked dirty. A warning displays if this is the case.




# 2D Enhancements

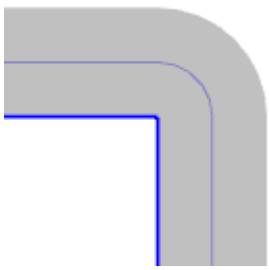
Listed below are enhancements made to 2D toolpath types, such as Contour.

## Miscellaneous Enhancements

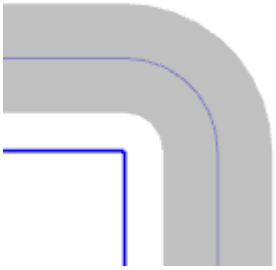
- Thread Mill toolpaths now can control the entry/exit arc length anywhere from 0° to 360°.
- The new **End of Center** option on the **Lead In/Out** page allows you to end the toolpath at the center of the selected arc.

## Maintain Sharp Corners

A new field, **Maintain sharp corners**, is now available on the **Cut Parameters** page for standard 2D Contour and Pocket toolpaths. This new option allows you to roll the toolpath around sharp corners. It is available when you enter a positive value for **Stock to leave on walls**.



- **Roll cutter around corners** set to **Sharp**.
- **Stock to leave on walls** set to **0.0**.
- **Maintain sharp corners** is not selected.



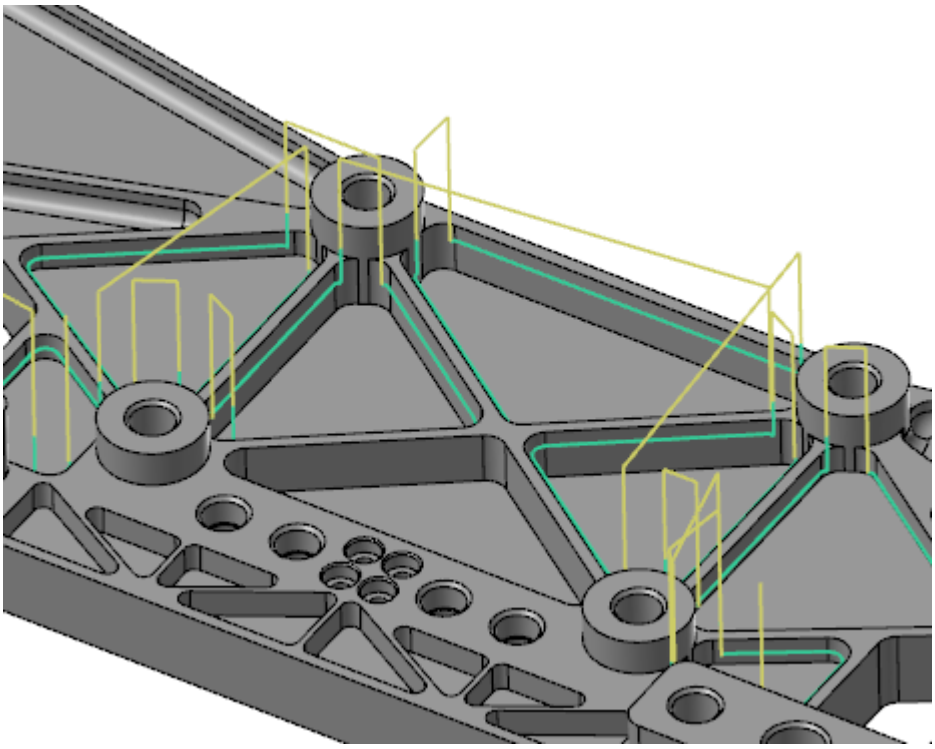
- **Roll cutter around corners** set to **Sharp**.
- **Stock to leave on walls** set to **0.175**.
- **Maintain sharp corners** is not selected.

- **Roll cutter around corners** set to **Sharp**.
- **Stock to leave on walls** set to **0.175**.
- **Maintain sharp corners** is selected.

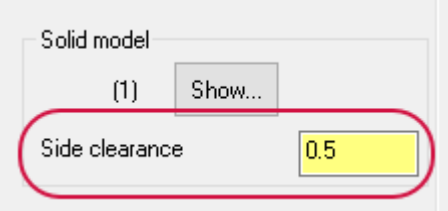
## New Model Chamfer Toolpath

A new toolpath is now available for 2D machining. The Model Chamfer toolpath allows you to machine safe horizontal chamfers on solid models. Model Chamfer only supports the following:

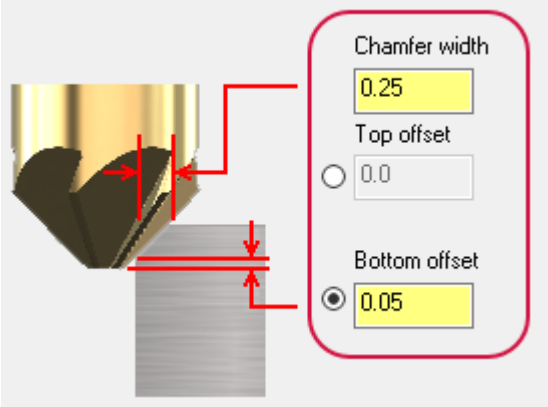
- Chamfer mill tool type
- Solid edges and faces for **Chain geometry**
- Surfaces, solids, and meshes for **Avoidance model**



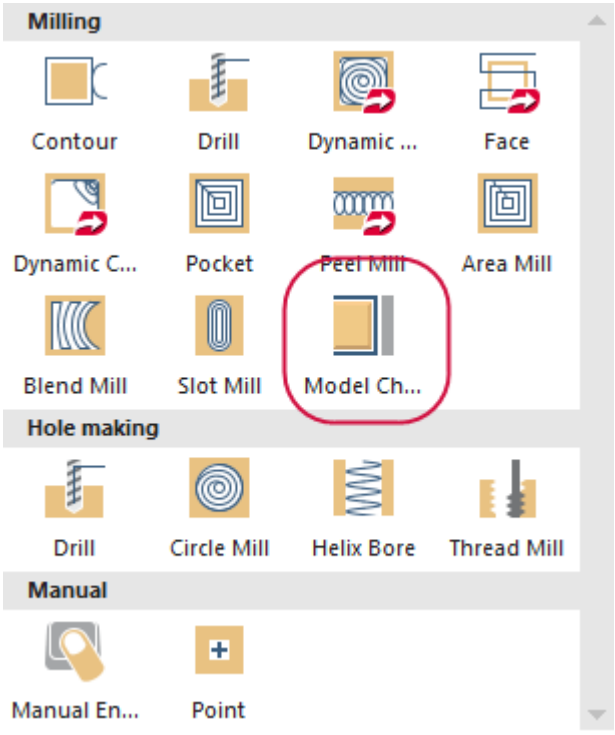
After selecting your **Chain geometry**, Model Chamfer has several unique parameters to further control the toolpath. You can enter a **Side clearance** for the **Solid model**, which is the distance away from the solid model you want the vertical part of your flute to clear. This option is set on the **Toolpath Type** page.



On the **Cut Parameters** page, you can set a **Chamfer width** value allowing Mastercam to measure the width from the chained geometry adjusted by the cut depth defined on the **Linking Parameters** page. Use **Top offset** to define how far the full diameter of the tool is above the top rail of the chamfer and use **Bottom offset** to define how far the tip of the tool is from the bottom of the chamfer.




Select **Model Chamfer** from the **2D** gallery in the **Mill Toolpaths** contextual tab.

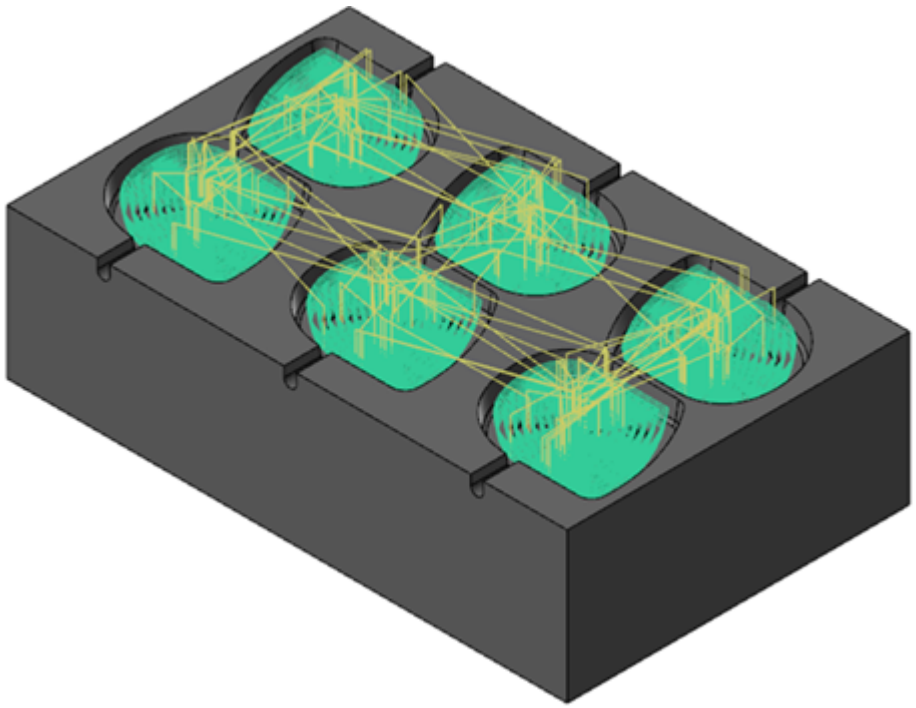


# 3D Enhancements

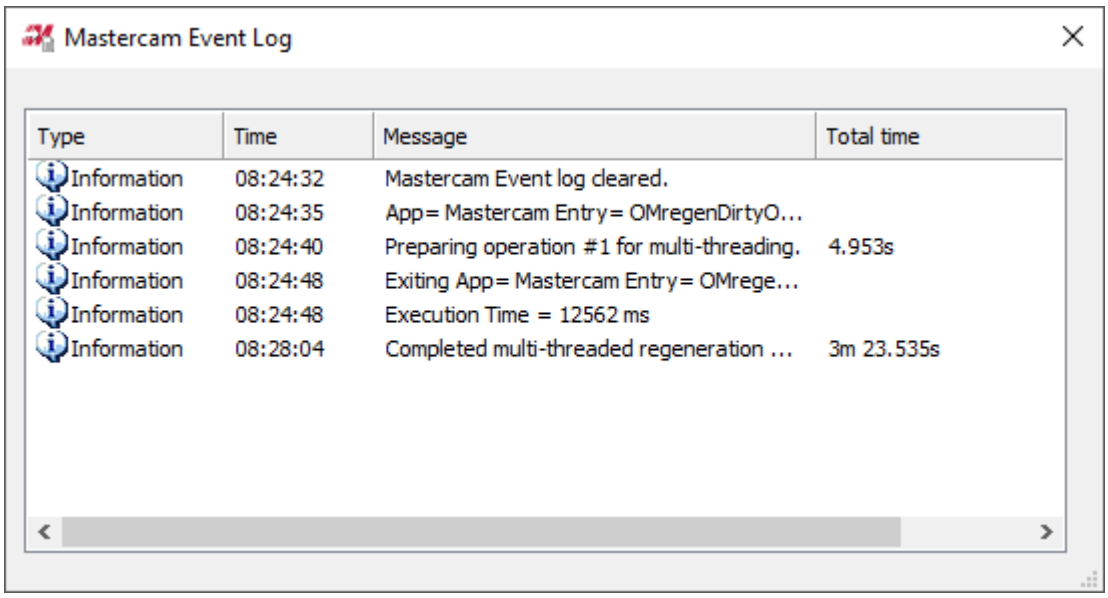
Listed below are enhancements made to 3D toolpath types, such as Hybrid.

## Area Roughing Improvements

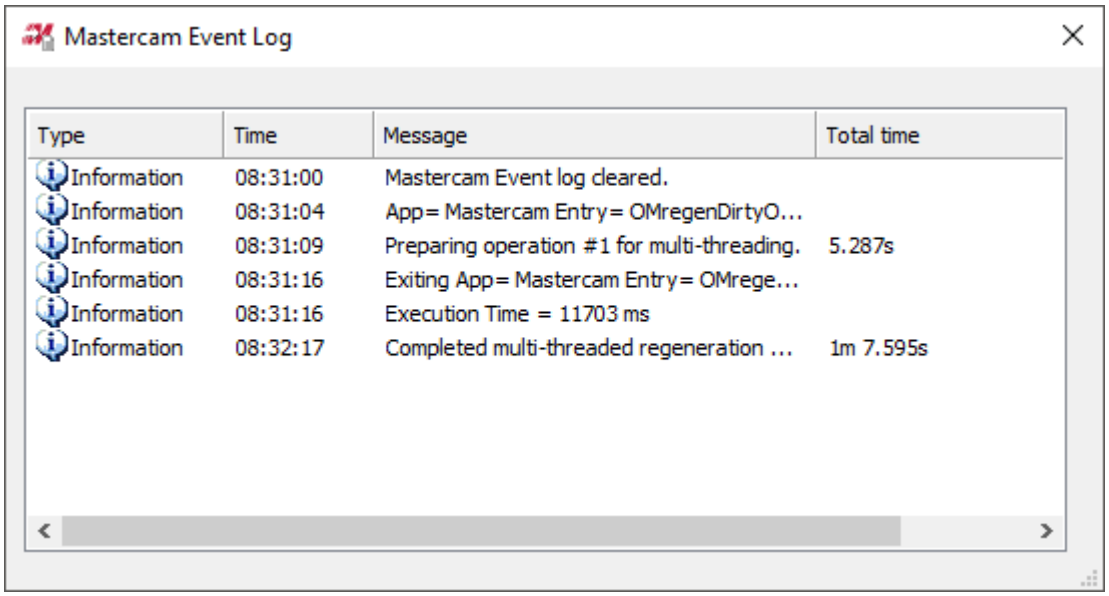
The linking processing time for Area Roughing toolpaths has improved, specifically for parts with many pockets and small stepdowns. This improves the overall processing time for high-speed machining applications that use Highfeed endmills. These tools typically use a smaller stepdown—for example, 10% of the tool diameter—which can generate many pockets to sort. You can see in the following images that the processing time for this toolpath is reduced between Mastercam 2018 and Mastercam 2019.



**Mastercam 2018**

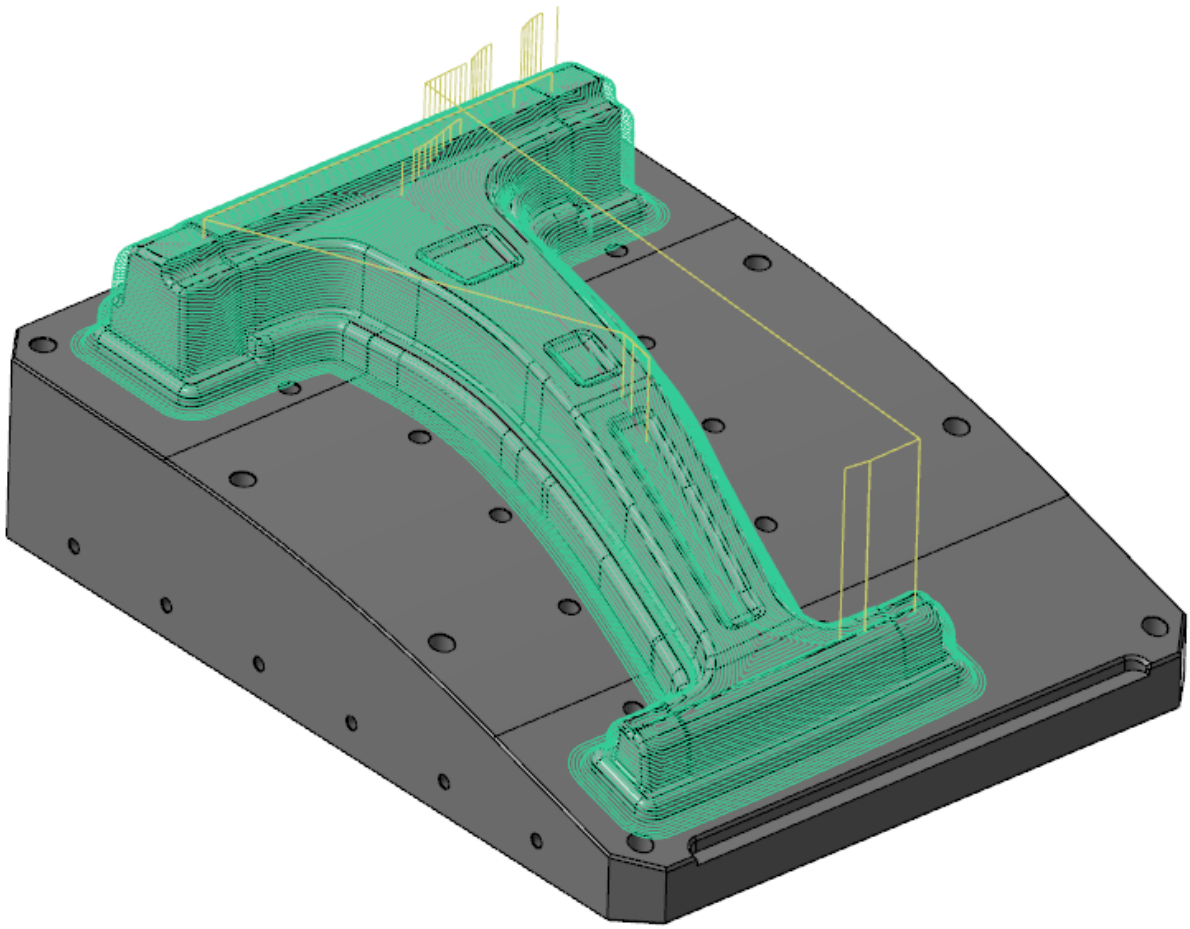


**Mastercam 2019**




## New Equal Scallop Toolpath

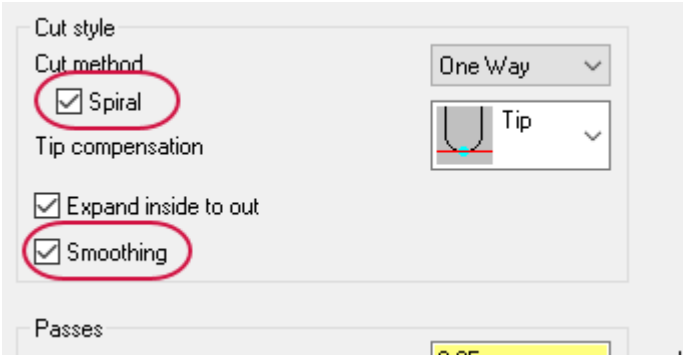
Mastercam 2019 introduces a new 3D high speed toolpath, Equal Scallop. This new toolpath creates a consistent scallop motion relative to stepover distance. Equal Scallop produces a superior surface finish compared to 3D HST Scallop.



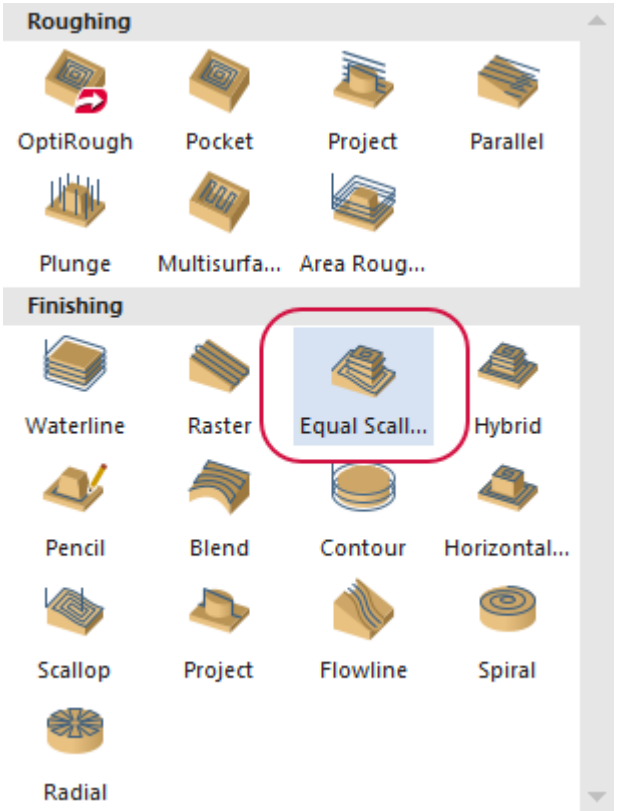
Equal Scallop generates clean, noise-free motion with the ability to smooth out sharp corners, or convert the motion to a spiral approach while avoiding stepover moves.

Equal Scallop supports avoidance geometry and **Projected boundary smoothing tolerance**, which was also introduced in this release. For more information, please read "Smoothing Tolerance" on the facing page.

There are two parameters that are specific to Equal Scallop, both available on the **Cut Parameters** page. **Spiral** eliminates stepovers when machining. **Smoothing** attempts to smooth sharp corners and replaces them with curves, resulting in a more even load on the tool.

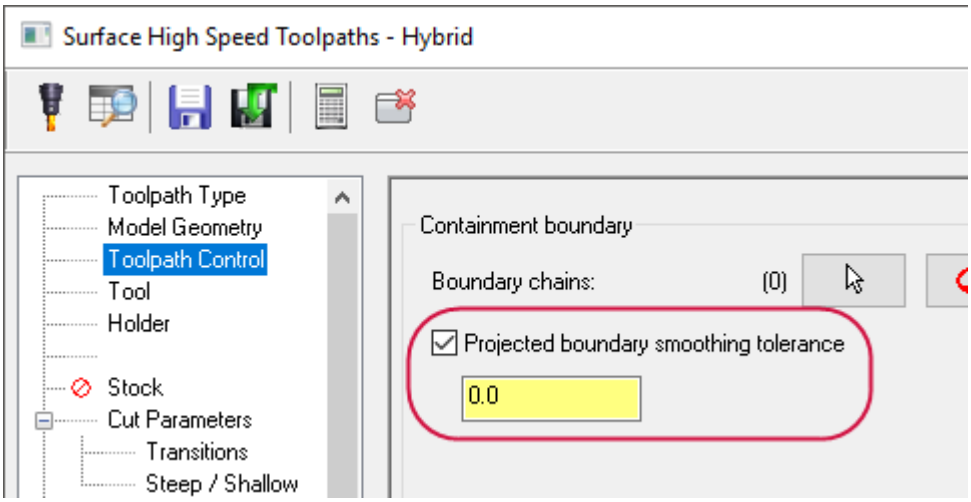



Select **Equal Scallop** from the **3D** gallery on the **Mill Toolpaths** contextual tab.



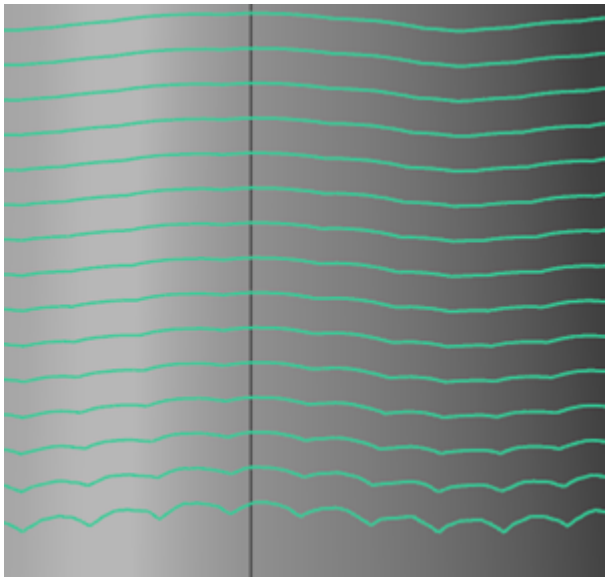
## Smoothing Tolerance

A new option, **Projected boundary smoothing tolerance**, has been added to the **Toolpath Control** page for Hybrid and Equal Scallop toolpaths.

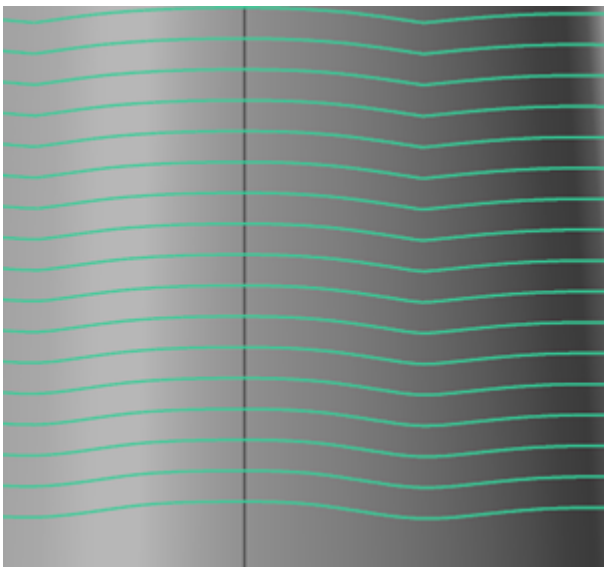



When you project a containment boundary onto the machining model, the resulting projected boundary can be jagged or noisy, especially near steep features. Use this option to allow Mastercam to improve the quality of the projected boundary.

**Without tolerance**



**With tolerance**



When you use this option with the new Equal Scallop toolpath with avoidance geometry and no containment boundary, Mastercam attempts to smooth the border between the machining and avoidance geometry if it is jagged or noisy.

### Transitions Feed Rate

You can now set which feed rate is used during transition moves for Waterline toolpaths on the **Transitions** page.

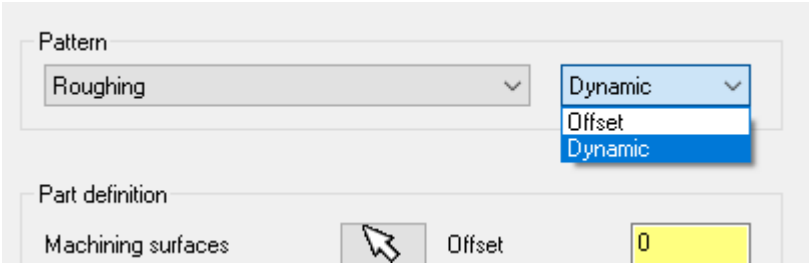
- **Plunge rate**: Uses the **Plunge rate** set on the **Tool** page for transition moves.
- **Feed rate**: Uses the **Feed rate** set on the **Tool** page for transition moves.

## Multiaxis Enhancements

Listed below are enhancements made to Multiaxis toolpath types.

### Miscellaneous Enhancements

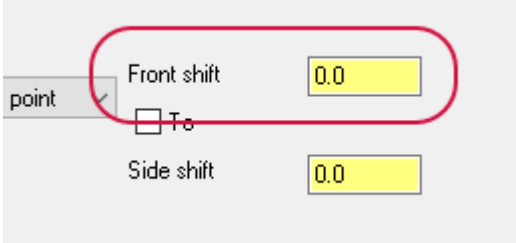
- Output from Swarf Milling toolpaths can now be locked to 4-axis, even if it deviates from the surface.
- You can now select a plane with the **Select Plane** button in the **3-axis Select Tool Plane** dialog box.
- You can now set the **Pattern** to **Dynamic** when using **Roughing** for Port Expert toolpaths. This option is available on the **Cut Pattern** page when **Output format** on the **Tool Axis control** page is set to **4 Axis**.



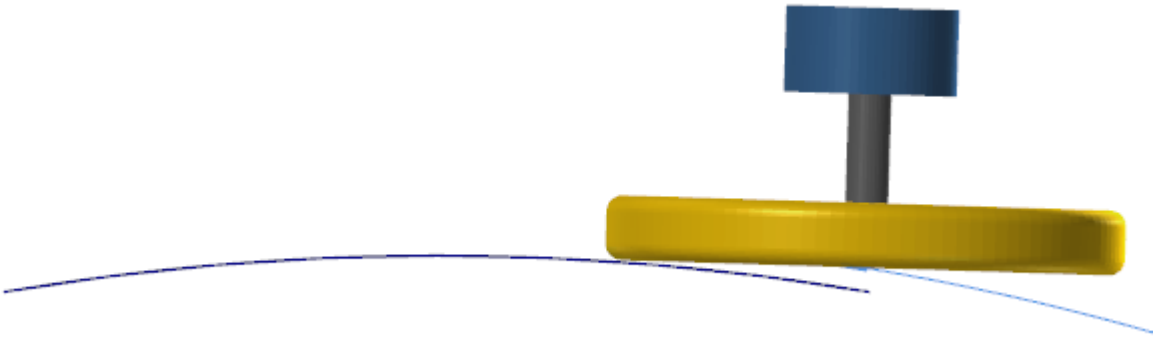

- Returned the ability to choose a named plane as a view direction.
- **Scallop Height** is now enabled for Morph toolpaths, as well as any other toolpath that supports it.

## Gradual Front Shift

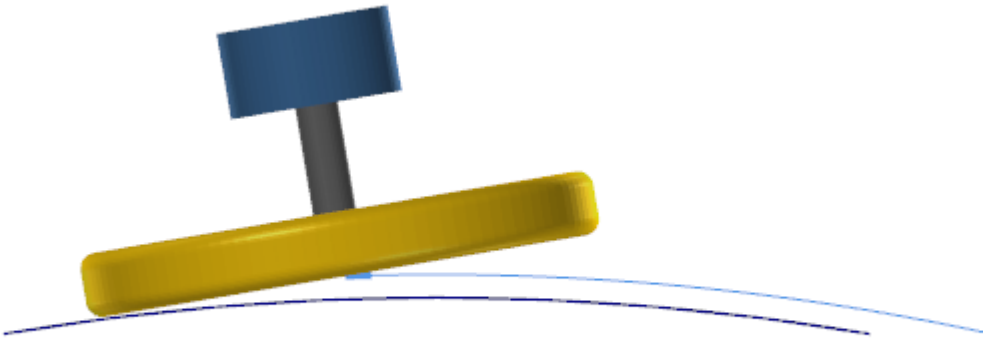
Morph and Parallel toolpaths now support a gradual **Front shift**, which keeps the tool in better contact with the part without placing excess wear on the tool's leading and trailing edges. Without gradual front shift, the tool cuts across the part with the tool's edge, as shown in the following images. **Front shift** is located on the **Tool Axis Control** page.



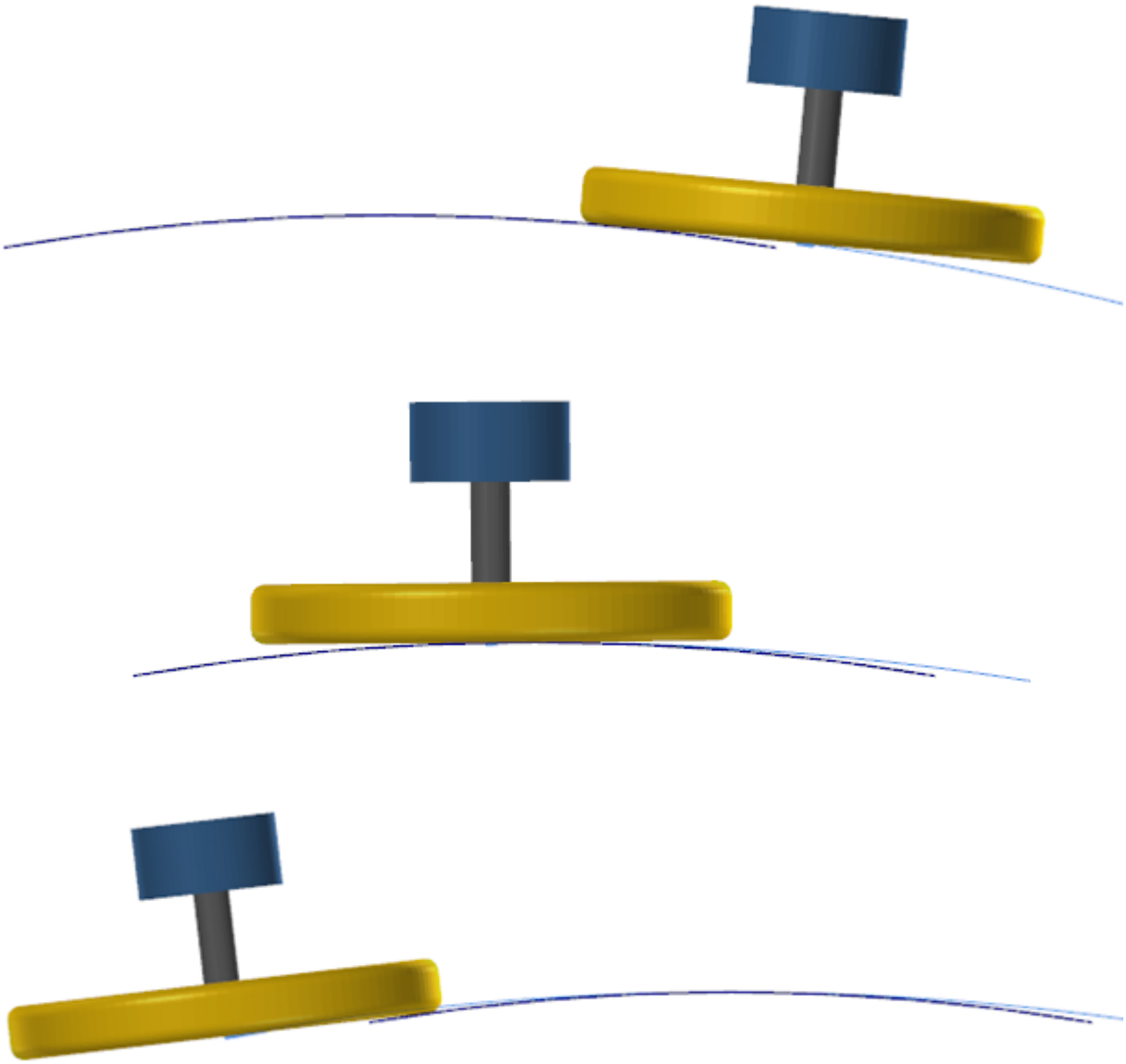

The first image shows the tool starting its cut on the edge of the tool.



The second image shows that the tool has cut across the entire arc with that edge.



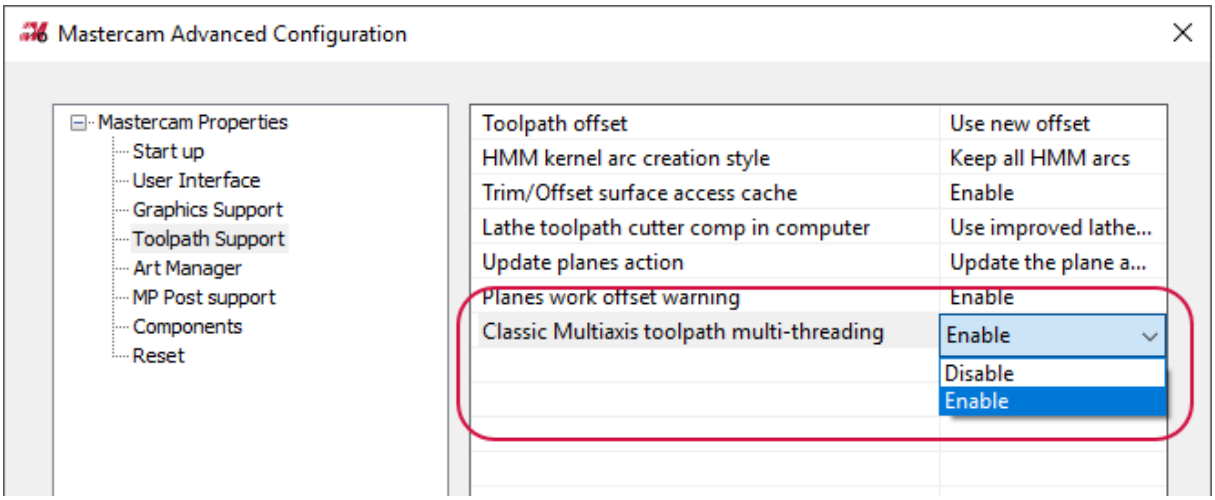
The images below show the same cut using gradual front shift. The first image shows the tool entering the cut, the second image shows the tool shifted to use the center of the tool, and the third image shows the tool near the end of its cut, with the opposite edge of the tool cutting.



## Multi-Threading

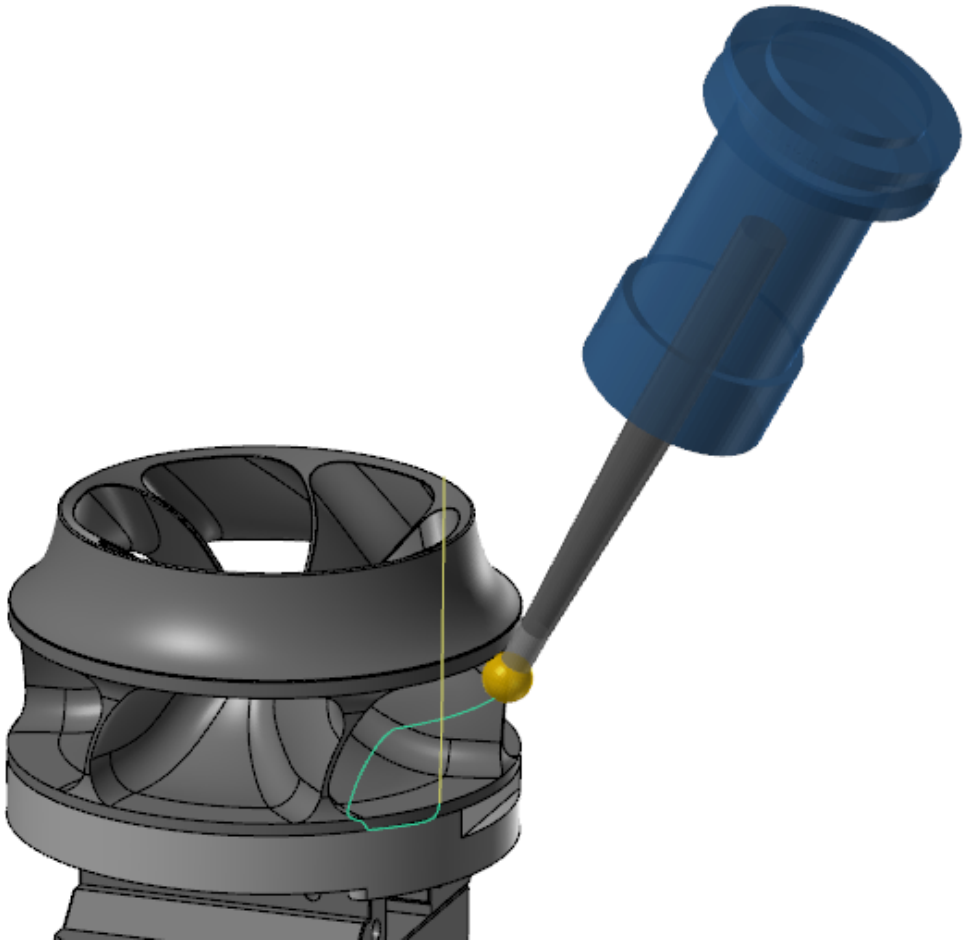
Curve, Swarf, Flow, Multisurface, Port, and Rotary toolpaths are now multi-threaded, which lets Mastercam operate while the toolpaths regenerate. You may experience speed improvements in multiple and offset cut calculations.

If necessary, multi-threading can be turned off by selecting **Disable** for **Classic Multiaxis toolpath multithreading** on the **Toolpath Support** page in the **Mastercam Advanced Configuration** dialog box.




## New Deburr Toolpath

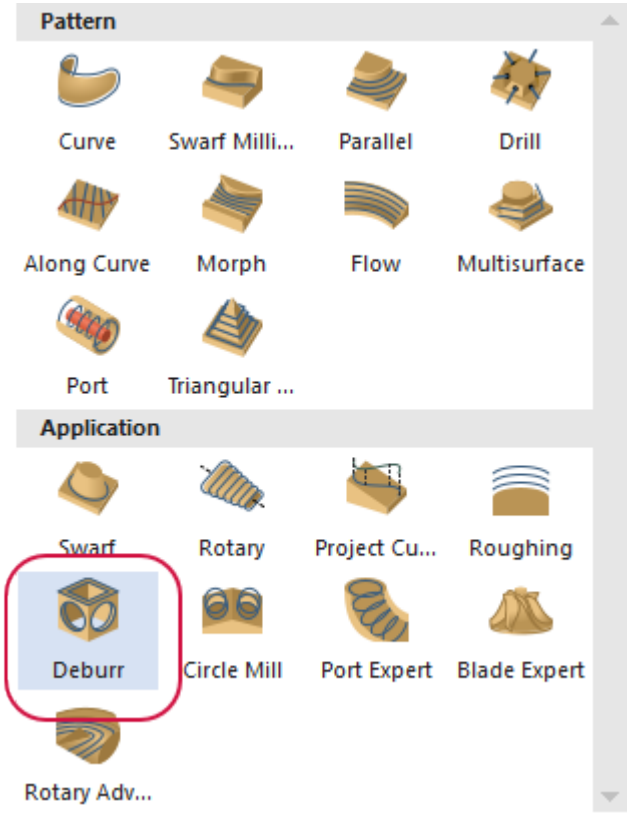
A new toolpath is now available for Multiaxis. Deburr is used to break edges for 3-to-5 axis, and to remove burrs.



The Deburr toolpath can be used with the following tools:

- Ball end mill
- Lollipop with undercut

Select **Deburr** from the **Multiaxis** gallery in the **Mill Toolpaths** contextual tab.

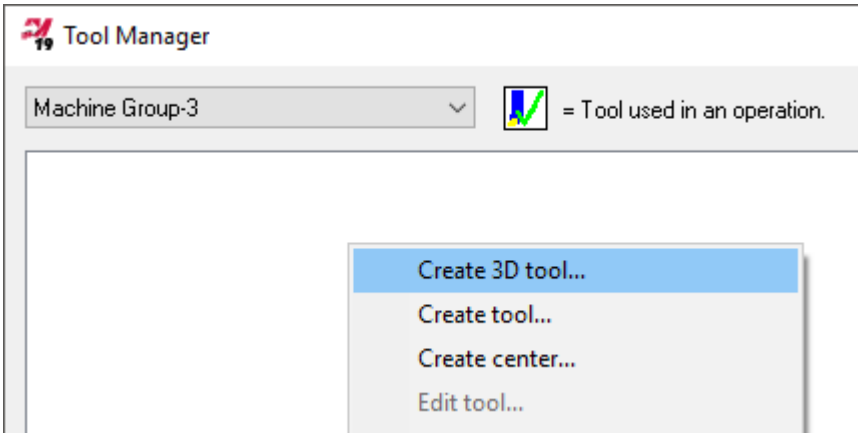


# LATHE ENHANCEMENTS

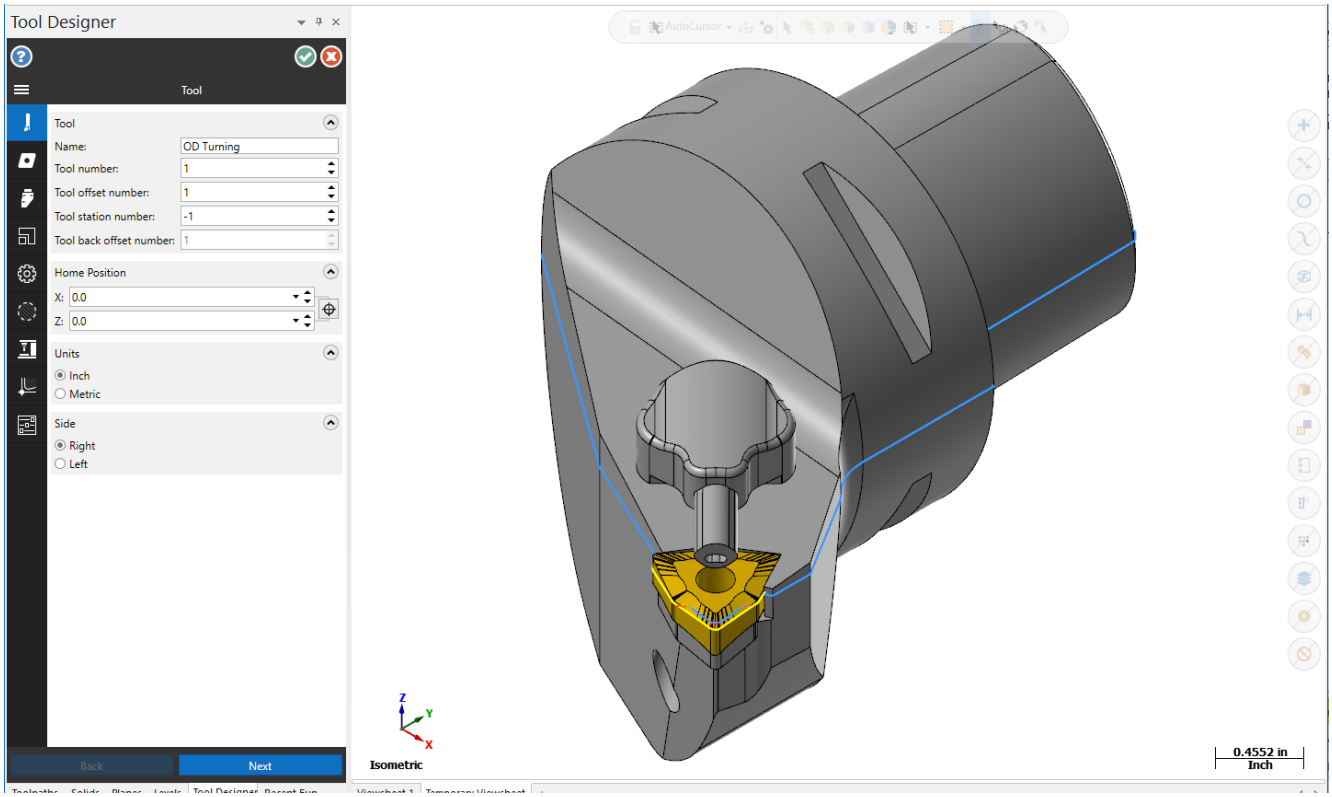
Listed below are Lathe enhancements made to Mastercam 2019.

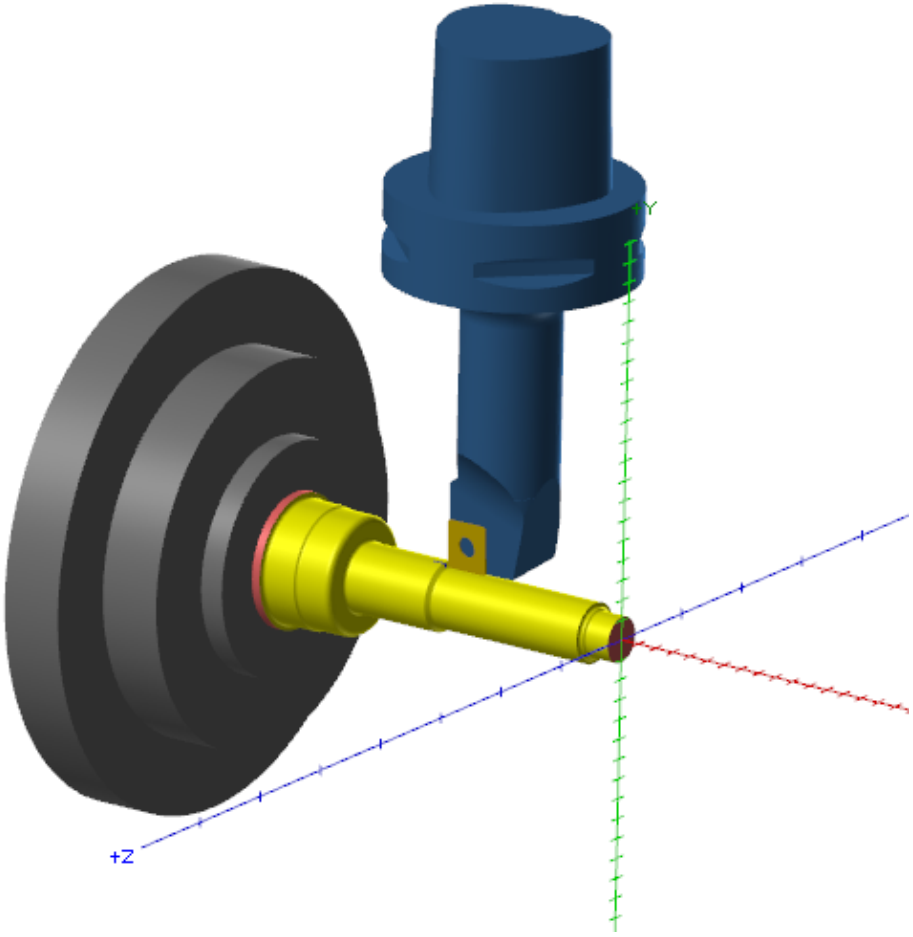
## 3D Tool Support

Mastercam now allows you to build tools from 3D STEP models, using the new **Tool Designer**. To create a 3D tool, select **Create 3D tool** from the right-click menu in the **Lathe Tool Manager** dialog box.




The **Tool Designer** is a function panel with tab-style navigation, providing a structured workflow that is similar to using a wizard. You can define tools, assign them to operations, and view them when running Classic Backplot and Mastercam Simulator.






## Improved Support for Cross-Centerline Turning

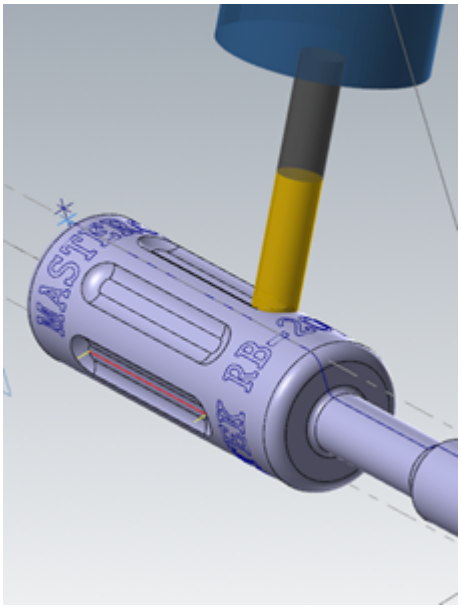
Support for cross-centerline turning has improved. When you select a tool or turret that is across the centerline from the chained geometry, Mastercam reverses the spindle direction of the operation. Earlier versions of Mastercam required you to artificially mount the tool in the opposite turret. This is no longer necessary.

Mastercam also includes new validation routines that prevent you from creating a cross-centerline toolpath in situations where it does not make sense, such as for a pinch-turning toolpath.

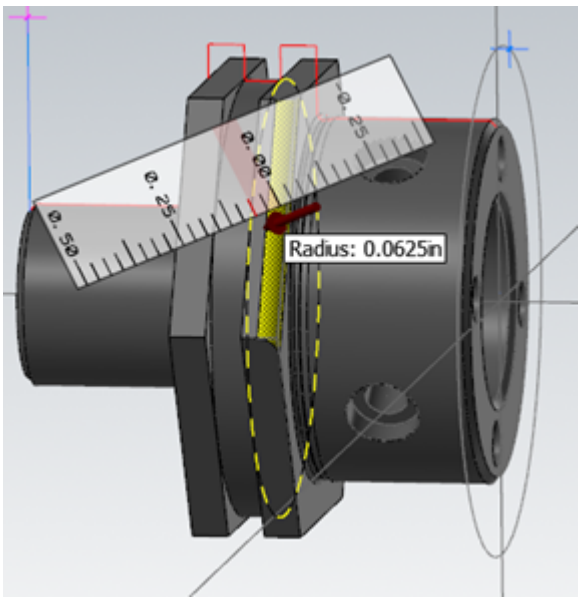
## Mastercam Lathe for Swiss Machines

**Note**: Lathe for Swiss Machines is only available in the United States.

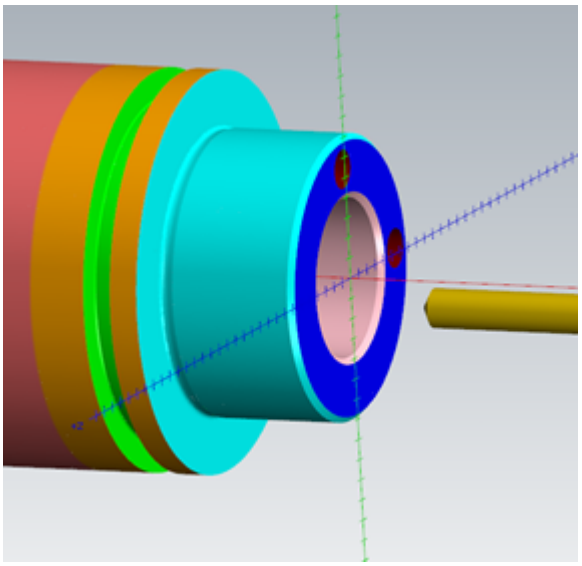
Mastercam 2019 now supports Swiss machines through its familiar Lathe solution. Swiss users with active Lathe and Mill/Mill 3D licenses can now take advantage of familiar Lathe and Mill toolpaths. Mastercam provides milling, grooving, and engraving strategies using milling tools.



A wide variety of post processors are available to support Mastercam Lathe for Swiss. These post processors are created and maintained by our Mastercam partners, Postability, Inc. and In-House Solutions, Inc. Currently there are posts for many of the 5-7 gang-style Swiss machines now available in the Unites States market.

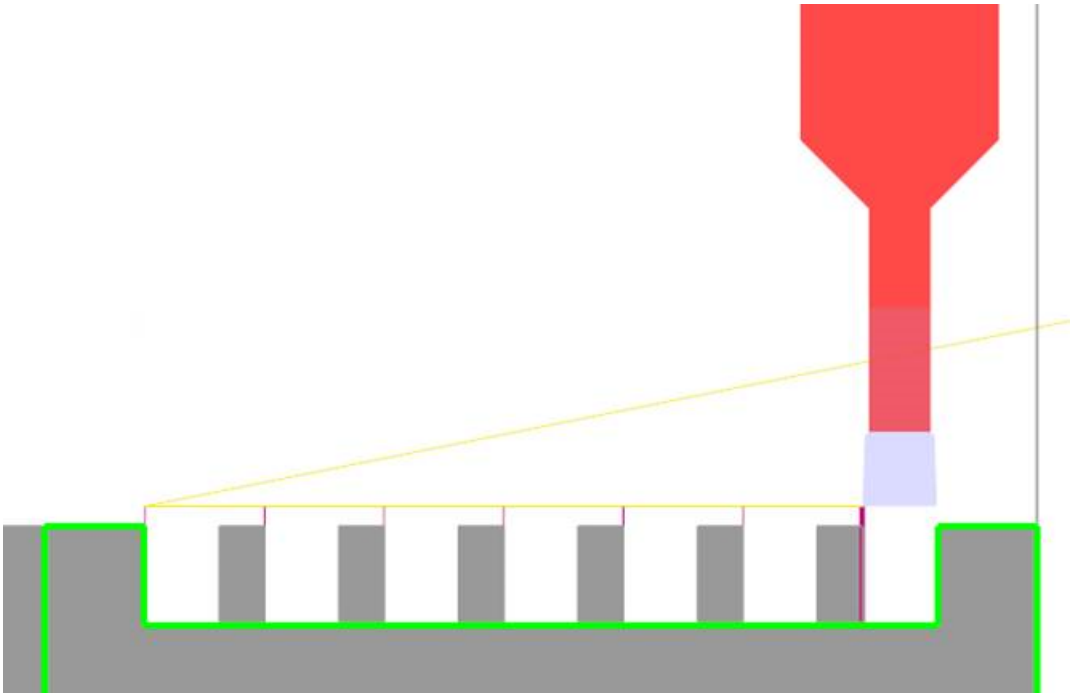


After creating your part, you can use Mastercam Simulator and Classic Backplot to simulate the toolpaths.



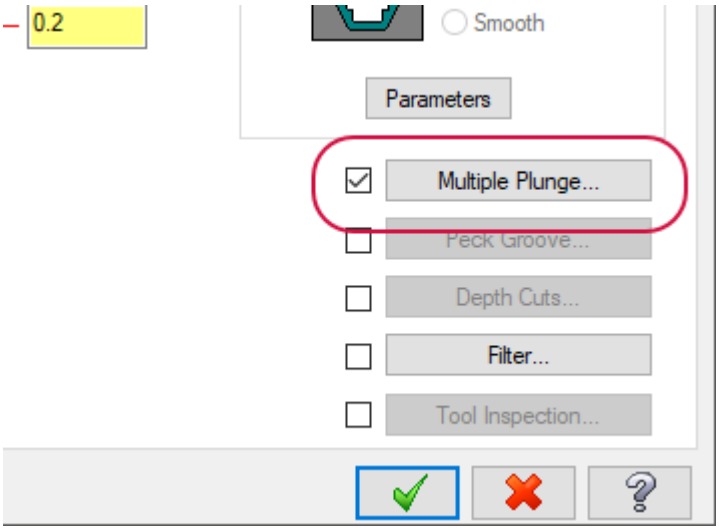
## Multiple Plunge

The Lathe Groove toolpath includes a **Multiple Plunge** option, allowing you to rough out a groove with rib cuts, as shown below:

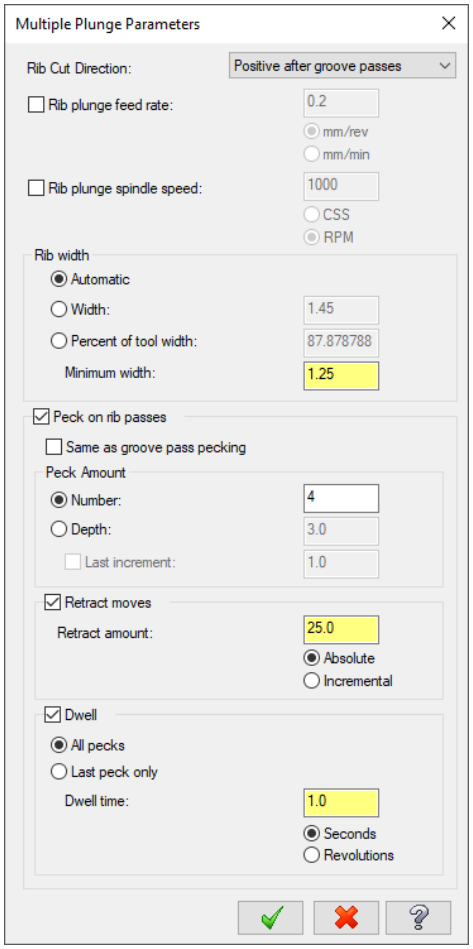


This is useful when you are concerned about tool deflection. Consistent tool pressure can result in better chip control, and more even tool wear. **Multiple Plunge** also gives you the option to machine the ribs with a more aggressive feed rate than the initial plunges.

To activate the new options, check and select **Multiple Plunge** on the **Groove rough parameters** tab.

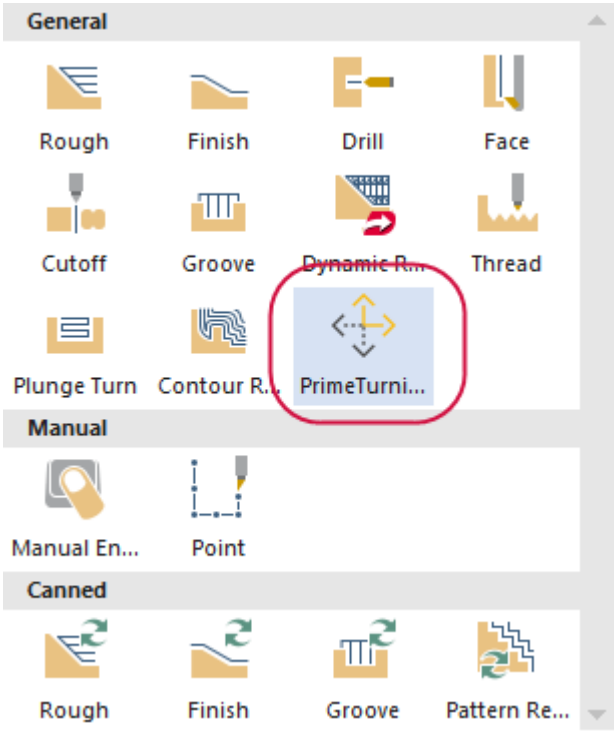



Use the **Multiple Plunge Parameters** dialog box to configure the rib cuts, such as the direction, plunge feed rate, plunge spindle speed, or the rib width.



# PrimeTurning™ Toolpath

The PrimeTurning toolpath, which was introduced in Mastercam 2018 as an Add-In, is now fully integrated into Mastercam. Select **PrimeTurning** on the **Lathe Turning** or **Mill-Turn Turning** contextual tab.




Mastercam has partnered with Sandvik Coromant on the development and support of their innovative PrimeTurning™ method and CoroTurn® Prime tooling. With these tools and methods, Mastercam allows you to turn in conventional turning and PrimeTurning directions. When combined with either CoroTurn Prime A-type or B-type inserts, PrimeTurning provides superior metal removal rate, productivity gains of over 50%, and increased tool life in a variety of materials.

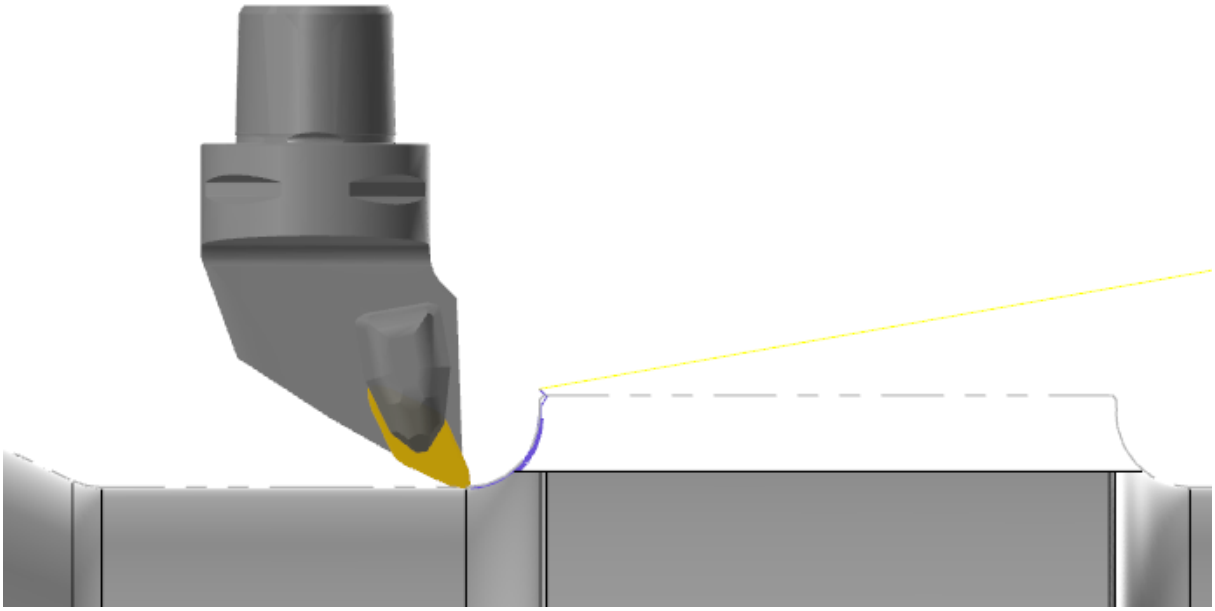
PrimeTurning has high metal removal rates that are the result of running the CoroTurn inserts at double the speed and feed, compared to conventional turning strategies. These inserts utilize a chip thinning design that reduces temperature at the leading edge and away from the tool nose. You experience greater chip control and increased tool life, requiring fewer tool changes and production stops.

## Improved Tool Libraries for PrimeTurning

The CoroTurn Prime tool libraries for PrimeTurning now include Sandvik's 0.4mm radius inserts. The libraries are installed with Mastercam 2019, and can also be downloaded from the Tech Exchange (https://community.mastercam.com/techexchange) on Mastercam.com.

## 3D Tool Libraries for PrimeTurning

With the introduction of 3D tool support ("3D Tool Support" on page 75), two 3D tool libraries are available for PrimeTurning. One library is in inch, the other library is in metric. These libraries can be downloaded from the Tech Exchange (https://community.mastercam.com/techexchange) on Mastercam.com.

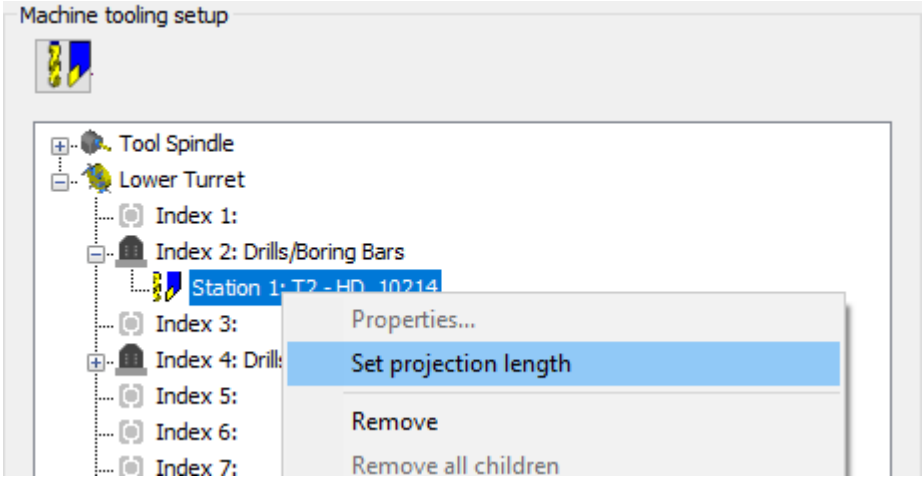


# MILL-TURN ENHANCEMENTS

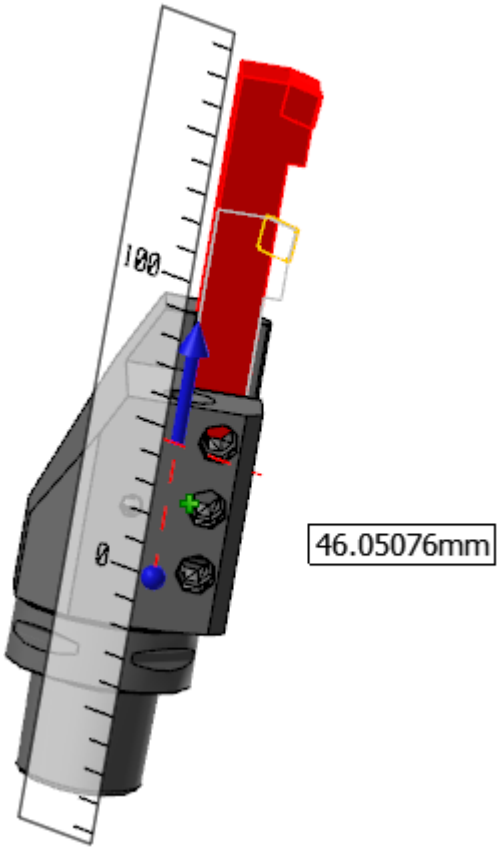
Listed below are Mill-Turn enhancements made to Mastercam 2019.

## Adjust Tool Projection

After loading tools, you now have the ability to adjust the tool's projection length by dragging the tool in the graphics window. In the Tool Setup Manager, right-click on the tool and select **Set projection length**.

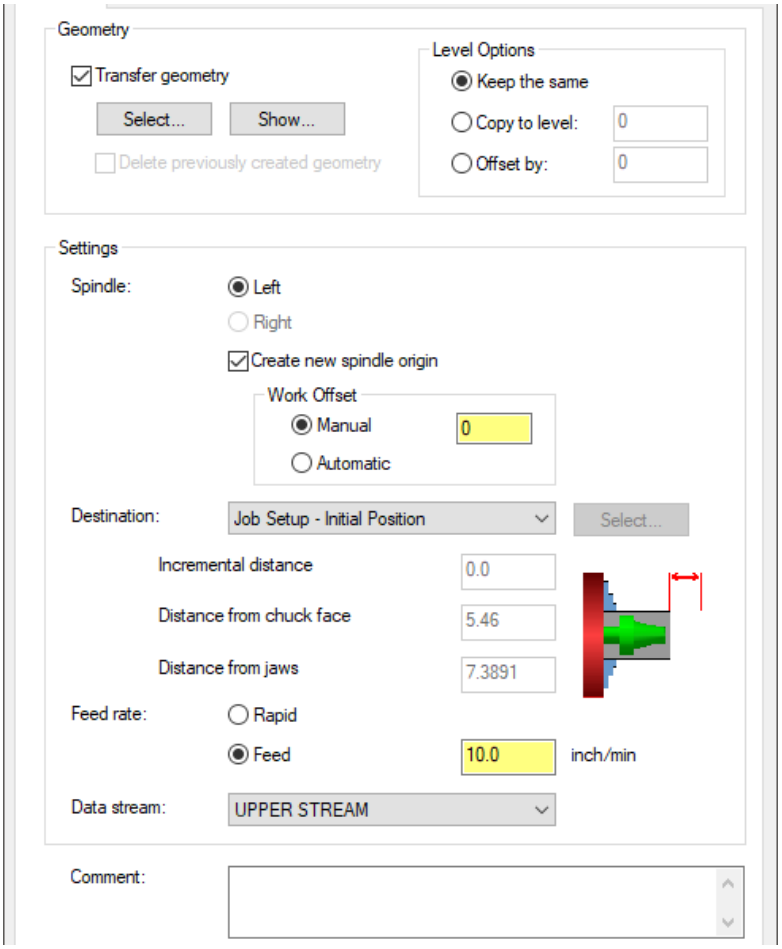



Mastercam displays the tool and its tool locator component in the graphics window. Drag the tool to the desired location.



## Bar Feed Enhancements

The Bar Feed toolpath has been improved:




- It is now supported in Simulation.
- There are new options to help manage geometry. These include selecting geometry to move with the Bar Feed toolpath, specifying a level for the transferred geometry, and the option to delete the original geometry.
- You can select either spindle, if it is supported by the machine.
- There are new options to update the spindle origin, create new planes based on the new spindle origin, or update the work offset.
- There are new options for specifying the destination and motion rate of the bar feed.

## Import and Export Operations

You can import operations from Lathe and Mill machine groups into Mill-Turn machine groups. In previous Mastercam versions, you could only import operations from other Mill-Turn machine groups.

In addition, you can copy and paste, or drag and drop operations, from Mill and Lathe machine groups. Mastercam includes validation routines to ensure that the imported or copied operations are compatible with the Mill-Turn machine definition.

## Improved Tool Loading

The tool loading algorithm has been improved in Mastercam 2019. This is especially helpful when loading tools for multi-station tool locators that support machining on either spindle. This also includes:

Turret tool locators can now be set to a specific angular orientation. In Mastercam 2018, this was only supported for spindle tool locators.

## Improved Work Offset Support

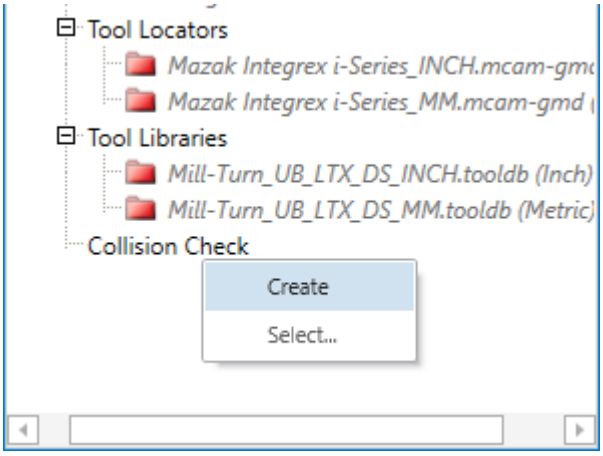
Mill-Turn now supports the **Automatic** option for work offsets in the same way as other Mastercam products. You can select **Automatic** as the **Work offset** when creating a toolpath, and Mastercam will search for the next available offset.

## New Options for Managing Collision Detection Files

Mastercam 2019 improves the way collision detection files (`.collision`) files are managed in Mill-Turn `.machine` files. You now have the option of saving a `.collision` file in a `.machine` file, or have Mastercam generate one when Simulation is starting.

- Saving a `.collision` file in your `.machine` file is recommended when you or your Reseller have customized the `.collision` file.
- Not saving a `.collision` file is recommended if you want to ensure that Simulation uses the most up-to-date `.collision` file. For example, if changes have been made to your machine definition.

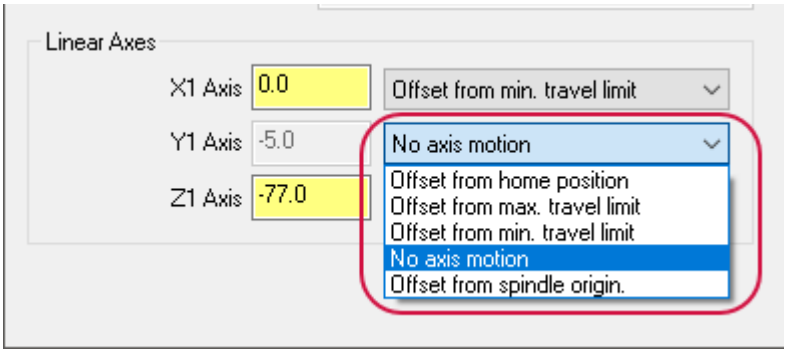
New right-click menu options in Code Expert let you managed `.collision` files. You can choose to create a `.collision` file from the current machine definition or select an existing `.collision` file, and add it to your `.machine` file.




Mastercam removes the `.collision` files from existing 2018 .machine files when they are migrated to 2019. If you want to keep the `.collision` file from your existing 2018 machine, make a copy of the file and use the right-click menu options shown above to add it to your 2019 `.machine` file after it has been migrated.

## Reference Positions

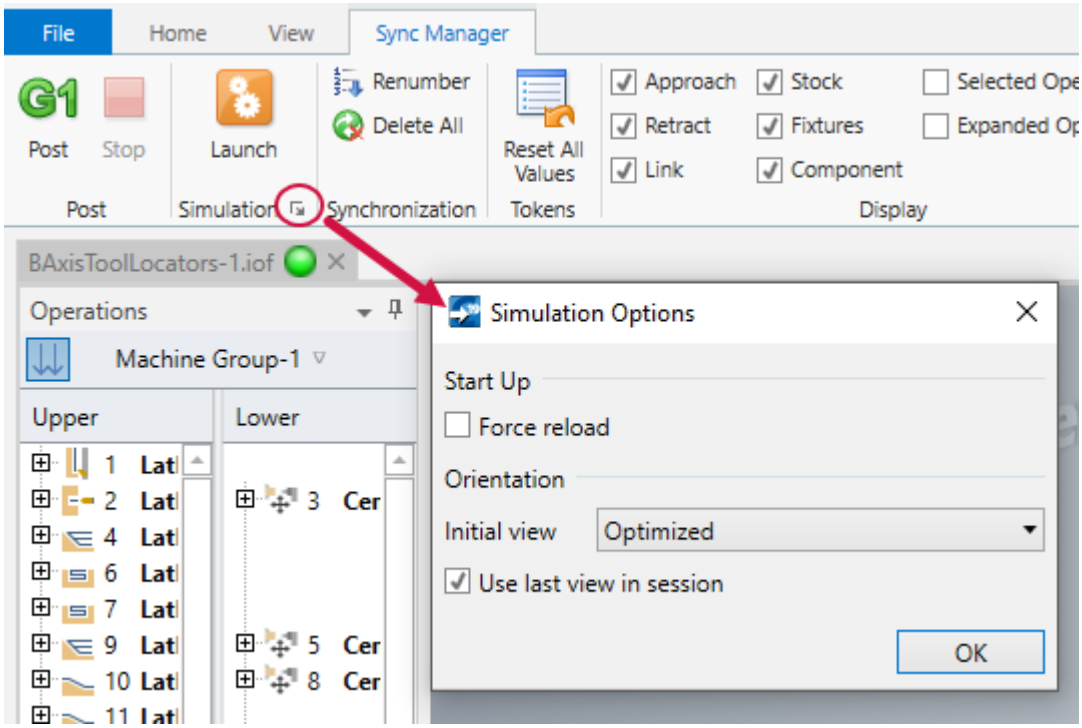
The Machine Reference Position Manager now includes a **No axis motion** option for linear axes that suppresses NC output for that axis.




For example, you can create a turret park reference position that only outputs an X-axis coordinate.

## Simulation Enhancements

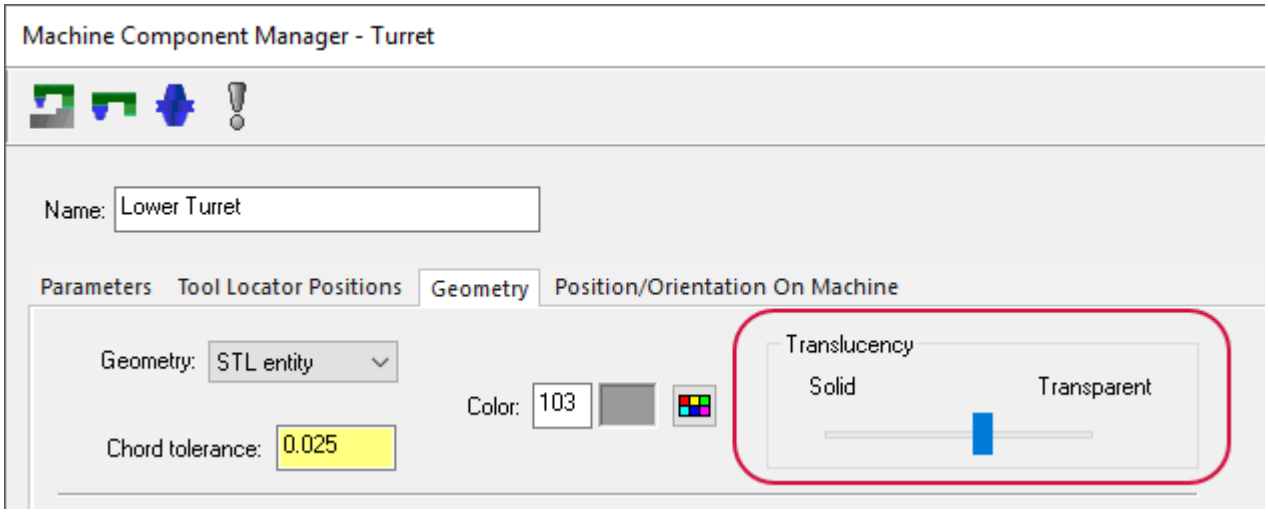
New options let you configure the **Initial view** when Simulation is launched. You can choose from:




- One of Mastercam's standard views.
- An **Optimized** view, which adjusts the Front view for a cleaner view of the work envelope.
- The most recent view from the last time you ran Simulation in the current session.
- **Force reload** causes Mastercam to reload the machine models and other supporting files when Simulation is launched. This ensures Simulation uses the most recent models if changes have been made to the .machine file.

These options can also be set in the **Application Options** dialog box, on the **Simulation** page under **Sync Manager**.

Simulation also now reads the **Translucency** that is stored in the machine definition for each component.



# POST ENHANCEMENTS

Listed below are enhancements made to posts.

## Cascading Postblocks

MP now supports cascading postblocks. This technique lets several postblock labels share the same set of postlines, so that a call to any of the postblock labels results in the same set of postlines being executed. This can help you create code that is more compact, modular, and reusable.

To define cascading postblocks, simply create postblock labels on consecutive lines with no indent. After the final postblock label, add the postlines with your processing logic.

This can be an effective technique whenever you have a postblock definition that does nothing except call a related postblock. For example, our `MPFAN.pst` post includes a series of "_2" postblocks that are called for additional points in a drill cycle. However, most of these do nothing except call **pdrill_2$**:

```
# Additional Holes
pdrill_2$ #Canned Drill Cycle, additional points
    pdrlcommonb
    pcan1, pbld, n$, pxout, pyout, pzout, pcout, prdrlout, feed, strcantext, e$
    pcom_movea
ppeck_2$ #Canned Peck Drill Cycle
    pdrill_2$
pchpbrk_2$ #Canned Chip Break Cycle
    pdrill_2$
ptap_2$ #Canned Tap Cycle
    pdrill_2$
pbore1_2$ #Canned Bore #1 Cycle
    pdrill_2$
pbore2_2$ #Canned Bore #2 Cycle
    pdrill_2$
pmisc1_2$ #Canned Fine Bore (shift) Cycle
    pdrill_2$
```

Using cascading postblocks, you can write the same logic as:

```
# Additional Holes
pdrill_2$     #Canned Drill Cycle, additional points
ppeck_2$      #Canned Peck Drill Cycle
```

```
pchpbrk_2$    #Canned Chip Break Cycle
ptap_2$       #Canned Tap Cycle
pbore1_2$     #Canned Bore #1 Cycle
pbore2_2$     #Canned Bore #2 Cycle
pmisc1_2$     #Canned Fine Bore (shift) Cycle
    pdrlcommonb
    pcan1, pbld, n$, pxout, pyout, pzout, pcout, prdrlout, feed, strcantext, e$
    pcom_movea
```

Before you can use cascading postblocks in your post, they must be enabled with the **x_mp_adv_func$** switch. This is a "license-plate" variable where each digit controls a specific advanced MP processing routine. Support for cascading postblocks is turned on by digit 2, the 10's place:

```
x_mp_adv_func$ : 10 # Enable cascading postblock support
```

The x_mp_adv_func$ switch is typically initialized immediately after the header line, before any other variables.

## NCI Updates

MP developers should note the following new NCI data for Mastercam 2019:

- A new parameter, number 20, has been added to the end of the 1016 line. A new predefined variable, `link_op$`, has been defined to store its value. It indicates that Mastercam's Safety Zone feature has been enabled for a Multiaxis Drill operation (`tool_op$` = 28).

  If the **Safety Zone** option has been selected for a Multiaxis Drill operation, the value of `link_op$` will equal 155 in the null tool change section between the drill points. In all other cases (regardless of the operation type), the value of `link_op$` will be the same as `tool_op$`.
- The following new values have been defined for `tool_op$`:
  - 73: Lathe PrimeTurning™
  - 140: Mill 2D Model Chamfer
  - 459: Multiaxis Deburr
  - 155 is reserved for use by `link_op$` for Multiaxis Drill Safety Zone operations.

Values have also been defined for the new tool types introduced in Mastercam 2019. For more information on the tools, please read "Mastercam's Accelerated Finishing™" on page 59 and "3D Tool Support" on page 75.

- The new Accelerated Finishing tool types, Taper form and Lens form, are tool types 26 and 27. This value is output on the NCI 20004 line (parameter 2) and also saved to the `tool_typ$` value (NCI 1016 parameter 2).
- 3D Lathe tools are output with the same NCI structure as existing custom tools. The value is based on the insert type. It is output on the NCI 20100 line (parameter 2) and also saved to the `tool_typ$` value.
  - 3D turning tools are output as tool type 50.
  - 3D threading tools are output as tool type 51.
  - 3D grooving/parting tools are output as tool type 52.

## New MP Function for Regular Expressions

Mastercam 2019 includes a new **regex( )** function for MP that searches a string to match a regular expression. Regular expressions let you apply patterns when searching text strings. The pattern can be as simple as a literal string, or can include wild cards, optional characters, and other expressions.

You can configure the function to either:

- Return a string that matches the expression.
- Modify the source string based on the regular expression.

The general form is:

```
string1 = regex (expression, string2, n)
```

where,

- **string1**: For modes 0 and 1, this is a string that contains the result of the operation.
  - If a match is found, this is the matching string. If no match is found, this is an empty string.
  - For mode 2, this is the input string that will be modified.
- **expression**: A regular expression enclosed in quotes or a string containing a regular expression.
- **string2**: For modes 0 and 1, this is the source string that will be searched. For mode 2, this is the replacement string that will be inserted in `string1`.
- **n**: Function mode. This is a two-digit value. The right digit has a value of 0, 1, or 2 and is required:
  - **0**: Search `string2` and test if any part matches `expression`. Return the matching substring in `string1`, or an empty string if no match.
  - **1**: Test if the entire `string2` matches `expression`. Return original string if match, or empty string if no match.
  - **2**: Replace string. If any part of `string1` matches `expression`, replace that portion of `string1` with `string2`.

  The left digit is optional. If the regular expression contains several blocks of sub-expressions within parentheses, you can use this digit to target one of the sub-expressions.

The **regex( )** function sets the helper variables `st_str_ix$` and `end_str_ix$` with the beginning and ending locations of the result string.

The following example searches a line of NC output for G1, G2, G3, or G01, G02, G03.

```
srgx : "G0?[123]" # Find G or G0 followed by either 1, 2, or 3.

sinput : "N1 G01 X1.5 Y1.25 F15."

p_mypostblock

sreturn = regex(srgx, sinput, 0)
```

The results of the function:

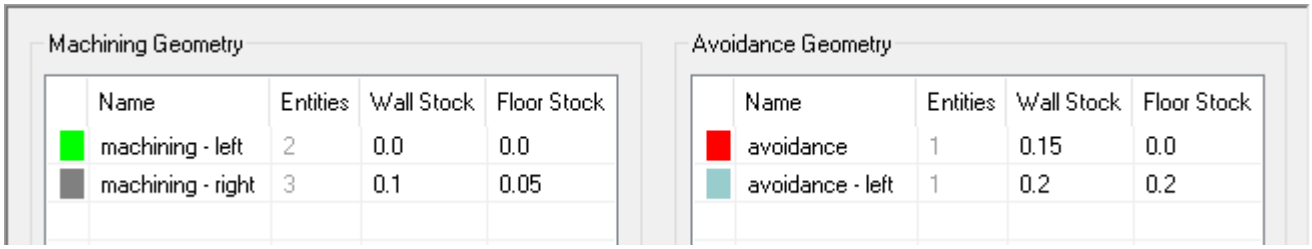
```
sreturn = "G01"

st_str_ix$ = 4
```

```
end_str_ix$ = 7
```

**Note**: The **regex( )** function will accept any regular expression.

## New Stock Function

Mastercam 2019 includes a new **stockinfo( )** function for MP. This function allows you to acquire stock model information from the **Model Geometry** page for 3D High Speed toolpaths.




The **stockinfo( )** function lets you target a specific row for your query. The first argument specifies the query mode, and the third argument specifies which row to query. The third argument can be either the name of the row or its number.

The function returns the information from the row as a string with eight items of data. Use **rpar( )** or another function to parse the string and retrieve the desired value.

The string returned by the function contains the following pieces of data:

| | |
|---|---|
| 1 | Wall stock. |
| 2 | Floor stock. |
| 3 | Number of entities. |
| 4 | Color. This number is associated with the color in the Mastercam interface. |
| 5 | The number of the row (beginning with 0 for the first row). |
| 6 | Reserved for future use. This is always 0. |
| 7 | Reserved for future use. This is always 0. |
| 8 | The name of the row. |

For example, the following string is returned when you query the second machining Geometry row pictured above:

```
0.1 0.05 3 8 1 0 0 machining - right
```

The general form is:

```
string = stockinfo(n, x, y)
```

where,

- **String**: The stock parameters encapsulated in a string.
- **n**: The function state.
  - **0**: Query drive (machining) rows by row number.
  - **1**: Query check (avoidance) rows by row number.

  - **2**: Query drive (machining) rows by name.
  - **3**: Query check (avoidance) rows by name.
- **x**: The ID of the operation whose stock settings you are querying. Use **op_id$** to query the current operation.
- **y**: Helper variable for the function state—either the number of a row or its name. Note that row numbers begin with 0, not 1.

## UTF-8 Encoding for MP Posts

In Mastercam 2018, we introduced a new XML format for post text. However, the .pst file itself is still encoded as ASCII text. With Mastercam 2019, the entire .pst (or .mcpost) file is encoded as UTF-8. This improves the support for post text in languages other than United States English. Mastercam will automatically make this conversion when you migrate or update posts from earlier versions.

## UTF-8 Encoding for NC Output

Mastercam 2019 allows you to select the character encoding scheme to use for each buffer or output stream. This includes:

- The main NC output stream
- Auxiliary output streams (**subprg$**, **auxprg$**, **extprg$**, and **lccprg$**)
- Text buffers
- NCI files

### Main NC Output Stream

A new variable, **ncprg$**, has been introduced to set the character encoding format for the main NC output stream. There are two possible values:

- **0** indicates that the NC output will be encoding as Windows code page (ANSI) text. This is the default value. It is also the only output format supported for versions of Mastercam before Mastercam 2019.
- **10** indicates that the NC output will be encoded as UTF-8 text. Do this when the control unit on your machine supports UTF-8 text.

### Auxiliary Output Stream

You can also set the character encoding scheme individually for each output stream:

- **subprg$**
- **auxprg$**
- **extprg$**
- **lccprg$**

To accomplish this, a second digit has been added to indicated character encoding.

- Set this digit to **1** for UTF-8 text.
- Set it to **0** (or omit) for Windows code page (ANSI) text. This is the default value and matches output from Mastercam before Mastercam 2019.

For example, before Mastercam 2019, you could set **subprg$** to **2** to open the sub output file in append mode. Beginning in Mastercam 2019, you can set **subprg$** to **12**, which will open the sub output file in append mode with UTF-8 text encoding.

## Buffers

In Mastercam 2019, you can select a text encoding method hen you define a string buffer. To do this, a second digit has been added to parameter 5. For example, in earlier versions of Mastercam, you could open a string buffer with the following statement:

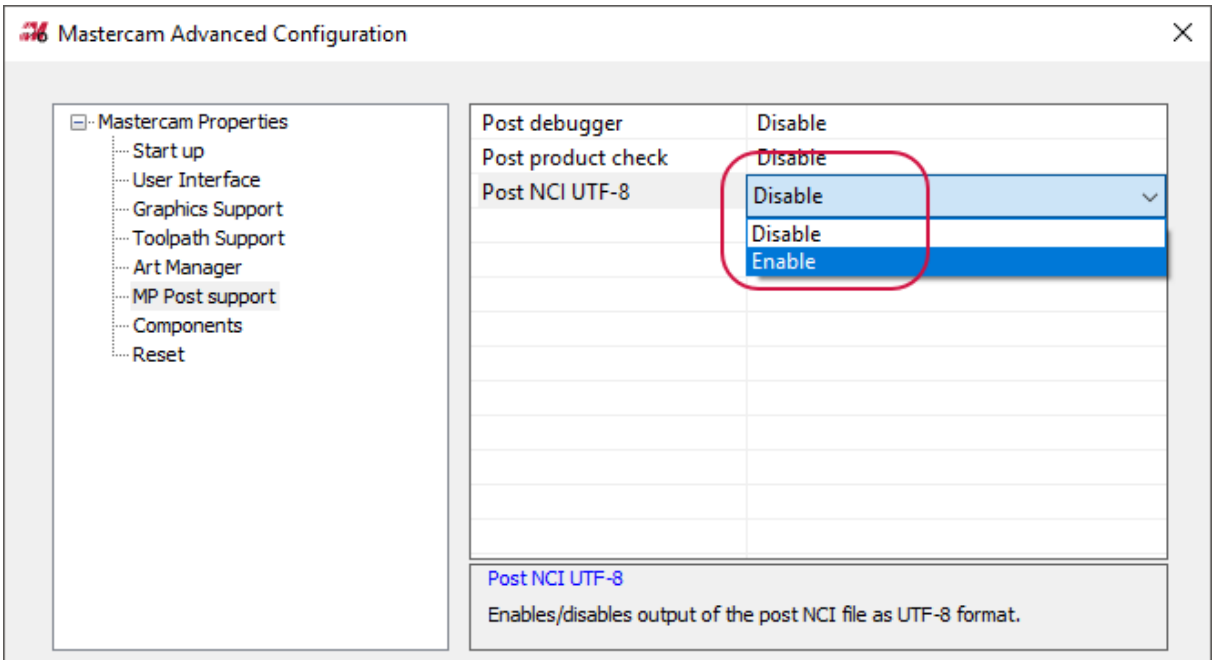
```
fbuf 4 0 2 1 1
```

In Mastercam 2019, use 11 for the final parameter to specify that the contents of the buffer will be encoded as UTF-8 text:

```
fbuf 4 0 2 1 11
```

## NCI Files

You can select UTF-8 encoding for NCI output. This option is set in the Advanced Configuration utility. If this option is not selected, the NCI file will be encoded as Windows code page (ANSI) text.



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

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

www.mastercam.com

*Mastercam® is a registered trademark of CNC Software, Inc. ©1983-2018. All rights reserved.*
*All other trademarks are property of their respective owners.*