



WHAT'S NEW IN MASTERCAM 2020

March 2019

WHAT'S NEW IN MASTERCAM 2020

March 2019

© 2019 CNC Software, Inc. – All rights reserved.

Software: Mastercam 2020

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
Masters of CAM™	9
OpenCL Warning	9
Mastercam Resources	10
Contact Us	10
General Enhancements	11
Analyze Enhancements	11
Analyze Entity	11
Check Solids	13
Volume Information for Closed Mesh Parts	14
Chaining Enhancements	14
Chaining Tangent Entities	14
Unselect and Previous	15
Solid Chaining	15
Bosses and Cavities	16
Convert to Edges and Convert to Loops	16
Face Selection Options	17
Linked Edges	18
Additional Enhancements	18
Wireframe Chaining	19
Additional Enhancements	19
Code Expert Enhancements	20
Improved Support for Multi-stream Output	20
New Search Functions	21
Contextual Ribbon Tabs	21
Deleting Referenced Entities	23

File Importers	25
Imported 3D Annotations	25
STL Exports	25
Additional File Support Enhancements	25
Graphics Support	26
Machine Definition and Component Files	26
Mastercam Folder Locations	26
Mastercam Simulator Enhancements	27
Rotary Axis Positioning	29
Merge Pattern	30
Position	30
Levels	30
Attribute Settings	31
Planes Manager	31
Persistent Plane Display	31
Find a Plane	32
Fit Origin	32
Plane Gnomon Enhancements	33
Show Gnomons	34
Selection Enhancements	35
AutoCursor® Settings Renamed Selection Settings	35
Face Center	36
Tangent Selection	36
System Configuration	37
Chain Highlighting	37
Color Options	37
Translucent View	39
Toolpaths Manager	39
Highlight Geometry	39
Dependencies	40

Additional Enhancements	40
Design Enhancements	42
Edit Surface and Edit Spline	42
Multiple Node and Control Point Selection	42
Gnomon Control	42
Curvature Comb	42
Hole Table	43
Selection	44
Display and Position	44
Reporting	44
TrueType® Fonts	44
Model Prep Enhancements	45
Add History	45
Align to Face	45
Push-Pull	45
Power Surface	46
Solids Enhancements	47
Boolean	47
Solid Hole	47
Assign Colors	47
Associativity	48
New Selection Controls	48
Top Chamfer Input	48
Wireframe Enhancements	49
Curve Enhancements	49
Curve All Edges	49
Curve Flowline	49
Curve Slice	49
Divide	50
Trim Functions	50

Additional Enhancements	50
Mill Enhancements	51
Accelerated Finishing™ Tools	51
Additional Catalog Properties	52
Hole-making Enhancements	53
Toolpath Hole Definition	54
2D Enhancements	56
Advanced Toolpath Display	56
Override Geometry Diameter	56
Spring Passes and Overlap	57
3D Enhancements	57
Curve Support	57
Dragging and Dropping Toolpath Geometry	58
New 3D Blend Toolpath	59
Offset Strategy	59
Silhouette Boundary	61
Tool Containment	62
Z Depth Enhancements	63
Multiaxis Enhancements	64
Deburr	64
Swarf Milling	64
Additional Enhancements	65
Lathe Enhancements	66
3D Tooling	66
Insert Designer	67
Holder Designer	68
Filters for Inserts and Holders	68
Improved Chuck Jaw Definitions	70
Mill-Turn Enhancements	73
Improved Machine Configuration Workflow	73

New Machine Configuration Page	73
New Tool Locator and Component Library Functionality	74
New Interface for Local Machine Definitions	76
Post Enhancements	78
MPBin Enhancements	78
MP Post Enhancements	78
Improved language support for post text	78
Improved support for UTF-8 encoding for MP posts	79

INTRODUCTION

Welcome to Mastercam 2020! Mastercam 2020 features new functionality focused on delivering speed and efficiency to your machining jobs.

Release Highlights

Listed below are some of the highlights for Mastercam 2020, including new toolpaths and enhanced functionality.

- ["New 3D Blend Toolpath" on page 59](#)
- ["Hole Table" on page 43](#)
- ["3D Tooling" on page 66](#)
- ["Filters for Inserts and Holders" on page 68](#)
- ["Improved Machine Configuration Workflow" on page 73](#)

Masters of CAM™

As the world's most widely used CAM software, Mastercam has a tremendous community of users around the globe. It is our intention to leverage that combined energy and intelligence for our mutual benefit. That is the heart of our new Masters of CAM initiative.

Over the coming weeks, months, and years, we will be collecting stories, tips, tricks, and solutions from our extended Mastercam community. This input will be available to everyone.

We will be collecting as many good ideas as we can to stay competitive with the growing market. Some ideas will help your shop's productivity. Some ideas will inspire creativity. And some ideas may direct our future software development. We know a lot of smart Mastercam users, and we are all smarter when we work together.

Our Masters of CAM program will grow in breadth and depth over time. Visit [MastersofCAM.com](https://mastersofcam.com) and see what amazing things people are making. Then share a story of your own.



OpenCL Warning

When you launch Mastercam 2020 you may receive a warning message about minimum system requirements for OpenCL™. By receiving this warning message, it means that your computer's video card does not support OpenCL.

As of Mastercam 2020 you can acknowledge the warning message, and continue running Mastercam. However, in a future Mastercam release you may be required to use a graphics card that does support OpenCL. For more information about the warning, troubleshooting, or updating your drivers, please refer to the Mastercam Knowledgebase (<https://kb.mastercam.com/>).

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 2020 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 university.mastercam.com/, or email training@mastercam.com.
- *Online Communities*—You can find a wealth of information at www.mastercam.com.
 - Follow us on Facebook (www.facebook.com/Mastercam), Twitter (twitter.com/Mastercam), and Instagram (www.instagram.com/mastercamcadcam/) for the latest tech tips and Mastercam news.
 - See Mastercam in action on YouTube (www.youtube.com/user/MastercamCadCam).
 - For more information on CNC Software, Inc., to find and apply to jobs, and connect with people using Mastercam, visit us on LinkedIn (www.linkedin.com/company/cnc-software/).
 - Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the Mastercam 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

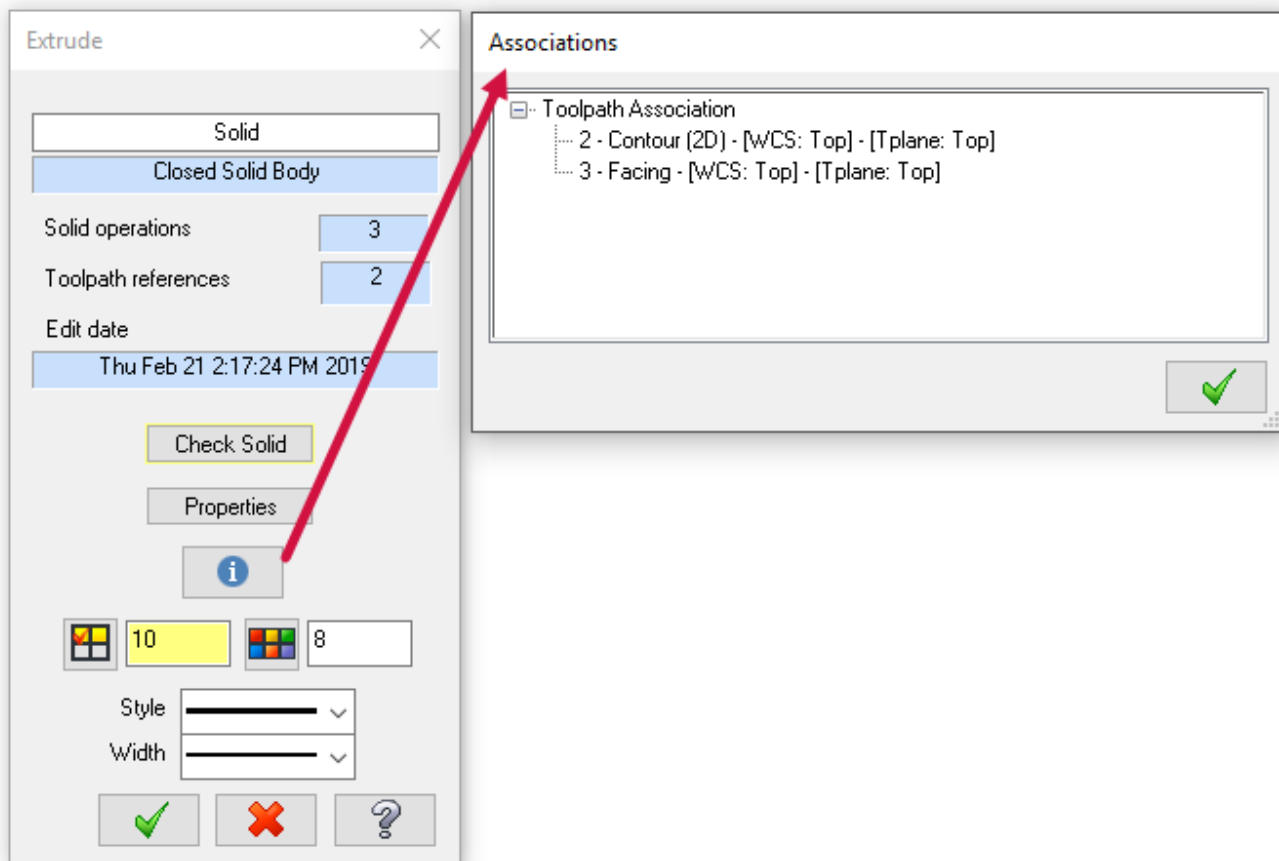
Listed below are general enhancements made to Mastercam 2020. This includes improvements to selection and other functions not specific to one product line.

Analyze Enhancements

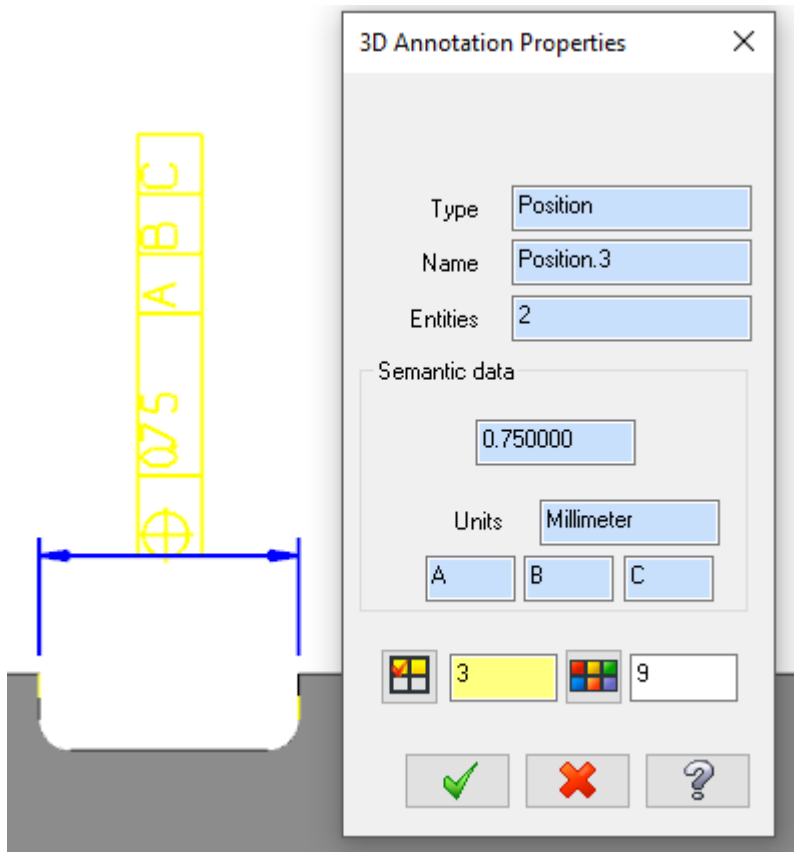
Listed below are enhancements made to the Analyze functions on the **Home** tab.

Analyze Entity

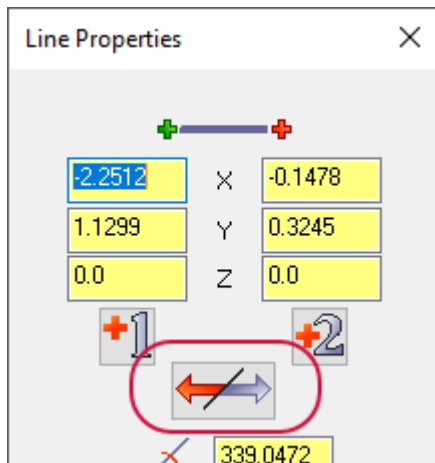
Analyze Entity now includes additional information about associated geometry and operations. Click **Association** in the **Analyze Properties** dialog box to check for association to a solid, toolpath, plane, dimension, 3D annotations, or group. This button displays if the analyzed entity has associated geometry or operations.



In previous releases, imported Model Based Definition (MBD) data displayed by Analyze Entity was limited to name, type, referenced entity, and presentation graphic information. In Mastercam 2020, Analyze Entity also includes the part's semantic data, such as tolerance values and sub-types.



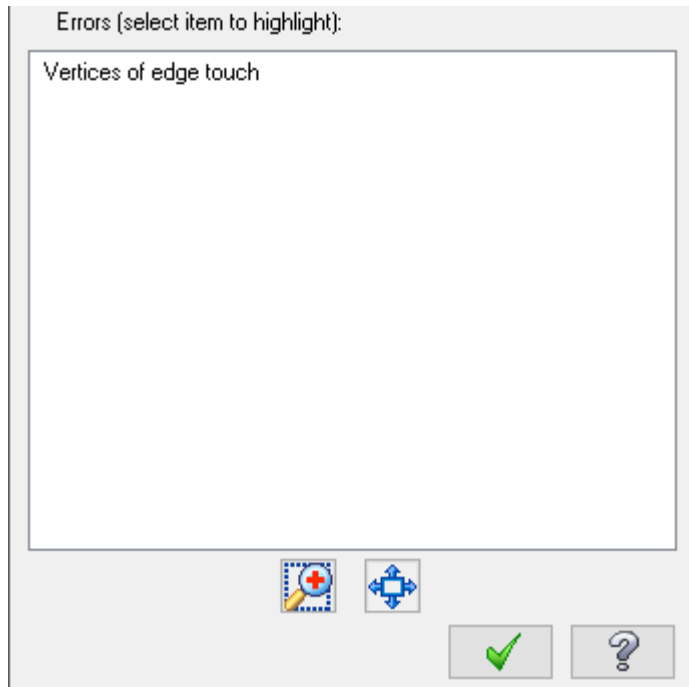
Analyze Entity also includes a **Flip** button, allowing you to reverse the start and end points of the selected line, spline, or arc.



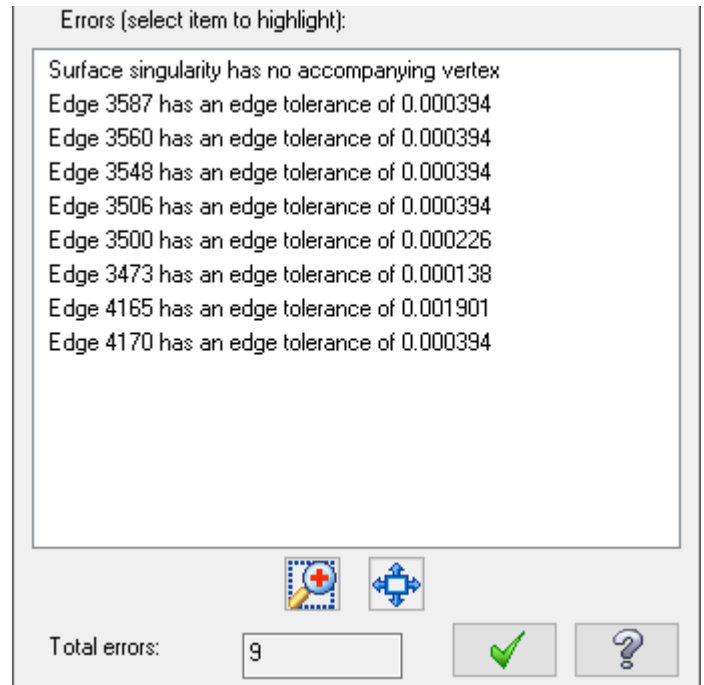
Check Solids

Check Solids will now check for tolerant edges and give a more detailed report in the **Check Solid** dialog box to help you locate any issues your part may have. The dialog box will also display a list of errors, even if not all of the entities are displayed.

Mastercam 2019

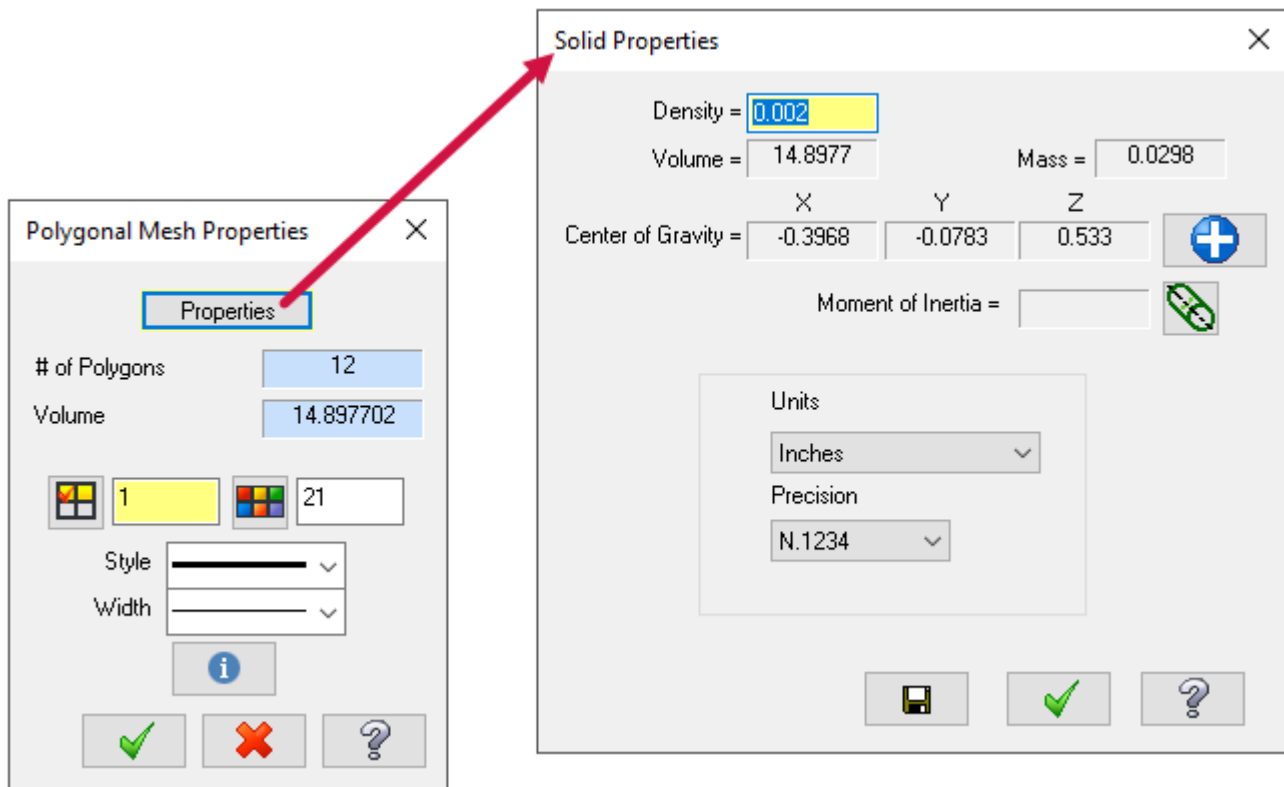


Mastercam 2020



Volume Information for Closed Mesh Parts

The **Polygonal Mesh Properties** dialog box now includes a **Properties** button. This opens the **Solid Properties** dialog box so you can view the following:



- Solid volume and mass relative to a defined density
- Center of gravity
- Moment of inertia relative to a selected axis line

Additionally, you can save solid data to a file.

Chaining Enhancements

Enhancements have been made to chaining and the **Chaining** dialog box for Mastercam 2020. These include:

- Improved **Chaining** dialog box layout for both Wireframe and Solids mode.
- Chain display and behavior improvements, including new configuration settings for chain color and line style.
- Removed the expand arrow from the upper left corner of the **Chaining** dialog box and moved the parameters to the main dialog box.

Chaining Tangent Entities

You can now automatically chain tangent entities or tangent edges by holding the **[Shift]** key while selecting an entity or edge. Chain tangent entities or edges when at a branch point by holding **[Shift]** while clicking a chain arrow, the **Previous** button, or the **Next** button. Control the angle used to determine tangency of entities using the new **Chaining tangency tolerance** option.

Unselect and Previous

The **Unselect** button now removes the last selected chain rather than the last entity of a linked edges chain when in Solid mode. **Unselect** and **Unselect all** behavior has improved during guided chaining for Wireframe mode.

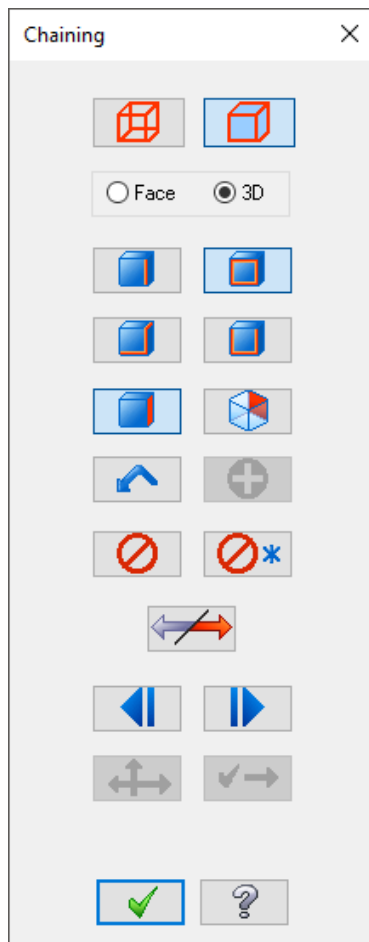
A new button, **Previous**, removes one branch or entity at a time when creating a linked edges chain in Solid mode. Clicking this button moves the endpoint of a solid edges chain to the previous branch point. In Wireframe mode, clicking the **Previous** button moves the endpoint of a chain to the previous branch point.



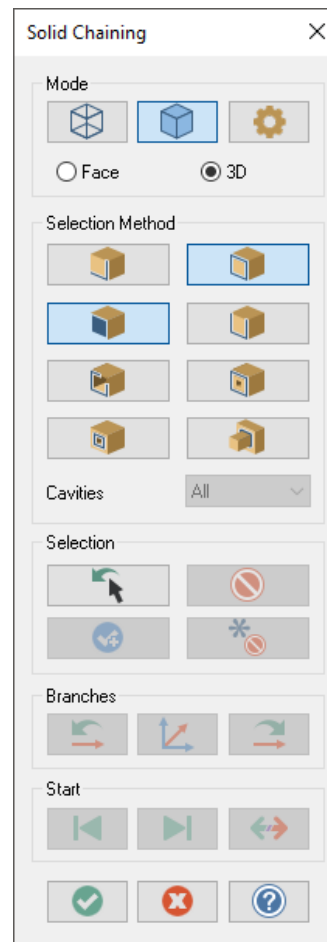
Solid Chaining

The following images depict the enhancements made to the **Chaining** dialog box when in Solid mode. Details of the enhancements follow.

Mastercam 2019

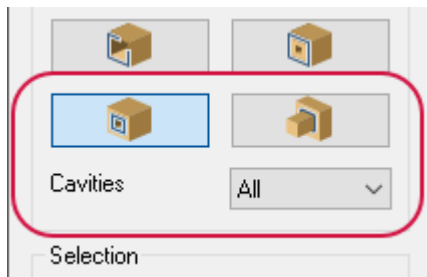


Mastercam 2020



Bosses and Cavities

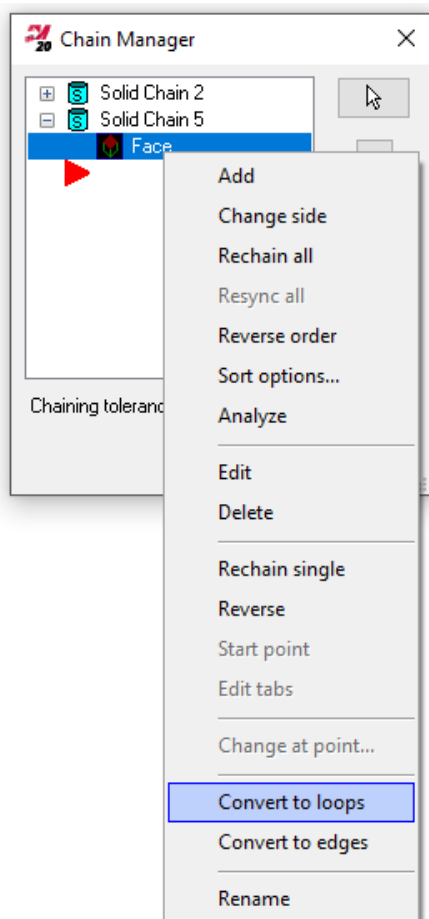
Two new selection options are available in **Chaining** dialog box:



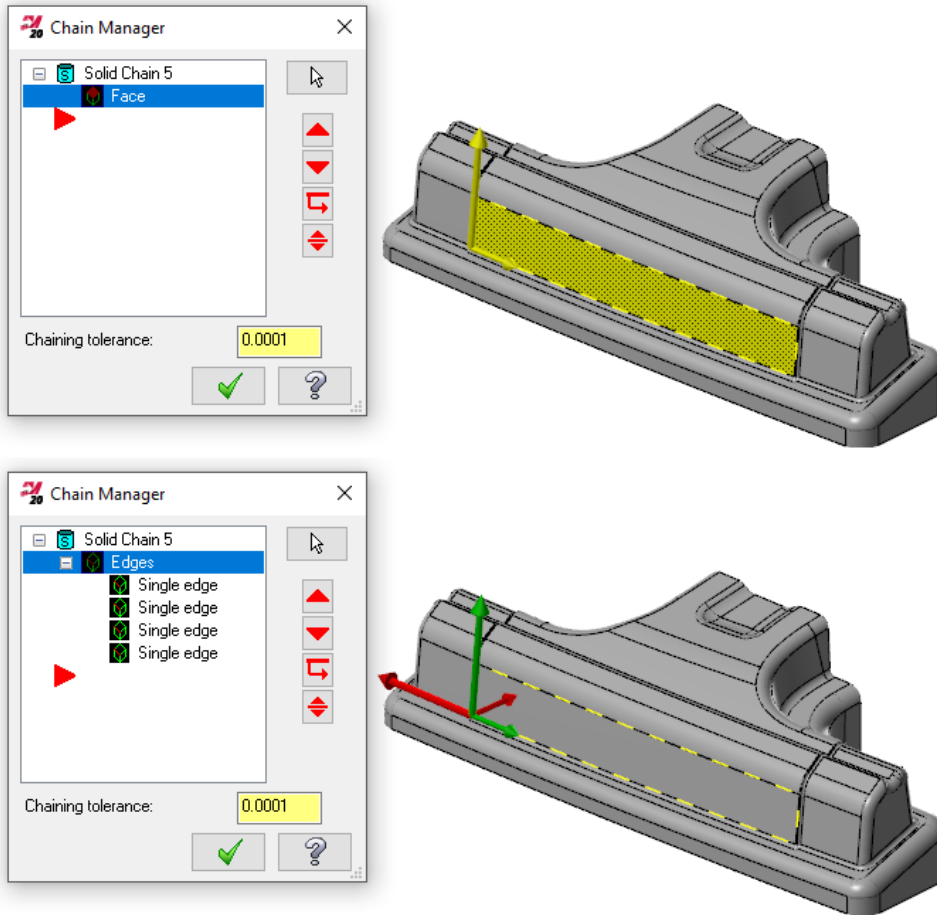
- **Bosses:** Selects only bosses on the selected face.
- **Cavities:** Selects only cavities in the selected face. Use the **Cavities** drop-down to filter by **Blind** or **Through** cavities.

Convert to Edges and Convert to Loops

Two new options, **Convert to edges** and **Convert to loops** are available when right-clicking on a loop, face, or partial loop chain for Convert to edges or a face chain for Convert to loops in the Chain Manager.



For example, after selecting **Convert to edges**, the face chain is converted to an edges chain.



NOTE

Any associativity will be lost after converting.

Face Selection Options

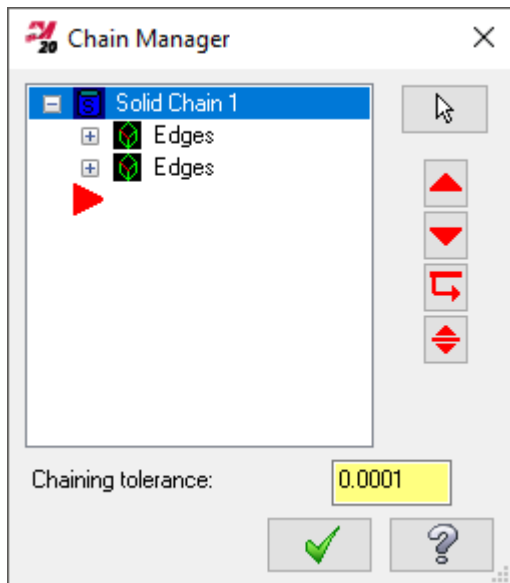
When selecting faces for solid chaining, the following shortcut keys (hotkeys) are available:

- **[Shift+click]**—Selects tangent solid faces.
- **[Alt+click]**—Selects vectors.
- **[Ctrl+click]**—Selects matching solid fillets and/or holes.
- **[Ctrl+Shift+click]**—Selects similar solid features.

You can also window select solid faces.

Linked Edges

The **Edge** option has been removed. **Linked Edges** has been moved and renamed **Edges**. The **Edges** option combines the functionality of Edge and Linked Edges. These changes also apply to the **Chain Manager** dialog box.

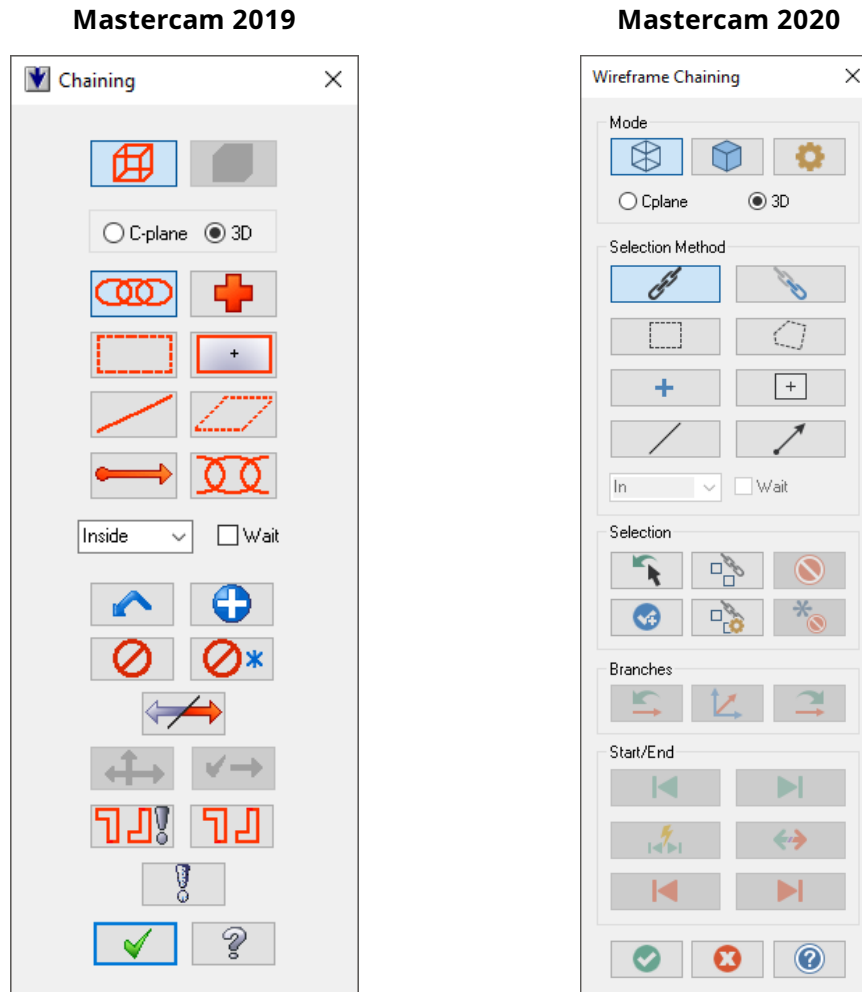


Additional Enhancements

- You can now edit solid chains in the Chain Manager by right-clicking a chain and selecting **Edit**.
- When chaining edges, you can now start a new chain by clicking an edge that is not part of the current chain.
- You can create partial loop chains across multiple solid bodies in the same chaining session.

Wireframe Chaining

The following images depict the enhancements made to the **Chaining** dialog box when in Wireframe mode. Details of the enhancements follow.



Additional Enhancements

- You can now adjust the start point for open chains.
- You can now start a new chain by clicking an entity that is not part of the current chain.
- You can create multiple disconnected chains in the same wireframe chaining session.
- The **Start** and **End** buttons are now available for Partial chaining.
- Chains with branch points now respect the cursor location for the start point.

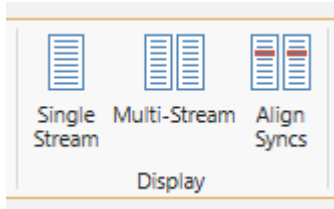
Code Expert Enhancements

Listed below are enhancements for Code Expert.

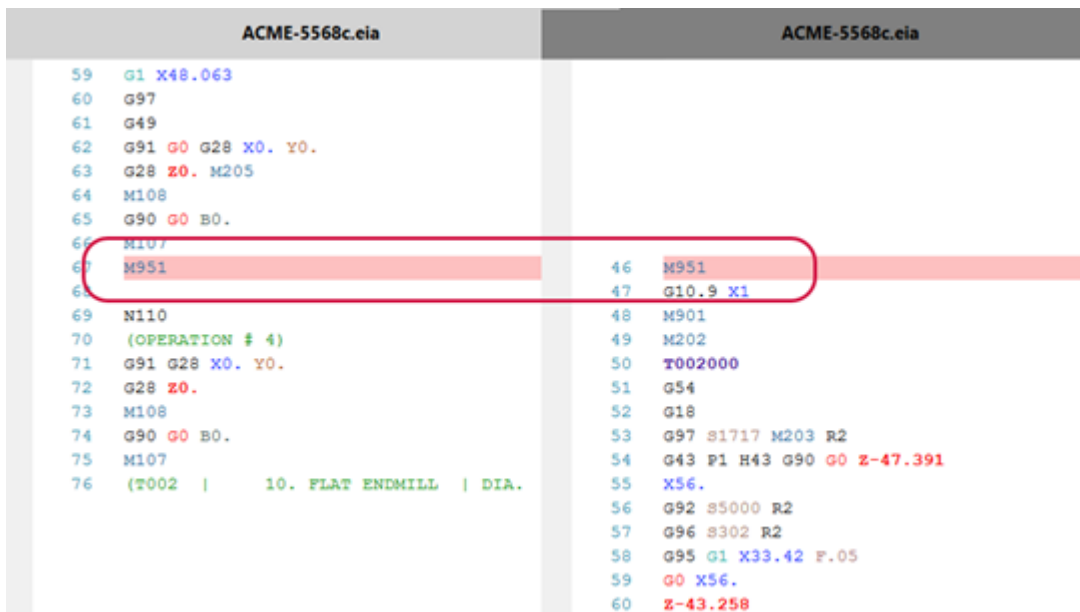
Improved Support for Multi-stream Output

Code Expert includes several new features for working with multi-stream NC files with Mill-Turn.

- New controls let you toggle between single-stream and multi-stream view without re-posting.



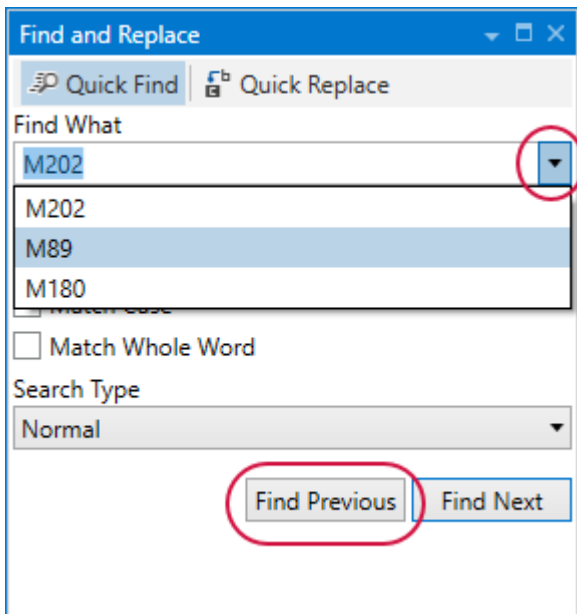
- New Align Syncs function displays multi-stream outputs with the syncs or wait codes lined up. You can also select the colors that will be used to highlight the syncs.



- New Save All function lets you save all the NC files associated with a multi-stream document instead of needing to save each file individually.

New Search Functions

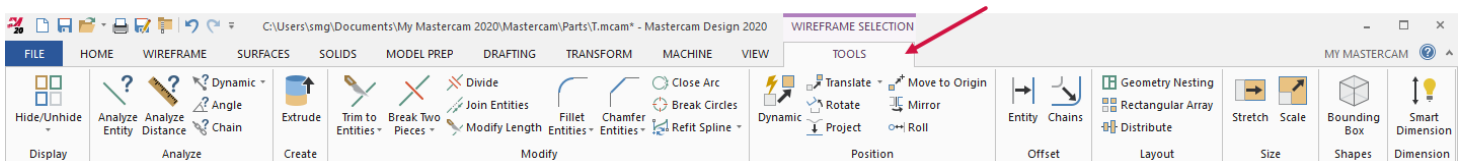
The search function in Code Expert includes two new features:



- Recall previous searches by clicking the arrow next to the search term field. The previous search terms are saved per Code Expert session.
- You can choose to search backward instead of forward by clicking the new **Find Previous** button.

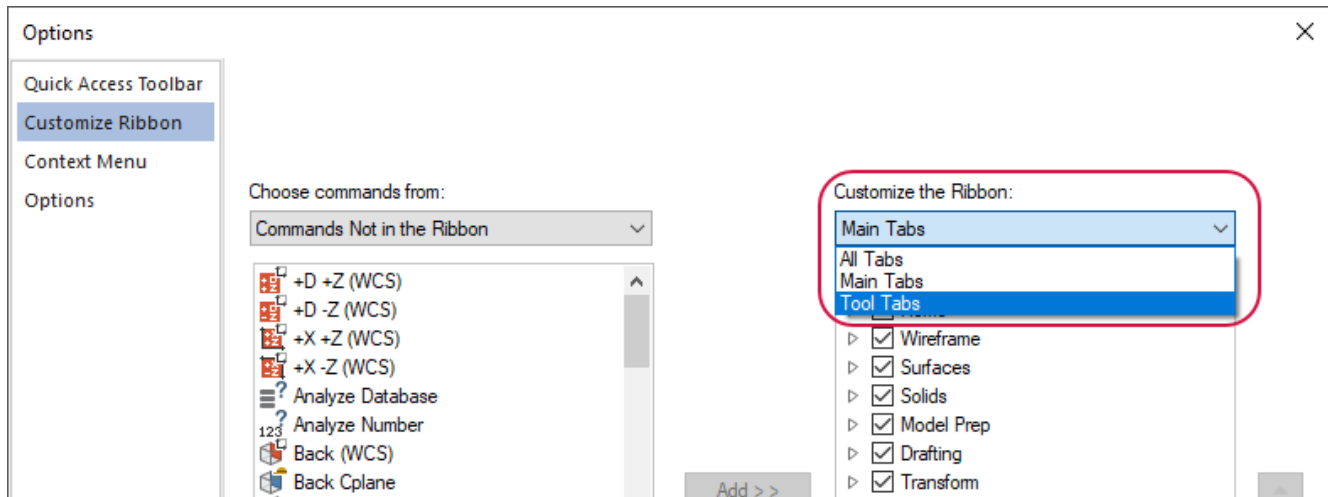
Contextual Ribbon Tabs

To speed up design changes to your parts, Mastercam 2020 includes contextual ribbon tabs that display when you select different types of geometry in the graphics window. For example, if you select only wireframe geometry, an extra ribbon tab titled **Wireframe Selection** displays tools that you might need for analyzing or editing.

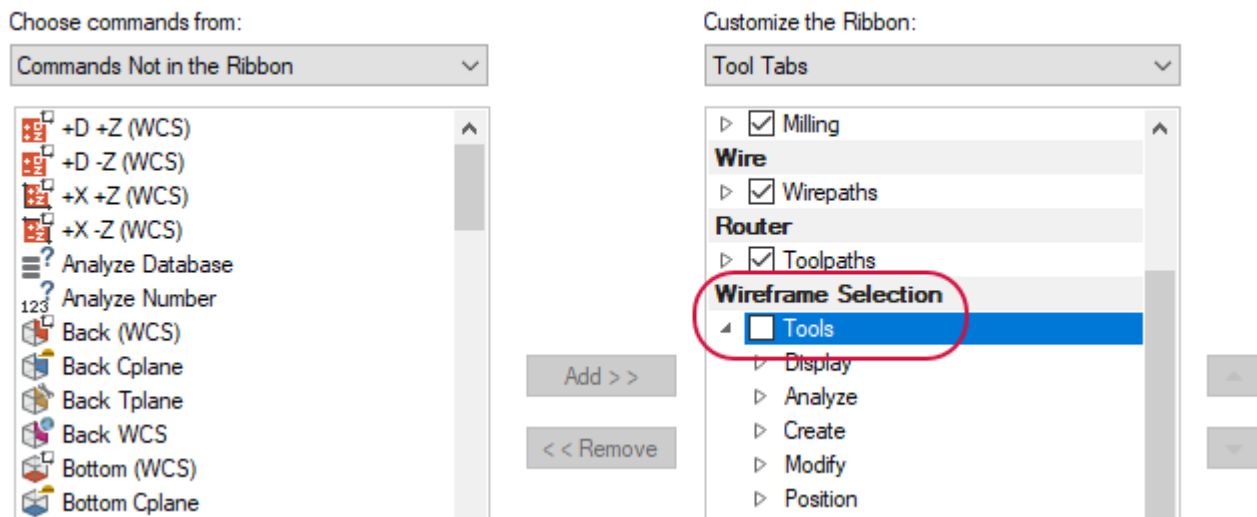


Similar tabs display when you select solid, surface, mesh, or a mixture of geometry types.

You can customize the tools that display on these ribbon tabs by going to **File, Options, Customize Ribbon**. Select **Tool Tabs** from the **Customize the Ribbon** column on the right side to see the selection tabs.



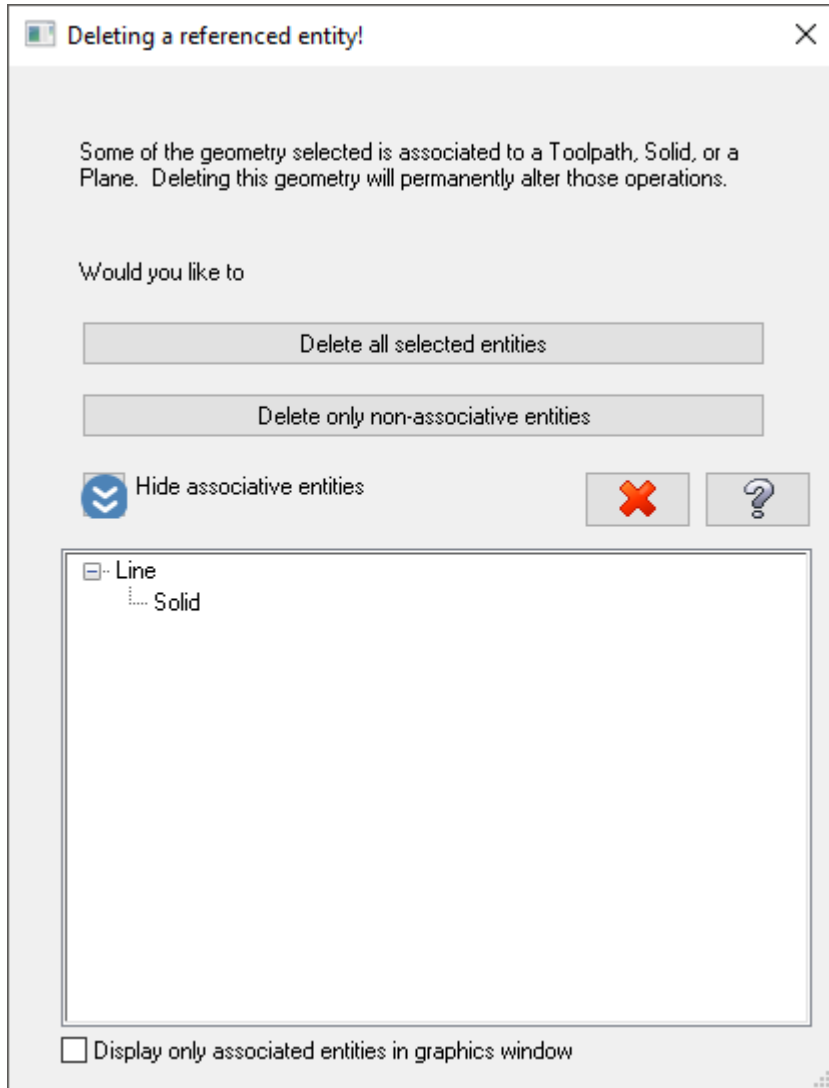
Add your own functions to the ribbon tabs, or remove groups of functions. To turn off these contextual ribbon tabs, deselect the **Tools** checkbox at the top of the list for the geometry type.



Deleting Referenced Entities

In previous versions, Mastercam would display the **Deleting a referenced entity** dialog box if the geometry selected for deletion included references to other entities. However, the actions of this dialog were limited to deleting all selected entities or deleting only non-associative entities. Moreover, you were unable to locate the geometry or references entities in the graphics window.

For Mastercam 2020, this dialog box has been enhanced so that you can visualize and have more control over your deletions.

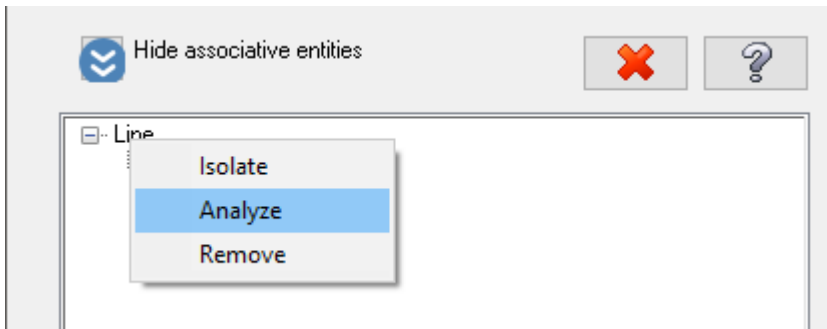


The **Show associative entities** button displays a list of all the referenced geometry that you have selected to delete. The tree-style control shows the association between selected geometry and referenced planes, toolpaths, solids, or entities. You can resize the dialog box to show as much or as little of the list as you like.

Use the **[Up]** and **[Down]** arrow keys or the scroll bar to navigate through the list. If you select an entity in the list, Mastercam locates it in the graphics window and highlights its references to toolpaths, solids, or planes.

Select the **Display only associated entities in graphics window** option to simplify what displays in the graphics window.

Right-click an entity and select **Isolate** to temporarily display only that entity in the graphics window, **Analyze** to get more information, or **Remove** to remove it from the list.

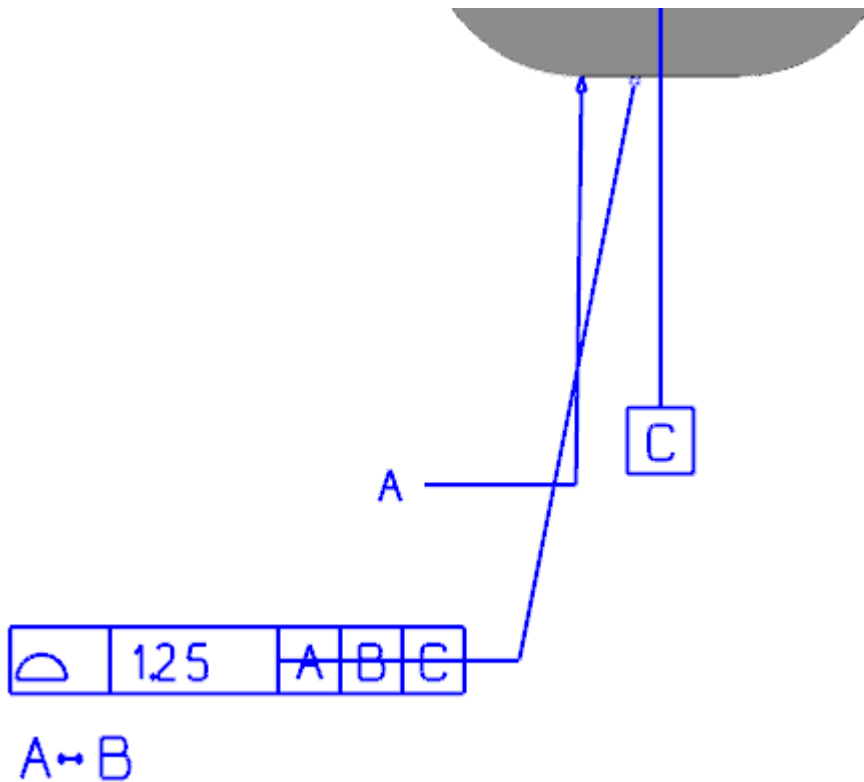


File Importers

Below are enhancements made to file importers for Mastercam 2020.

Imported 3D Annotations

The 3D annotations in imported Unigraphics/Seimens NX™, ProE/Creo™, STEP, Inventor, and CATIA™ files now point to the referenced geometry. In Mastercam 2020, this was only supported in imported STEP files.



STL Exports

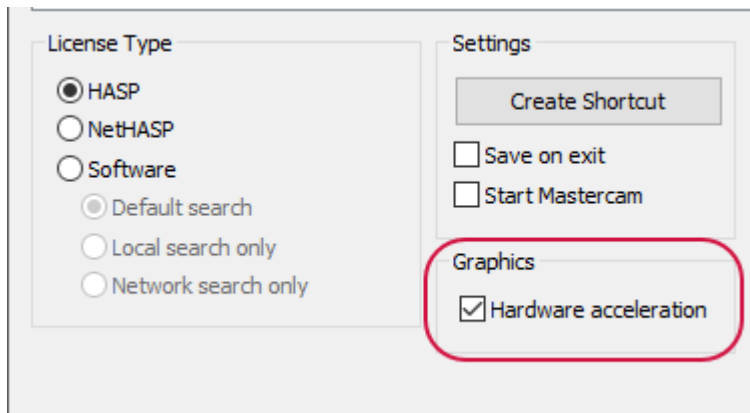
In previous releases, Mastercam exported STL data with the same units used in the current Mastercam session. In Mastercam 2020, a **Unit** drop-down has been added to the **Save as an STL File** dialog box and to the **Configuration/Converters** page in the **System Configuration** dialog box. Use this parameter to include unit data when exporting to software packages that require it, such as software that interfaces with certain 3D printers.

Additional File Support Enhancements

- You can now import Rhino 6, Solid Edge 2019, Catia V5 R28, and SOLIDWORKS 2019 files.
- Support for Spatial 2019 1.0 files when importing ACIS **.SAT** and **.SAB** files.
- Support for importing coordinate systems from **.STEP**, ProE/Creo, and Unigraphics/NX files. These coordinate systems will come into Mastercam as named planes.
- The ProE/Creo file imported has been enhanced, yielding better results for most files and allows for the import of 3D annotations.

Graphics Support

The **Hardware acceleration** option has been moved from the **Graphics Support** page in the **Mastercam Advanced Configuration** dialog box to the **Mastercam Launcher** dialog box, accessible from the Mastercam 2020 Start menu folder.



Machine Definition and Component Files

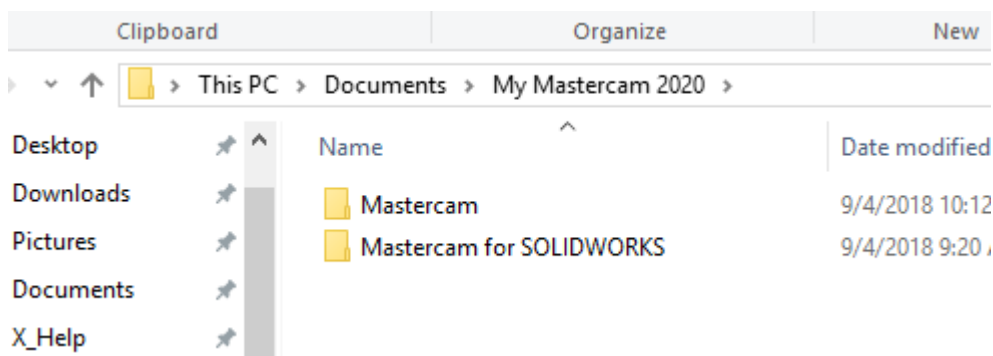
The Machine Definition and component files (*.mcam-mmd, *.mcam-lmd, *.mcam-rmd, *.mcam-wmd, and *.mcam-gmd) now open directly into the Machine Definition Manager or the Control Definition Manager when using Windows Explorer or a command line. Previously, the files opened as a Mastercam part file that required you to open the Machine Definition Manager or Control Definition Manager as an additional step.

Mastercam Folder Locations

In previous versions, Mastercam used the following default names and locations for installed folders:

Folder name	Default folder location
my mcam2019	C:\Users\User name\Documents
shared Mcam2019	C:\Users\Public\Documents
Mcam2019	C:\Program Files

For Mastercam 2020, the file structures have changed.



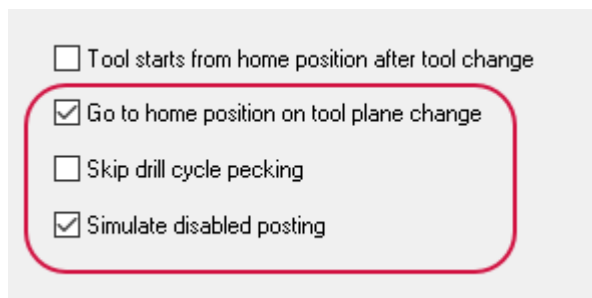
If you have Mastercam and Mastercam for SOLIDWORKS installed, both products now use the same root folder. If you only have one product installed, only that product folder will be in the root folder.

Folder name	Contains	Default folder location
My Mastercam 2020	<ul style="list-style-type: none"> • Mastercam • Mastercam for SOLIDWORKS 	C:\Users\User name\Documents
Shared Mastercam 2020	<ul style="list-style-type: none"> • Shared files and folders for all 2020 products. 	C:\Users\Public\Documents
Mastercam 2020	<ul style="list-style-type: none"> • Mastercam • Mastercam for SOLIDWORKS 	C:\Program Files

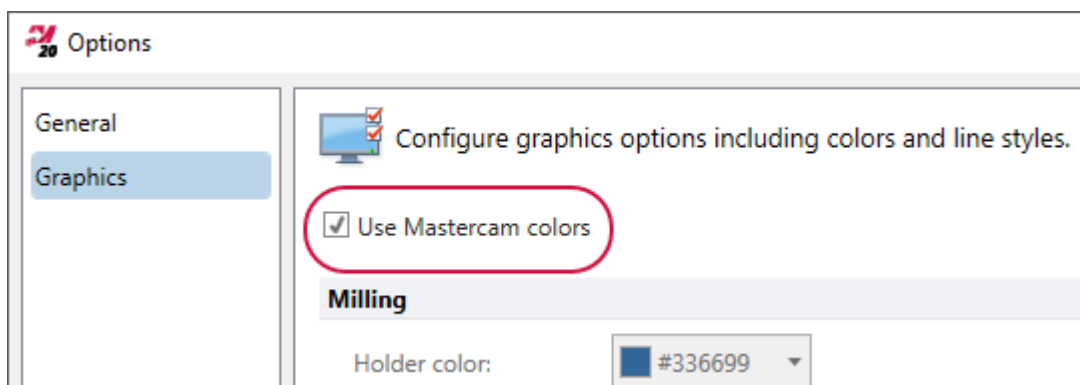
Mastercam Simulator Enhancements

Listed below are enhancements made to Mastercam Simulator.

- Two new options have been added to the **Simulator** page in the **System Configuration** dialog box:
 - **Go to home position on tool plane change:** Moves the tool to a home position between operations when there is a plane change.
 - **Simulate disabled posting:** Enables the simulation of operations for which posting has been disabled.

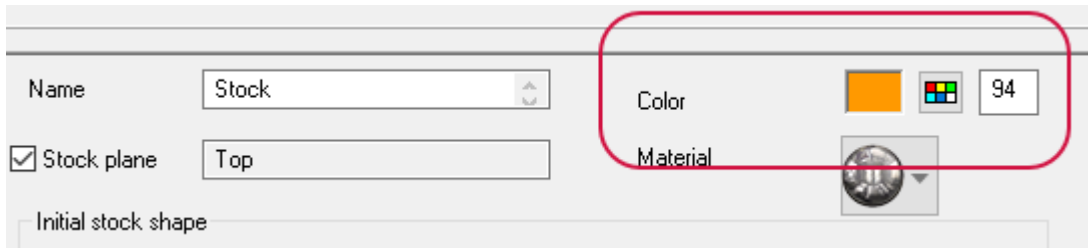


- The **Use Mastercam colors** option, available on the **Graphics** page, will inherit colors set in Mastercam and the color settings in Mastercam Simulator will be disabled.

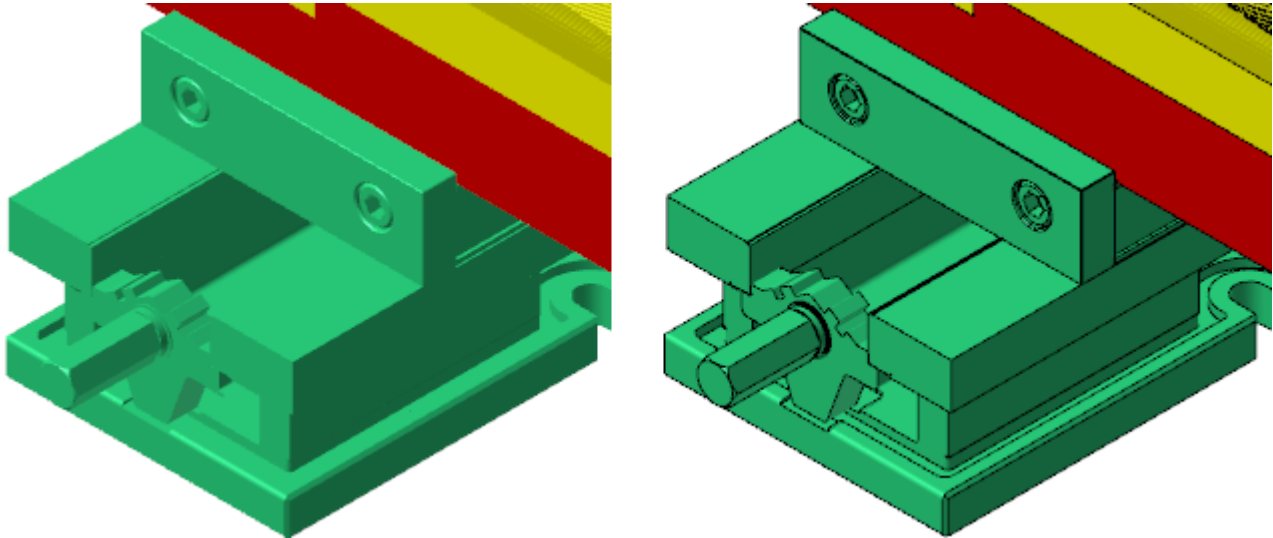


- Mastercam Simulator now colors workpieces, including those made up of several different models (surfaces, solids, or meshes), as one singular color making it easier to identify them during sessions.

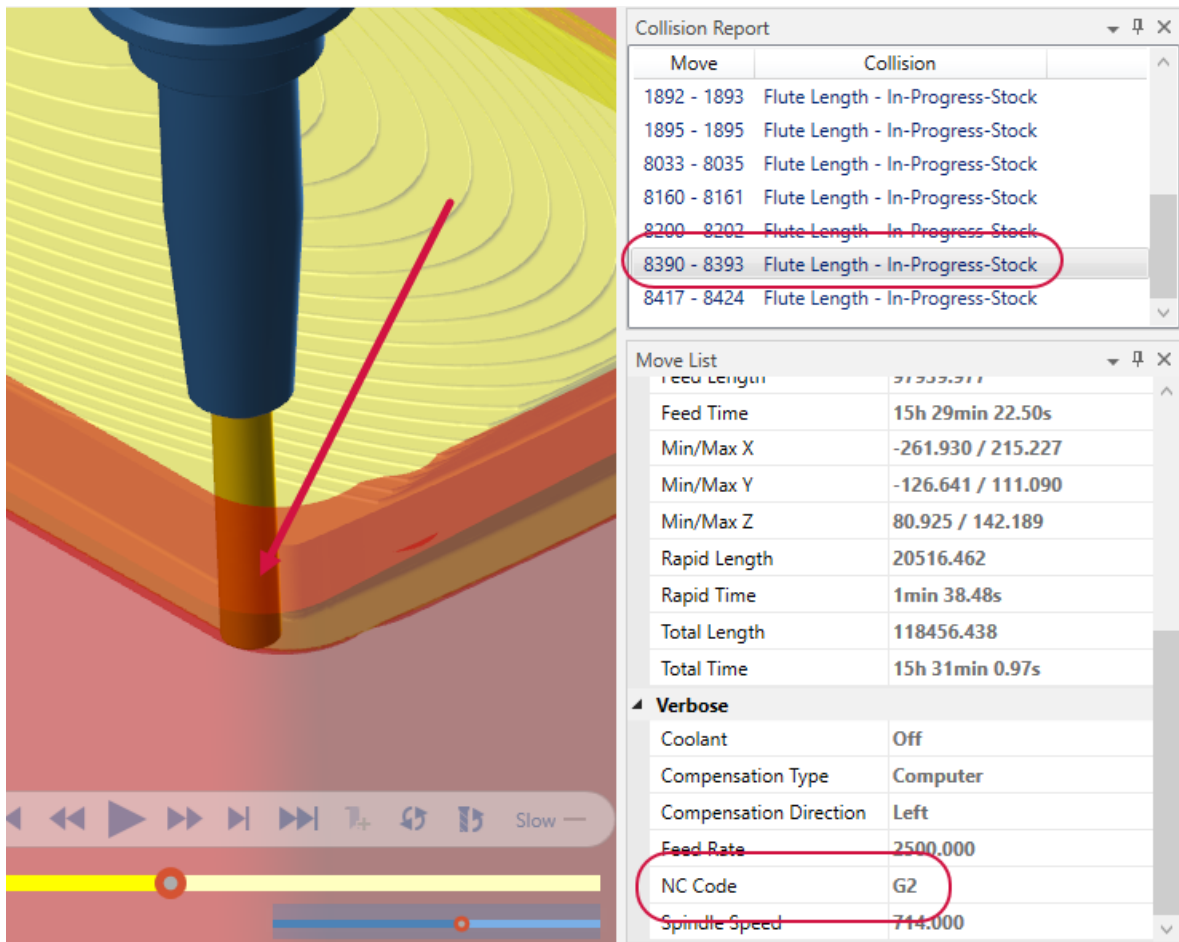
- The stock model color that is set on the **Stock Definition** page of the **Stock model** dialog box now displays in Mastercam Simulator.



- Edge highlighting has been improved for stock models, workpieces, and fixtures.



- In previous Mastercam versions, Mastercam Simulator displayed collisions when verifying back feed rate/microlift tool motion. Microlift moves are now correctly checked and will be identified in Mastercam Simulator if they encounter material.



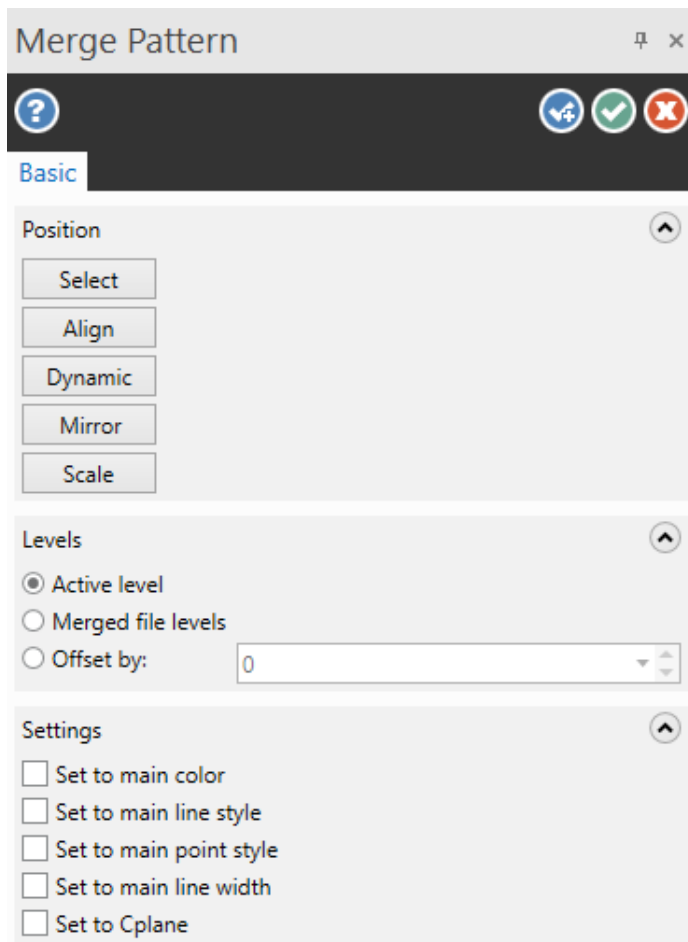
Rotary Axis Positioning

Mastercam Simulator, when using **Verify** mode, now simulates the **Rotary axis positioning** option located on the **Rotary Axis Control** page of toolpaths. This allows you to simulate and visualize multiaxis movement of the machine tool. When this option is enabled, Mastercam Simulator will:

- Display the rotation of stock between tool planes.
- Display the tool axis orientation.
- Display cutting moves in the rotated tool plane.

Merge Pattern

The Merge Pattern function (accessed by selecting **Merge** on the **File** tab) has been improved for Mastercam 2020.



Position

You can now use Align to Face or select a transform method (Dynamic, Mirror, or Scale) to position the merged part geometry. Mastercam applies the result color to the merged geometry so it is easy to identify for further editing.

The **Reselect** button has been renamed **Select**, but retains the same functionality seen in previous versions of Mastercam.

Levels

In previous versions of Mastercam, you were limited to merging part geometry into the active level. For Mastercam 2020, you can choose on which level merged geometry will be placed.

The new Levels group on the **Merge Pattern** function panel has three options:

- **Active level:** Merges part geometry into the current file's active level.
- **Merged file levels:** Merged part geometry remains on its original levels.
- **Offset by:** Mastercam places the merged part geometry on new levels whose numbers are offset by the value entered in the field. The merged part geometry stays separate from the target part geometry.

Mastercam applies your level preferences after you click **OK** or **OK and Create new operation**. Merge Pattern accounts for any level modifications you make to the **Levels** option when you use the Dynamic, Mirror, or Scale methods to position the geometry. However, if you select **Active levels**, Mastercam ignores these modifications and merges the part geometry into the original part's active level.

Attribute Settings

You now have more control over the attributes of your merged part. In previous Mastercam releases, the **Use current attributes** checkbox applied only the system color and level to the merged part. In Mastercam 2020, you can now apply the current system attributes for color, level, line style, point style, and line width to the entities in your merge part.

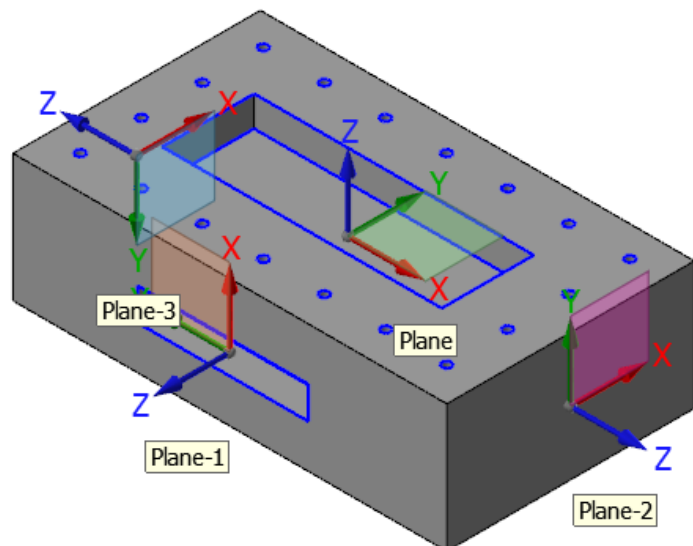
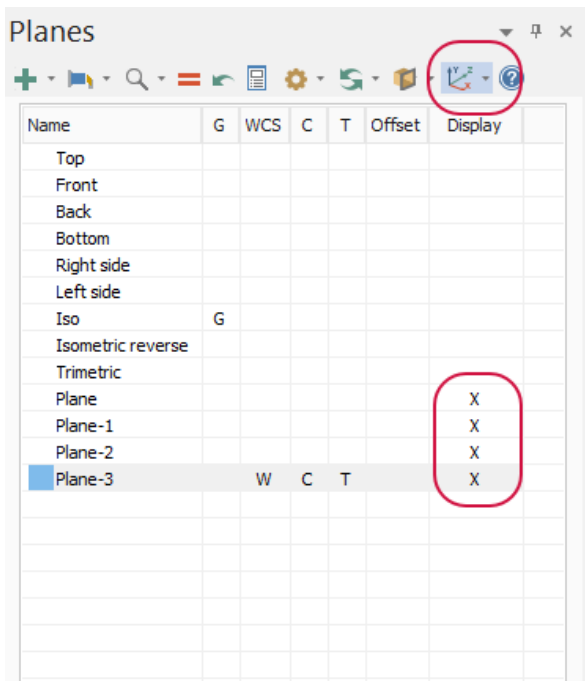
Planes Manager

Listed below are enhancements that have been made to the Planes Manager for Mastercam 2020.

Persistent Plane Display

In previous Mastercam versions, the **Display** column in the Planes Manager determined which planes display when using **Find plane**, **From plane**. In Mastercam 2020, this column controls which plane gnomons display in the graphics area when the **Show Gnomons** button is toggled on.

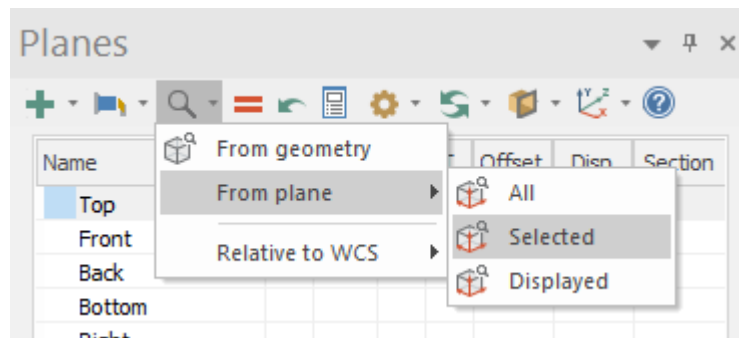
If you select the **Display** column for a plane, an "X" will appear in the column. With the **Show Gnomons** button toggled on, the gnomon for each plane that contains an "X" in the **Display** column will remain on the screen. By selecting the **Display** column again (removing the "X"), the gnomon will no longer display on the graphics window. With the **Show Gnomons** button toggled off, all gnomons are removed from the graphics window, even if the **Display** column is selected.



Find a Plane

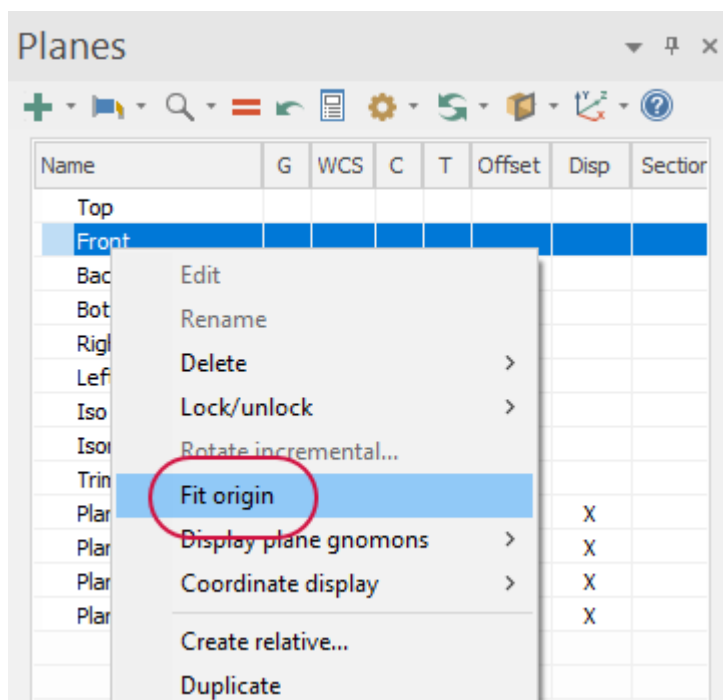
The **Find a plane, From plane** option now includes the following options:

- **All:** Displays all plane gnomons and prompts you to select one in the graphics window.
- **Selected:** Displays all gnomons for the planes selected in the Planes Manager and then prompts you to select one in the graphics window.
- **Displayed:** Displays only the gnomons for the planes that have the **Display** column selected and prompts you to select from the graphics window.



Fit Origin

A new option, **Fit origin**, is available in the right-click menu of the Planes Manager. Fit origin will fit both your part and the origin of the selected plane in the graphics window.



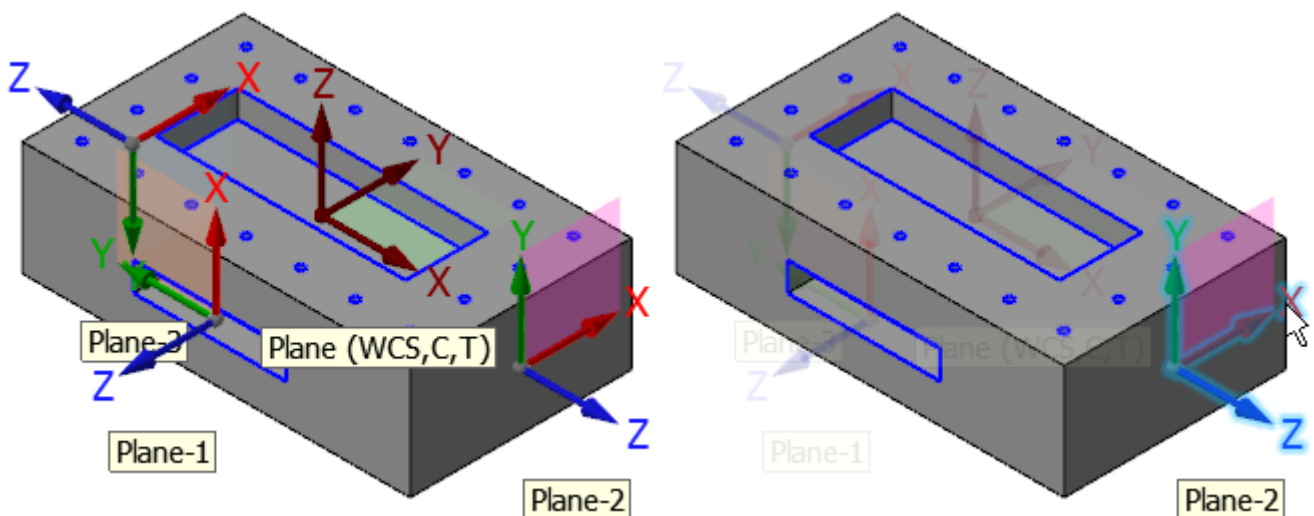
Plane Gnomon Enhancements

The **Plane Gnomon Settings** dialog box includes new options to control the display of the plane gnomons.

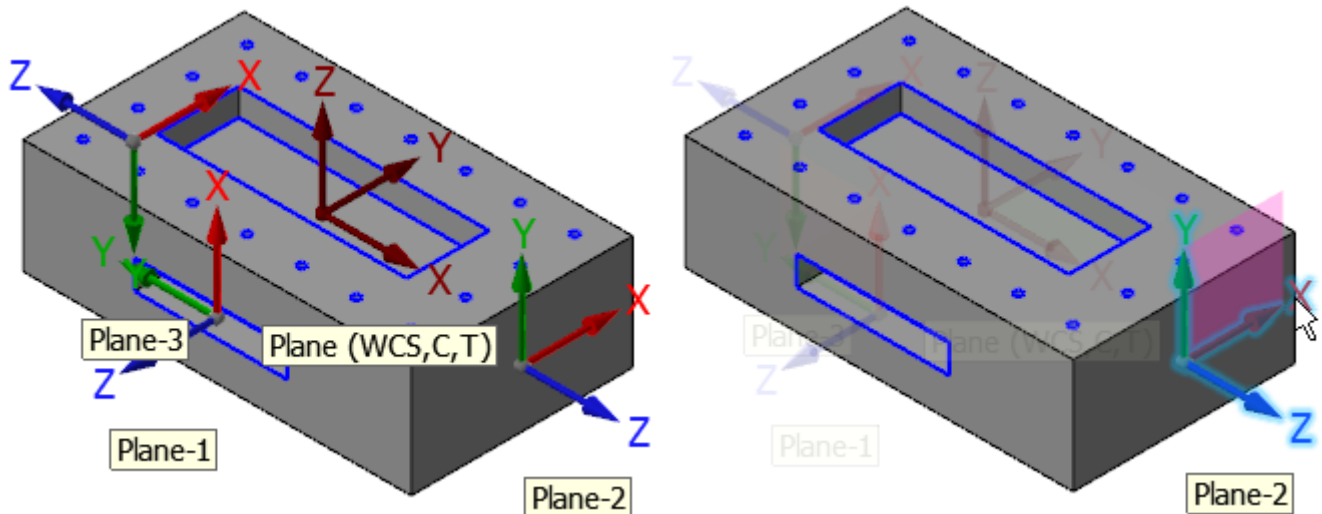


Use the **Opacity** slider under **Axes display** to control the opacity of the plane gnomon axes. Additionally, when **Show Gnomons** is toggled on, the follow options are active:

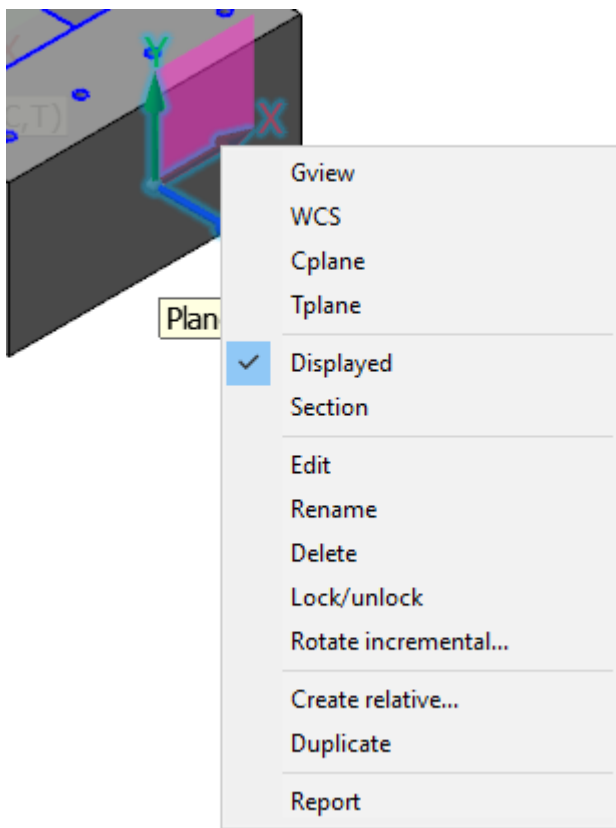
- You can hover over an individual plane to highlight it in the graphics window. Any other displayed plane gnomons are dimmed.



- Select **Only when highlighted** to display the plane of a gnomon when the gnomon is highlighted in the graphics window.



- You can right-click on a plane gnomon to access functionality without having to reopen the Planes Manager.



Show Gnomons

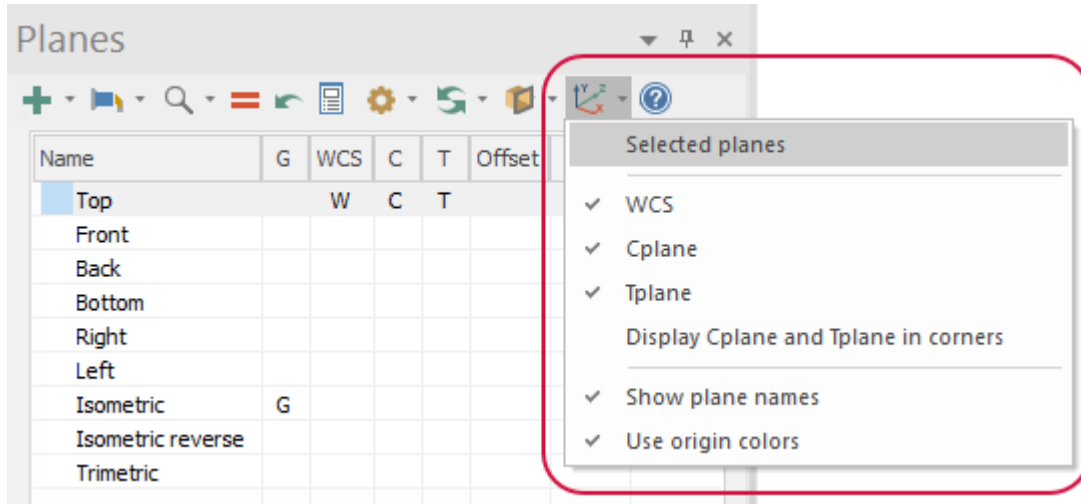
In previous Mastercam versions, the **Show Gnomons** button contained a drop-down that controlled the display of the WCS, Cplane, and Tplane gnomons. This button was only available on the **View** tab.

In Mastercam 2020, **Show Gnomons** has also been added to the Planes Manager with these new options:

- **WCS:** Displays a static plane gnomon at the WCS origin, rather than a single color gnomon.
- **Cplane:** Displays a static plane gnomon at the Cplane origin.

- **Tplane:** Displays a static plane gnomon at the Tplane origin.

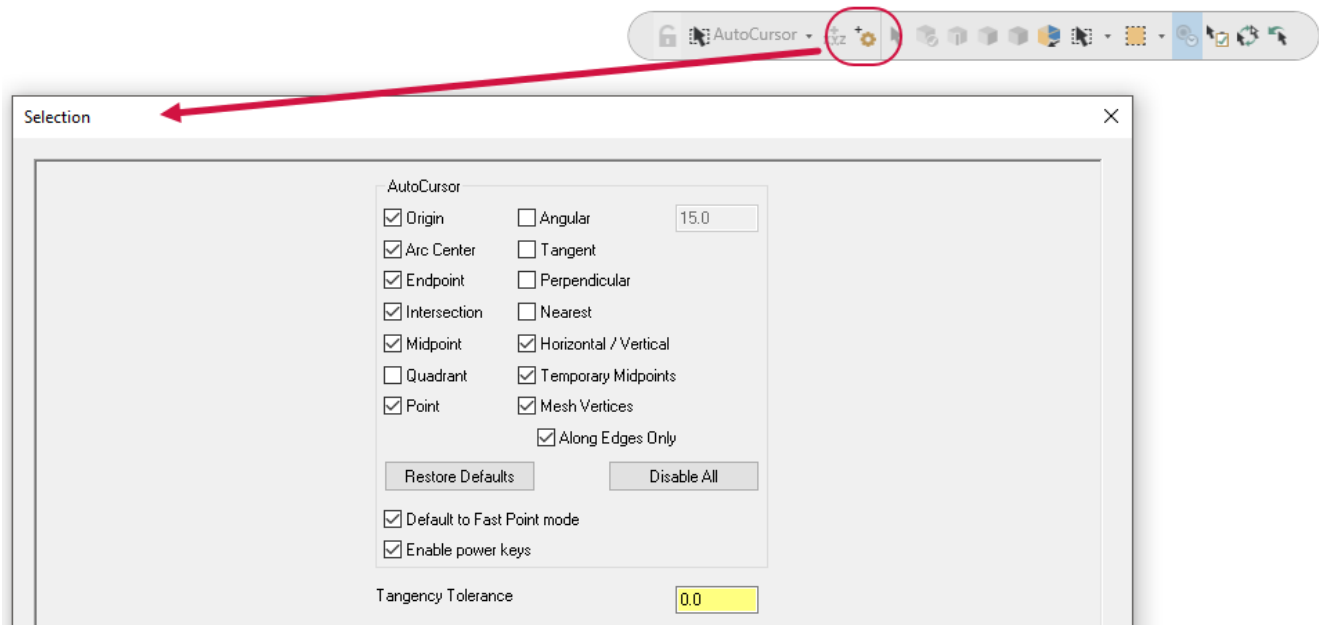
Additionally, the **Always display gnomon** option has been moved from the **Display Options** menu to the **Show Gnomons** menu and has been renamed **Selected planes**. This option has the same functionality as it did in previous versions. When **Selected planes** is selected and **Show Gnomons** is toggled on, the gnomons for all selected planes will always display in the graphics window.



Selection Enhancements

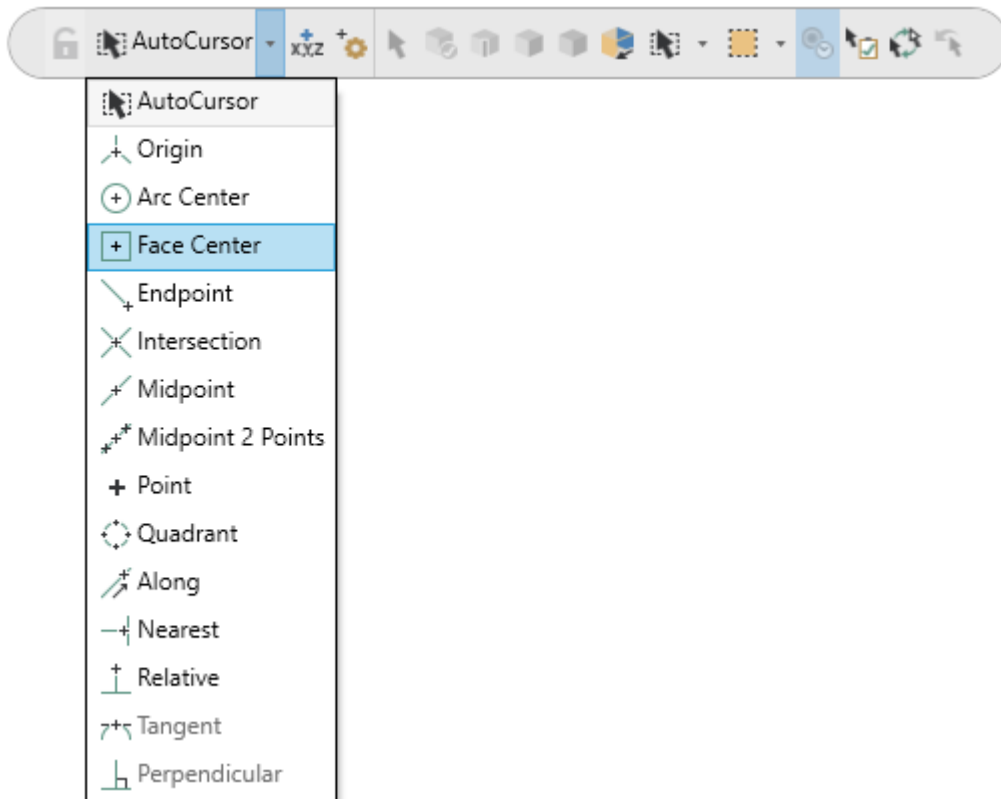
AutoCursor® Settings Renamed Selection Settings

The **AutoCursor Settings** dialog box, opened by selecting **AutoCursor Configuration** on the Selection Bar, has been renamed **Selection** and added to the **System Configuration** dialog box. The option on the Selection Bar has also been renamed to reflect this change.



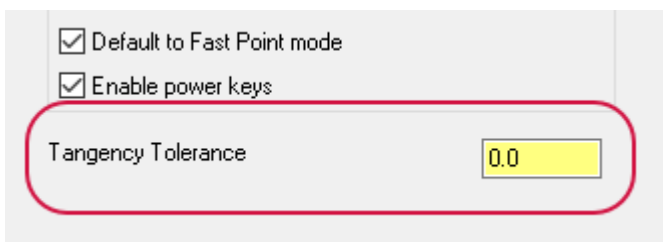
Face Center

AutoCursor now supports a face center override mode that places the AutoCursor at the center of gravity of the selected face. Select **Face Center** from the **AutoCursor** drop-down on the Selection Bar, or press [F] to activate this override.



Tangent Selection

A new option, **Tangency Tolerance** has been added to the **Selection** page of the **System Configuration** dialog box. In previous releases, when expanding a selection to tangent faces or edges, the tolerance was limited to neighboring entities that aligned with the selected entity. Mastercam did not recognize a neighboring entity as tangent if the angle variation was greater than 0. This new option allows you to expand the entities that Mastercam recognizes as tangent by entering in a value.

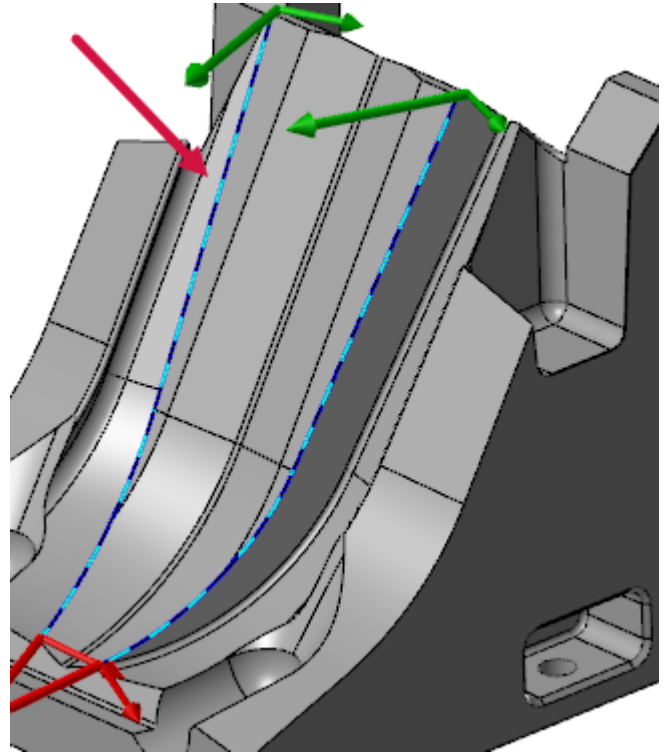
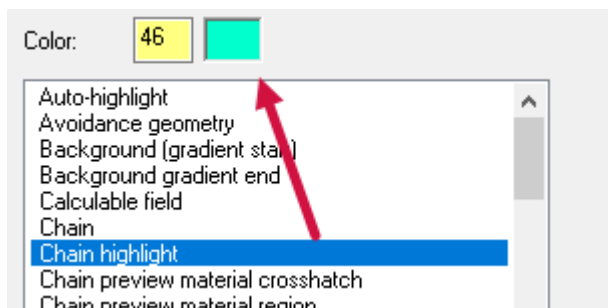


System Configuration

Listed below are enhancements to options found in the **System Configuration** dialog box, accessed from the **File** tab.

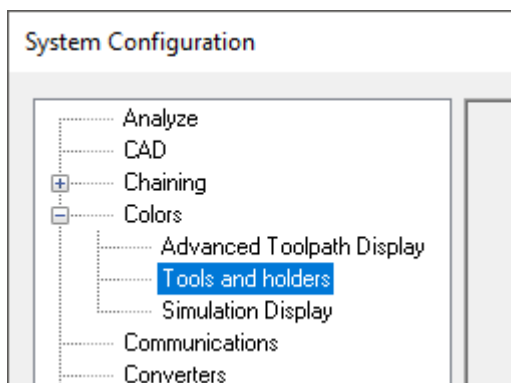
Chain Highlighting

When viewing chains in the Chain Manager, the chains are highlighted in the graphics window. In previous versions of Mastercam, you could not control the color of this highlight. In Mastercam 2020, you can use **Chain highlight** in the **Colors** page to control the highlight color.

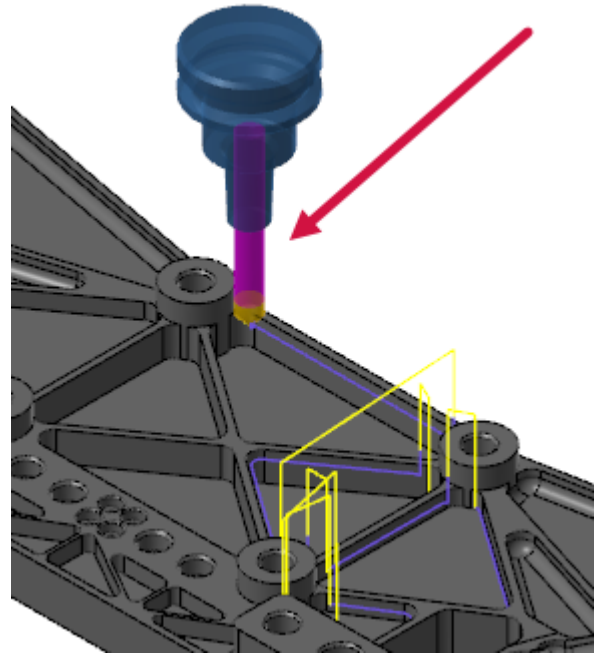
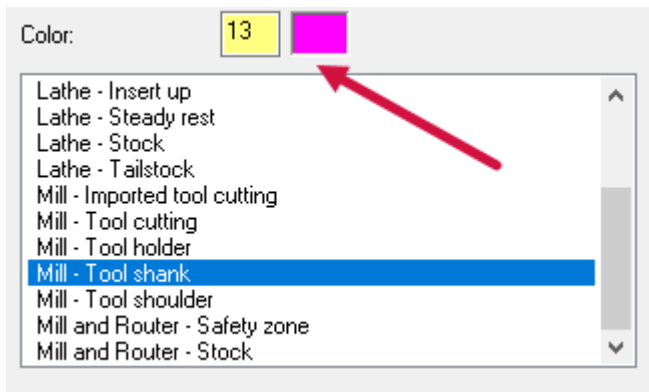


Color Options

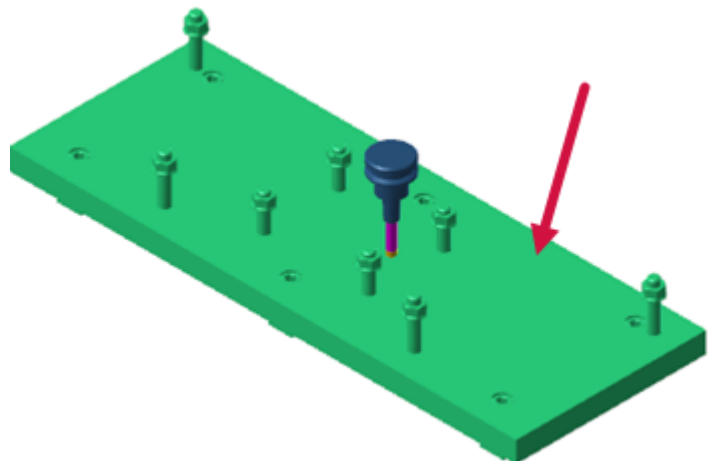
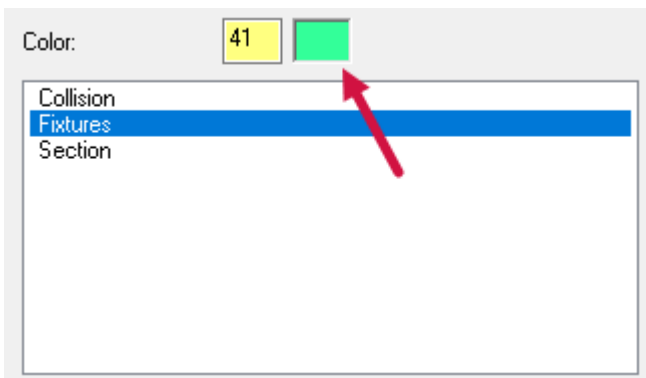
Expanded color options are available in the **System Configuration** dialog box. These colors display in various areas of Mastercam.



Use the **Tools and holder** page to set colors for lathe, mill, and router tools .

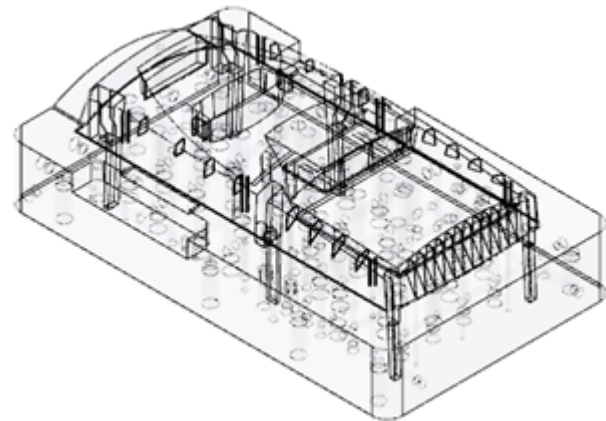
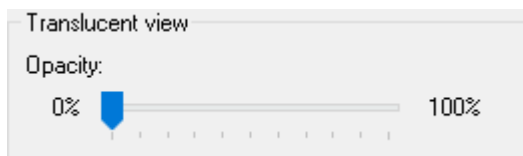
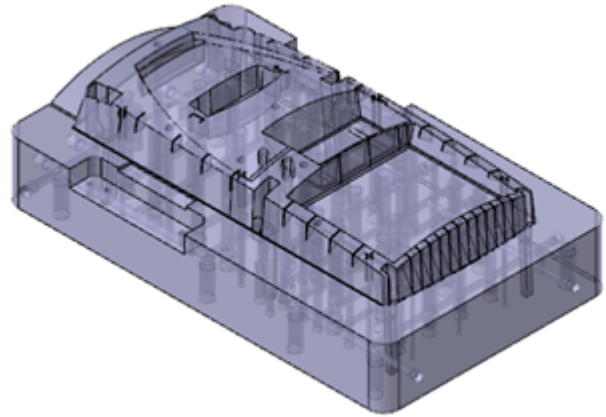
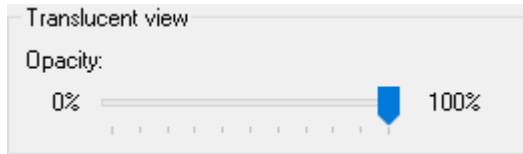


Use the **Simulation Display** page to set colors for collisions, fixtures, and sections. These colors only apply to Mastercam Simulator.



Translucent View

You can now change the opacity level when **Translucency** (on the **View** tab) is toggled on. Use the **Opacity** slider under **Translucent view** on the **Shading** page to make adjustments.

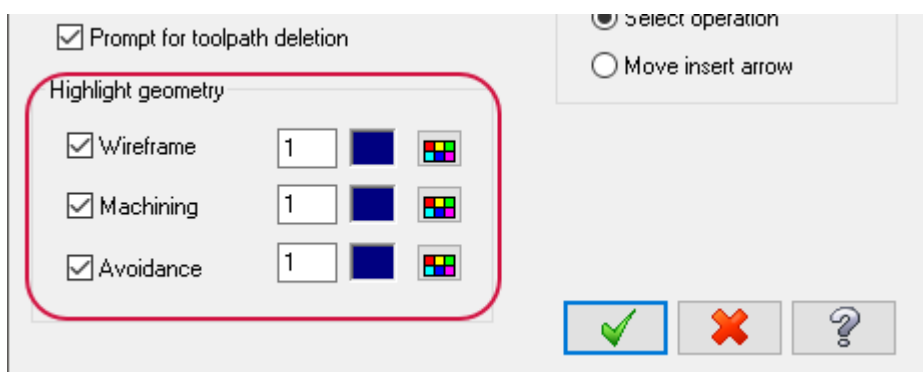


Toolpaths Manager

Listed below are enhancements made to the Toolpaths Manager.

Highlight Geometry

The **Highlight geometry** group, which was removed in a previous Mastercam version, has been returned to the **Display Options** dialog box.



When selected, **Wireframe**, **Machining**, or **Avoidance** geometry is highlighted by the chosen color for the selected toolpath.

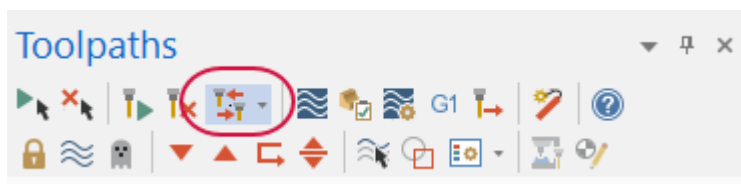
Dependencies

The **Dependencies** group has been moved from the **Display Options** dialog box to the Toolpaths Manager toolbar. The new **Dependencies** toggle allows you to pick one or both of the following:

- **Selection:** Selects all dependent operations when you select the parent operation.
- **Regeneration:** Regenerates all dependent operations when you regenerate the parent operation, even if the dependent operations are not selected.

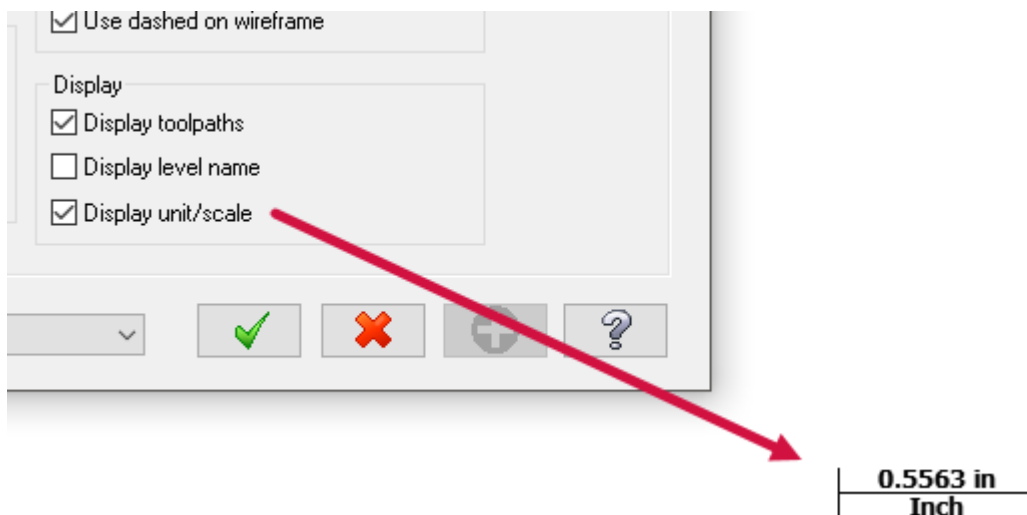
To enable these options, select one or both options from the **Dependencies** drop-down. Then select the **Dependencies** button to toggle the selected option on. Select again to disable the options.

When enabled, the button has a dark background.

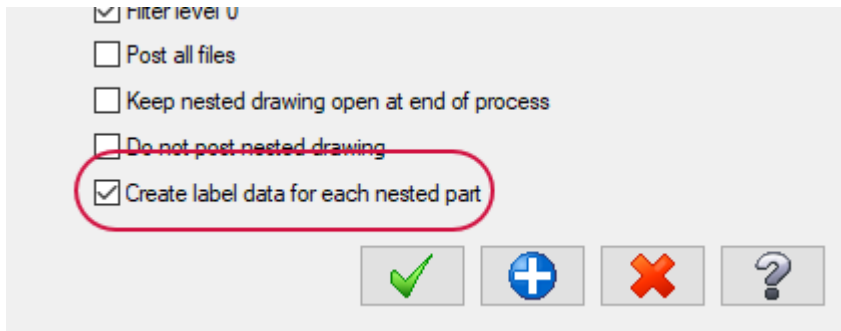


Additional Enhancements

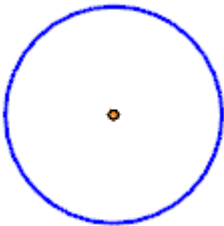
- The Dynamic gnomon can now change between 2D and 3D mode. Use the **Change gnomon style based on 2D/3D Cplane** option in the **Gnomon Settings** dialog box to control this. This option is on by default. Right-click the Dynamic gnomon when you are using it in the graphics window to display the **Gnomon Settings** dialog box.
- You can toggle the display of the screen unit and scale by selecting **Display unit/scale** on the **Screen** page in the **System Configuration** dialog box.



- You can print a label for each instance of a part on a nested sheet. Select **Create label data for each nested part** on the **Options** dialog box for Automatic Toolpathing (ATP).



- The **Arc Center Points** option on the **Home** tab is now toggled off by default. Also, when toggled on, the center points are easier to see.



- The Fit function now compensates for the Selection Bar and Quick Masks. The shown geometry will not longer be obscured by the overlays.

DESIGN ENHANCEMENTS

Listed below are enhancements to design functions.

Edit Surface and Edit Spline

Edit Spline has been modified to be more consistent with the Edit Surface function. The more intuitive design and similar workflow should enhance your ability to make changes to splines and surfaces.

Multiple Node and Control Point Selection

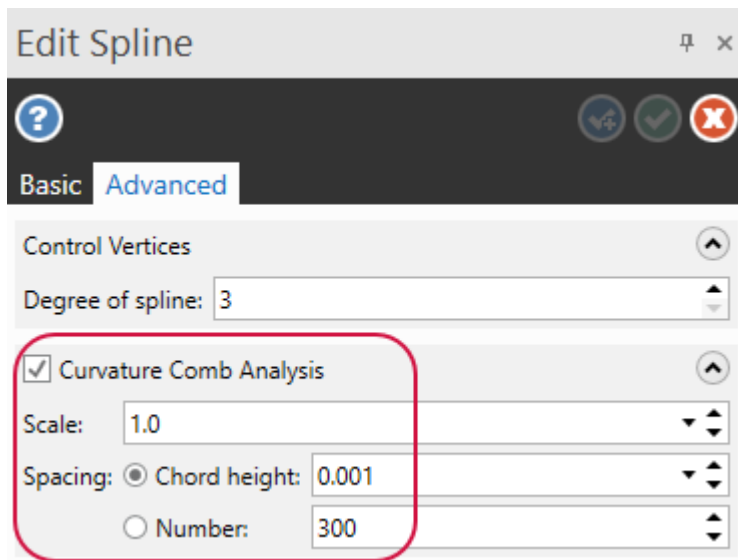
You can now select and manipulate multiple nodes and/or control points for Edit Surface and Edit Spline. Press **[Ctrl+click]** to select multiple nodes and control points. Press **[Enter]** to begin editing.

Gnomon Control

Mastercam 2020 gives you greater control of gnomon alignment when you edit node and control points in the XYZ mode of the binormal plane. Select a node or control point to align the gnomon normal to the surface, spline (default), or to the Cplane.

Curvature Comb

The sliding scale used to control the curvature comb in previous Mastercam versions has been replaced. In Mastercam 2020, control the display of the curvature vectors by entering a **Scale** value. You can also control comb spacing with the new **Chord height** and **Number** options.



Curvature combs represent the curvature of the surface. They display curvature continuity or discontinuity.

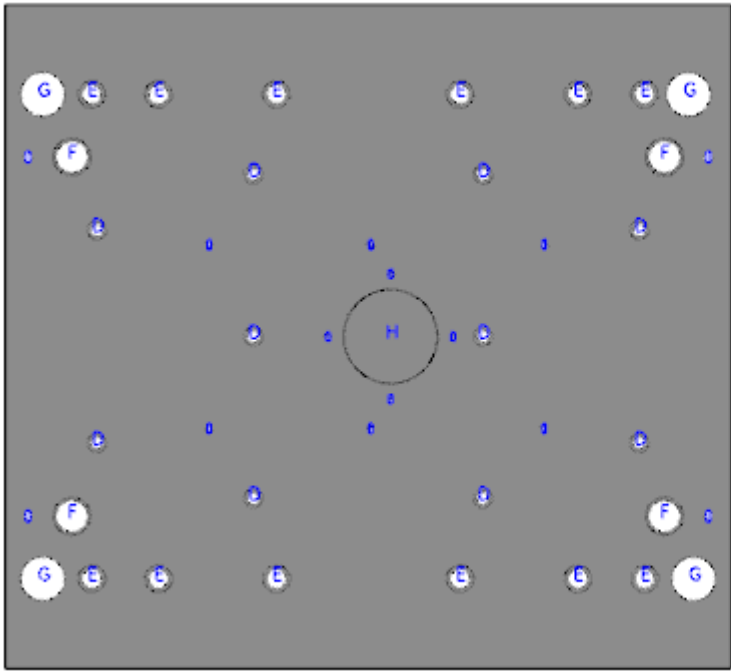
Hole Table

Hole Table, which was previously an add-in in previous Mastercam releases, is now integrated with Mastercam and has been improved. Select **Hole Table** on the **Drafting** tab to use this new function.

The screenshot shows the 'Hole Table' dialog box with the following sections:

- Selection**:
 - Method: ☒ Wireframe, ☒ Faces, ☐ Edges, ☐ Bodies
 - Holes: (Empty list box)
 - Buttons:
- Table**:
 - Move: (Button)
 - Text height: 5.0 (Spin box)
- Label**:
 - Alignment: ☒ Center, ☐ Endpoint

Hole Table creates parameter notes for all holes (circles) in your part file, allowing you to gather hole data and record it in a table for part inspection. In previous Mastercam releases, it was limited to finding hole geometry only on the active plane and only recognized wireframe geometry.



Ref.	Diameter	Count
A	6.0	4
B	7.0	6
C	8.0	4
D	18.0	10
E	26.0	12
F	36.0	4
G	42.0	4
H	90.0	1

Selection

As an add-in, Hole Table automatically recognized wireframe hole geometry. Now, Hole Table requires you to select geometry, but the type of geometry you can select has expanded to include wireframe, faces, edges, and bodies. You can filter your selection to any combination of these types, as well as individually remove a selection. Additionally, you can control the direction of each selected hole.

Display and Position

You can define the scope, size, and position of the table and labels describing the hole geometry. You can describe the holes in radii or diameters. You can also add location coordinates (in 2D and 3D) for each hole relative to the origin/world Cplane, Tplane, and WCS.

On the **Advanced** tab, you can show or hide the hole table and direction arrows in the graphics window.

Reporting

You can create a report directly from Hole Table by selecting **Create** under the **Active Reports** group.

TrueType® Fonts

Mastercam now supports single-stroke TrueType fonts for Create Letters (on the **Wireframe** tab) and Note (on the **Drafting** tab).

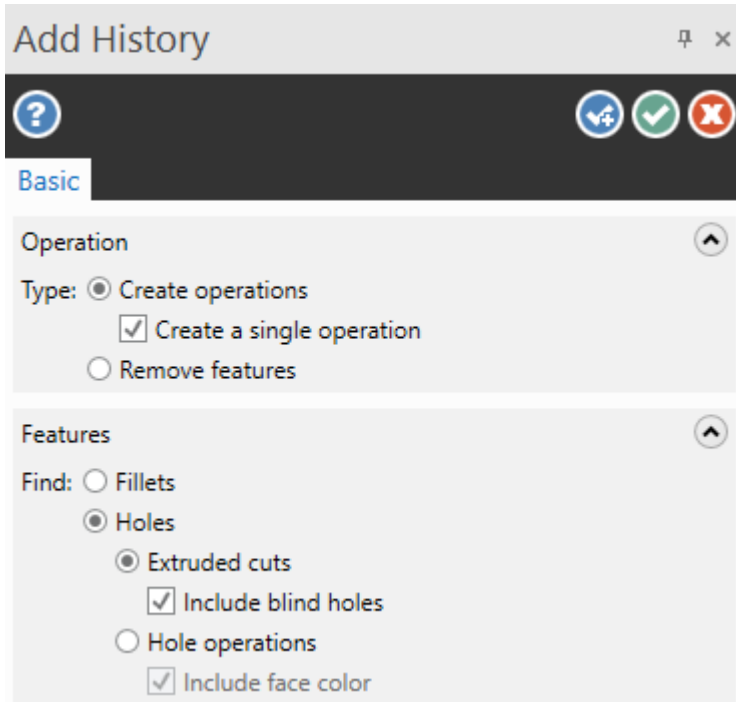
Mastercam also includes a new single-stroke font called OLF Simple Sans CJK OC. This font is similar to Arial and has over 35,000 characters including 400 glyphs supporting English and Romance languages, 200 Cyrillic characters, 300 Greek/Coptic characters, 70 Hebrew characters, several hundred accented Latin characters, and over 33,000 Chinese, Japanese, and Korean characters.

Model Prep Enhancements

Listed below are enhancements made to Model Prep functions.

Add History

Find Features has been renamed Add History. This new name reflects the enhancements that have been made to the function for Mastercam 2020.



Add History now creates hole operations, which were introduced in Mastercam 2019. When you select the **Hole operations** option, holes in a selected solid without history are recognized and saved in a new Hole operation. The Hole operation then displays in the Solids Manager.

Additionally, Add History now supports the selection of multiple bodies.

Align to Face

You can now slide a solid along a vector until it is within **.001** inches or millimeters of the face of the target solid. Use the gnomon to move the solid along a vector. Press **[Shift]** to stop the motion when it contacts another solid face. Press **[Shift]** again to allow the solid to move freely through solid faces. Additionally, Align to Face now supports multiple undo and redo commands.

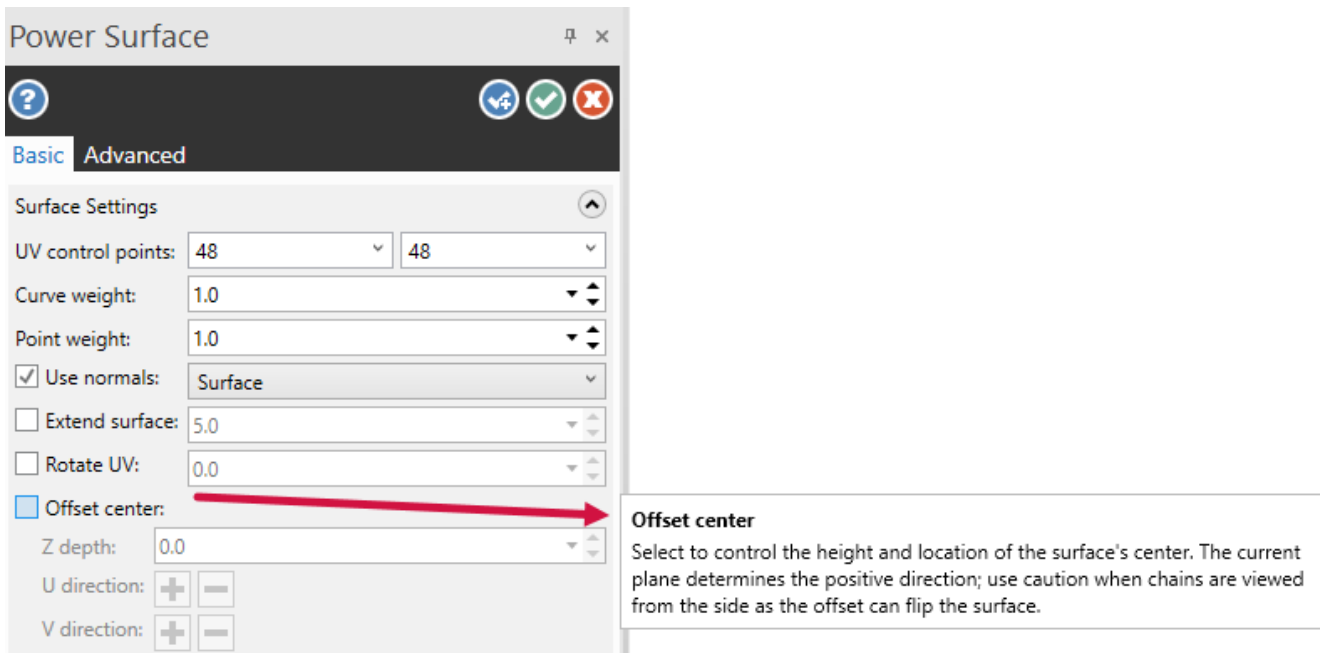
Push-Pull

You can now use Push-Pull to create a new solid body, instead of modifying the source solid.

- Select **Move** to extend or shrink the solid body.
- Select **Copy** to create a new body using the selected face as the base.

Power Surface

The Power Surface function, located on the **Surfaces** tab, now uses the same panel interface as other Mastercam CAD functions. The new interface offers more robust tooltips to help you with each option without requiring you to open the Mastercam Help.



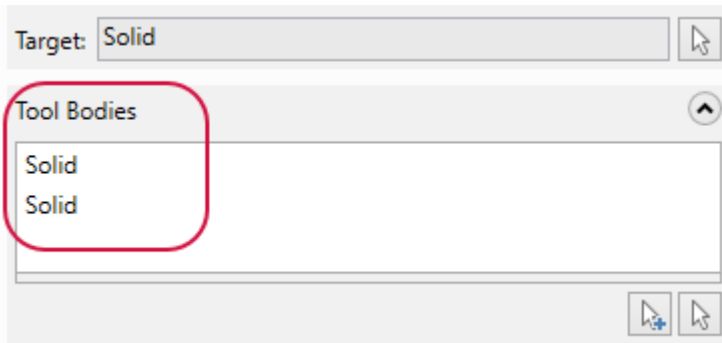
Solids Enhancements

Listed below are enhancements made to Solids functions.

Boolean

In Mastercam 2019, if the **Target** and **Tool Bodies** were selected before the panel opened, the **Boolean** function panel would be in an error state. In Mastercam 2020, Boolean recognizes if you have pre-selected the **Target**, or not. If the **Target** is not pre-selected, you are prompted to select one when you start the function.

Additionally, you can now select multiple **Tool Bodies**.



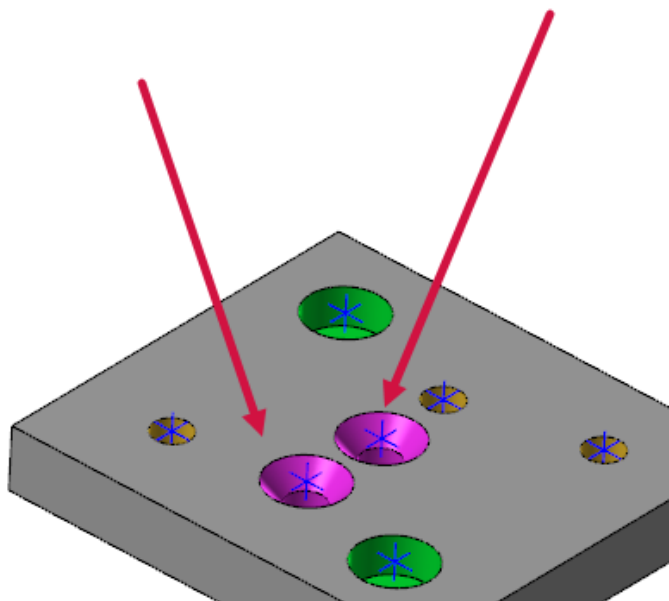
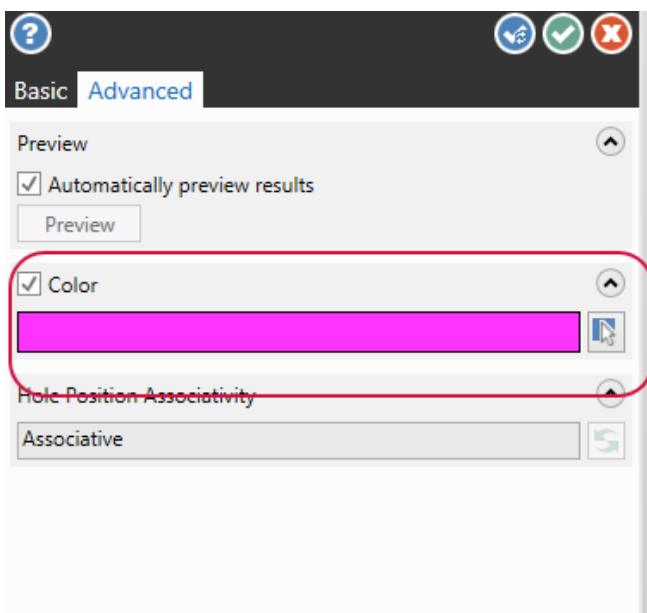
This is helpful if you are extruding multiple bodies, such as letters for a sign. Use window-select in the graphics window, [**Shift+click**] or [**Ctrl+click**] in the Solids Manager to select multiple solids. You must select the solid body at the top level.

Solid Hole

Mastercam 2020 includes many enhancements to the Solid Hole function.

Assign Colors

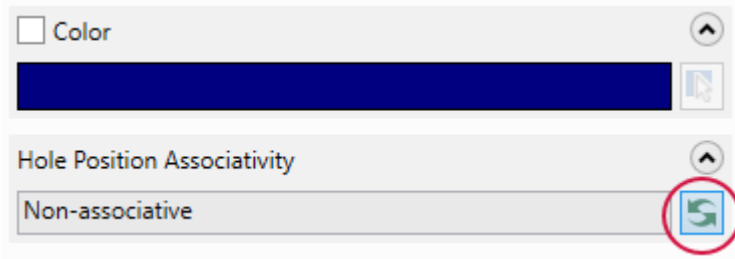
You can now assign colors to Solid Hole operations. Feature color can be saved as part of a custom hole definition when you add a hole to a **Presets** list. Click the **Advanced** tab and select **Color** to modify the operation.



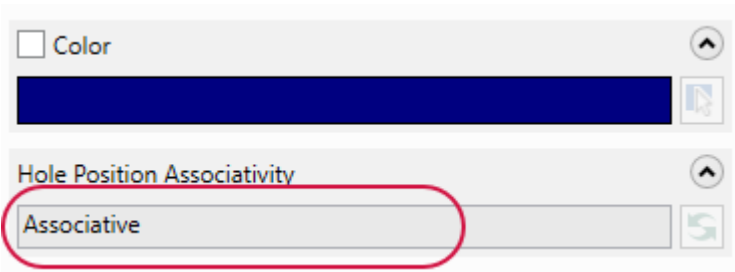
This enhancement is especially useful when automating drilling operations with ProDrill.

Associativity

A new option allows you to make Solid Hole operation positions that were created using Mastercam 2019 associative. To update operations made in Mastercam 2019, edit the Solid Hole operation, select the **Advanced** tab, and **Update**.



The **Hole Position Associativity** parameter updates from **Non-associative** to **Associative**. Mastercam creates point entities if no points or arc centers were associated with the original operation.



New Selection Controls

Solid Hole has two new selection controls:

- **Add AutoCursor Position:** In addition to point and arc selection, you can now select a solid hole position using the AutoCursor. Mastercam creates a point entity at any cursor or AutoCursor position.
- **Remove All:** Deletes all prior selections in the **Position** list box.



Top Chamfer Input

When you create a chamfer at the top of the holes, you can now directly enter the actual chamfer size instead of entering a percentage of the diameter.

Wireframe Enhancements

Listed below are enhancements made to Wireframe functions.

Curve Enhancements

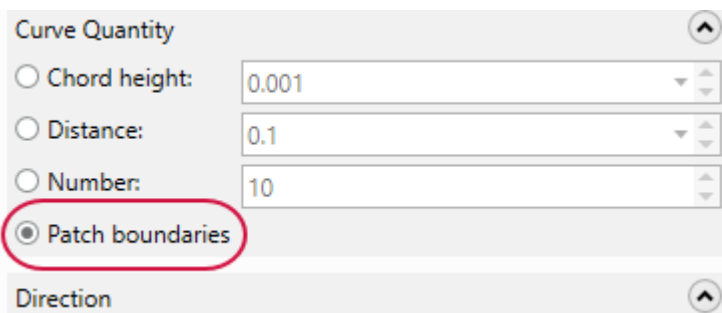
Curve Slice, Curve All Edges, and Curve Flowline functions have new enhancements.

Curve All Edges

In previous Mastercam versions, you could only create curves from the outer boundaries of selected solid faces. In Mastercam 2020, you can now create curves on the inner edges of selected solid faces.

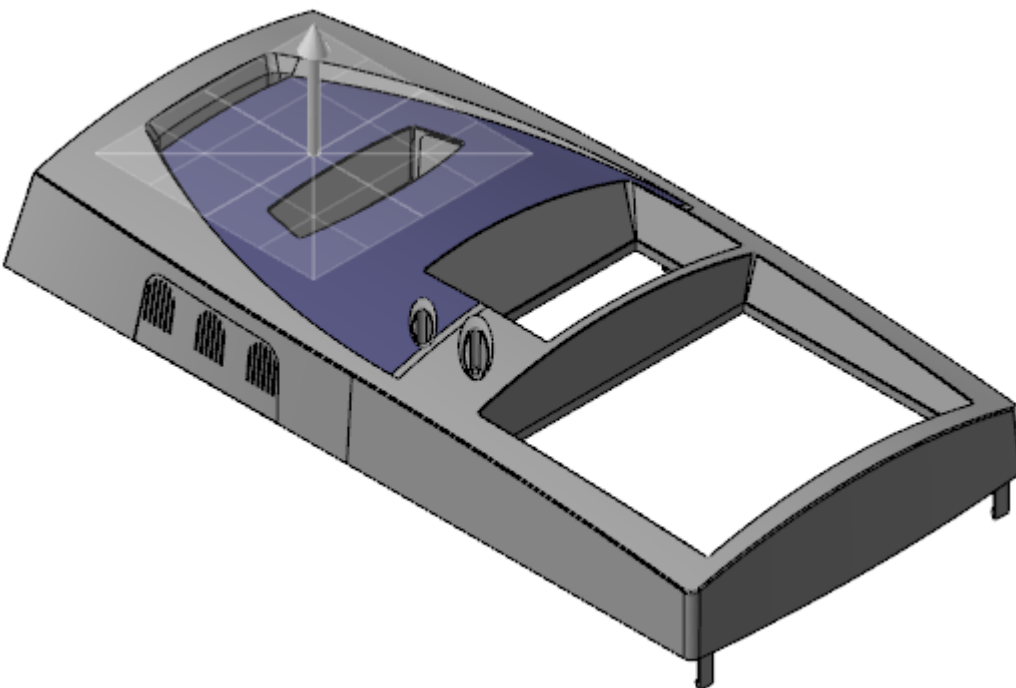
Curve Flowline

Curve Flowline now creates curves that follow the surface's patch boundaries. Patch boundaries are true representations of the underlying surface.



Curve Slice

The plane indicator that displays when using the Curve Slice function has been improved, giving a better picture of the plane.



Divide

The Divide function which allows you to trim or break entities graphically by dragging the mouse over them while holding down the left mouse button, now includes a graphical undo. To undo a change, continue to hold the left mouse button down and move your mouse over the marker that Mastercam displays at the trim/break location. Mastercam remembers the sequence of your trim and break actions, so you can undo more than one past action.

If you complete a divide sequence (releasing the mouse button), you still have the options to use the controls on the Quick Access Toolbar, or [Ctrl+Z] and [Ctrl+Y] to undo or redo your actions.

Trim Functions

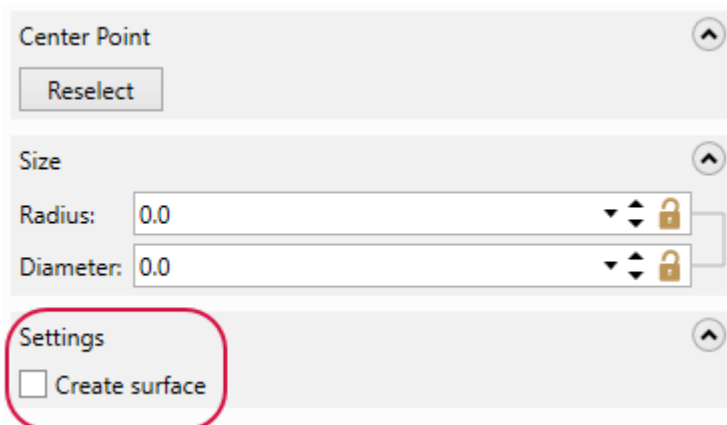
The different functions that were previously bundled in Trim Break Extend are now separate functions located on the **Wireframe** tab in the Modify group. The new functions are:

- Trim to Entities: Trims or extends one entity to another.
- Trim to Point: Breaks geometry at selected points on the entity.
- Modify Length (previously called Extend): Extends or trims the selected entity by an entered distance.

Additional Enhancements

Listed below are additional enhancements made to design functions.

- Mastercam has been updated to use the Parasolids 31 kernel.
- A new, more efficient workflow is available for Trim by Plane and Trim by Surface/Sheet functions. The new workflow eliminates the **Solid Selection** dialog box and defaults to **Line** as the method of specifying the selected plane.
- Line Normal now supports arc selection. When you select an arc, Mastercam creates a line from the arc center in the normal direction.
- When using the Join Entities function, you can now preview the join before selecting **OK**.
- You now have the option to create a surface when creating wireframe with Circle Center Point and Circle Edge Point.



MILL ENHANCEMENTS

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

Accelerated Finishing™ Tools

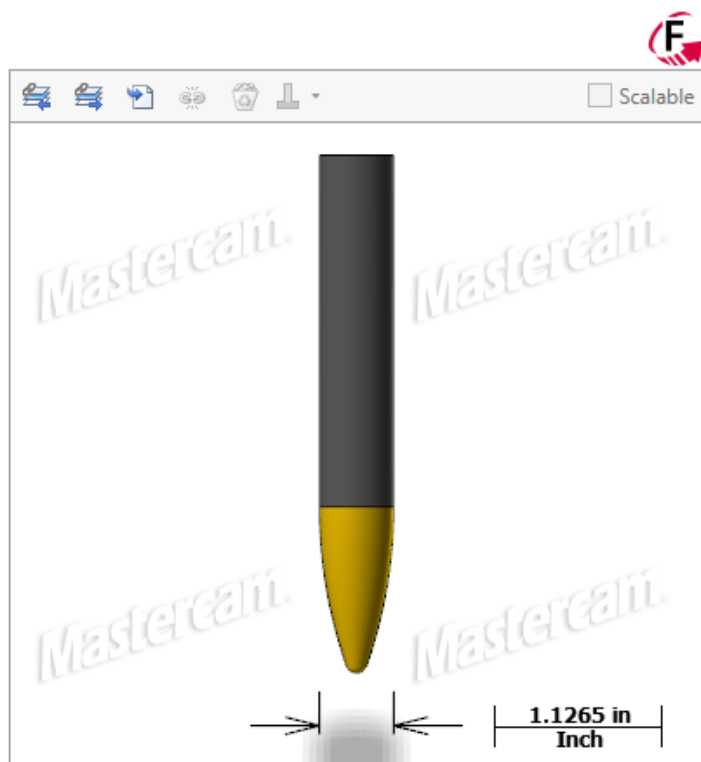
The new Oval form tool and Lens barrel form tool are now available. They are listed in the **Accelerated Finishing** group.

Oval form

Define Oval form

Adjust geometric properties used to define the tool shape.

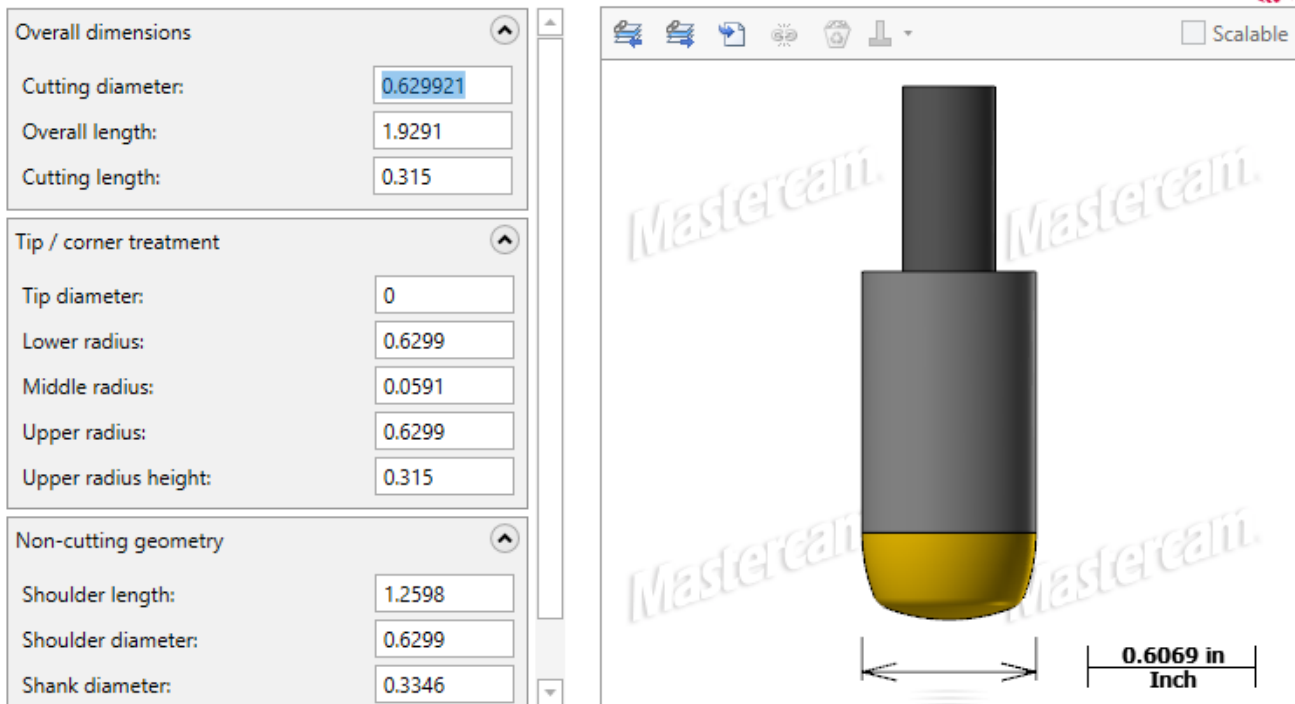
Overall dimensions	
Cutting diameter:	0.5
Overall length:	3.5
Cutting length:	1.125
Tip / corner treatment	
Profile radius:	3.25
Corner radius:	0.0625
Non-cutting geometry	
Shoulder length:	1.125
Shoulder diameter:	0.5
Shank diameter:	0.5
<div style="display: flex; align-items: center;"> <div style="border: 1px solid black; width: 100px; height: 20px; background-color: #add8e6; margin-right: 10px;"></div> <div style="display: flex; align-items: center; margin-right: 10px;"> { } </div> <div style="border: 1px solid black; width: 100px; height: 20px; background-color: #d3d3d3; margin-right: 10px;"></div> <div style="display: flex; align-items: center;"> { } </div> </div>	



Lens barrel form

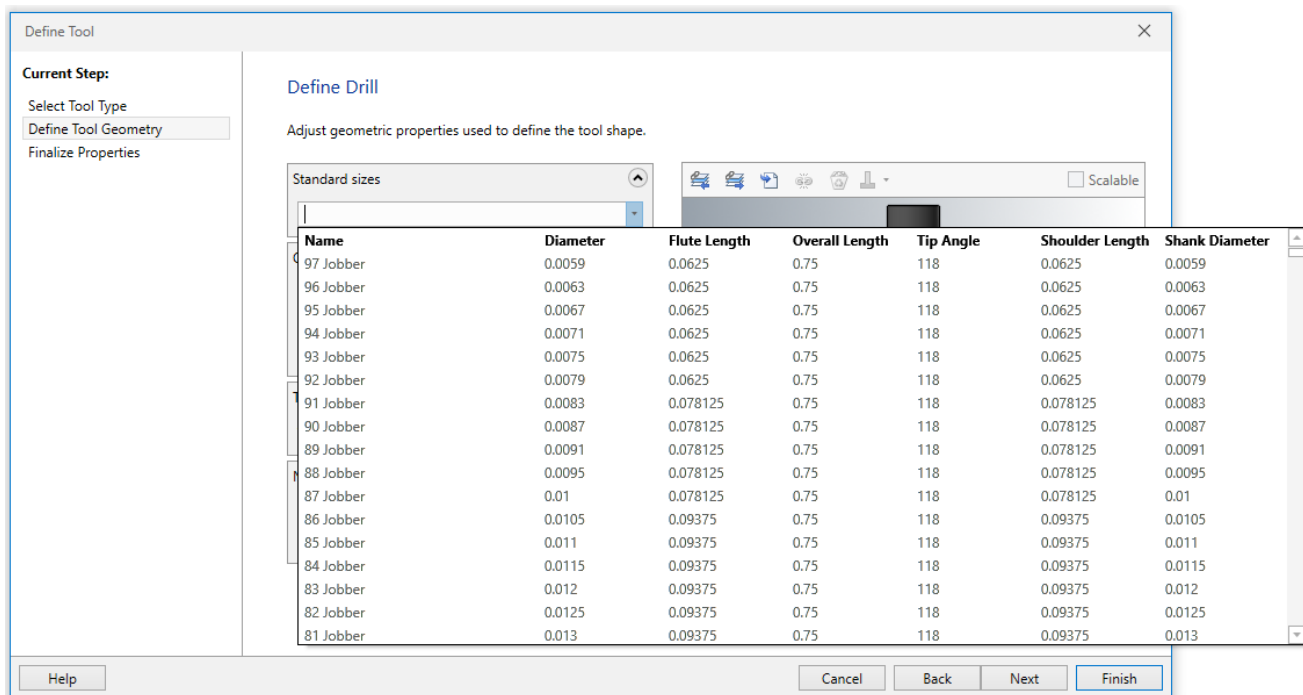
Define Lens barrel form

Adjust geometric properties used to define the tool shape.



Additional Catalog Properties

When creating a tool, you can view a more extensive list of size properties. To access these properties, use the Mill Tool Manager to create a tool, and then select the **Standard Sizes** drop-down.



Tools that display additional properties are Tap, Drill, Spot drill, Center drill, and Brad point drill. This functionality is also available in the stand-alone Tool Manager.

Hole-making Enhancements

NOTE

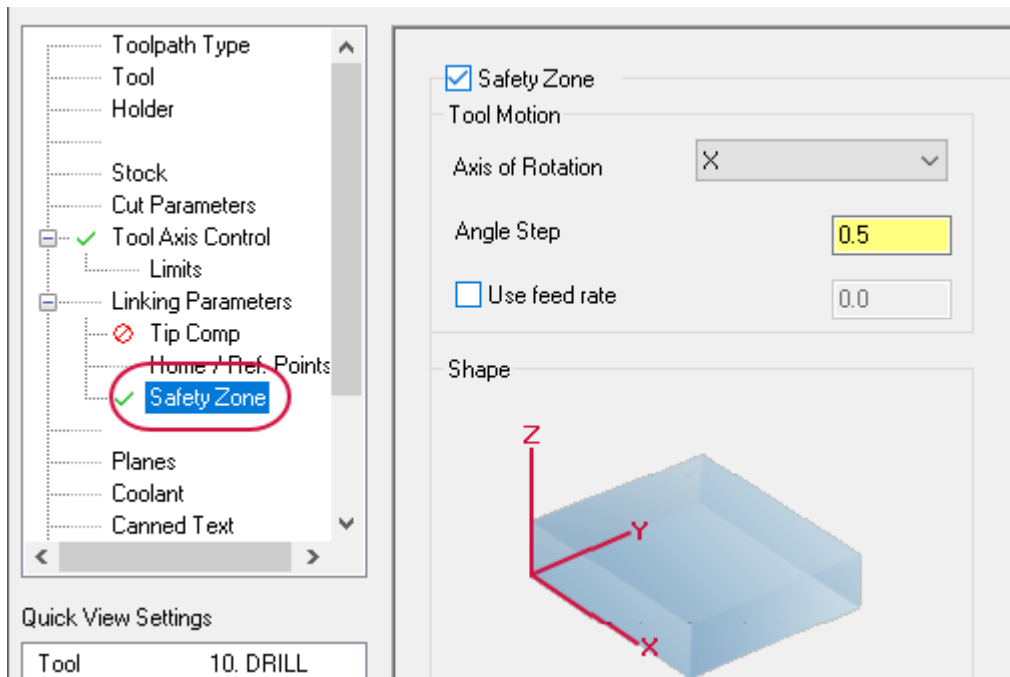
You must have a Multiaxis license to access this functionality.

The Multiaxis Drill and Circle Mill toolpaths have been removed from Mastercam 2020. Helix Bore, Thread Mill, 2D Drill, and 2D Circle Mill toolpaths now have multiaxis capability, accessed through the **Tool Axis Control** page in their respective toolpath dialog boxes.

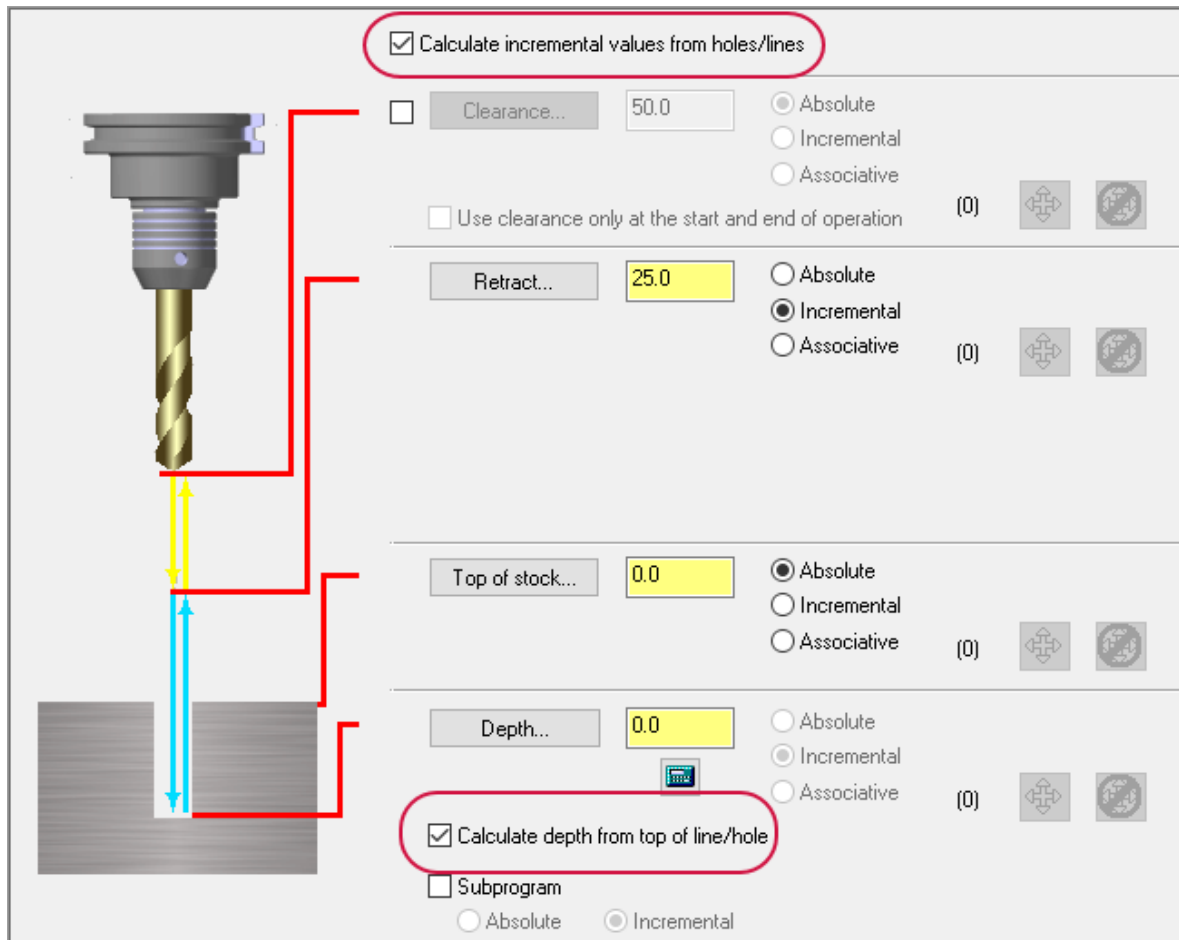


Additionally, if you open a file created in a previous version of Mastercam that had **Entity type** set to **Points/Lines**, the selection now retains the lines and will display them in the **Toolpath Hole Definition** panel.

If you are creating a 4-axis or 5-axis toolpath, you can also access the **Safety Zone** page.



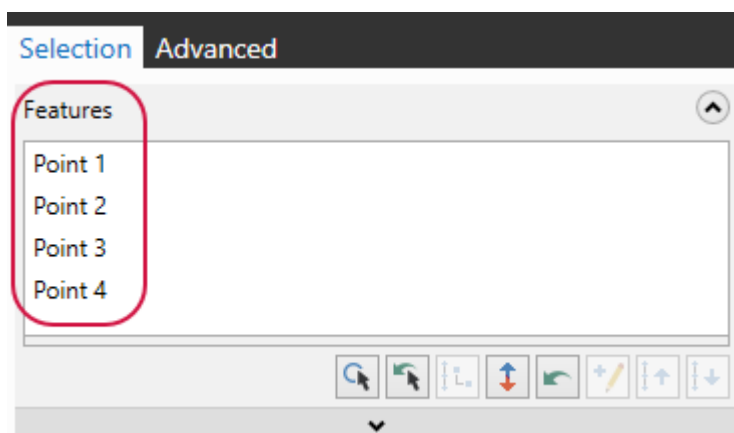
On the **Linking Parameter** page, you can choose to calculate incremental values from holes and lines, or calculate the depth from the top of the holes and lines.



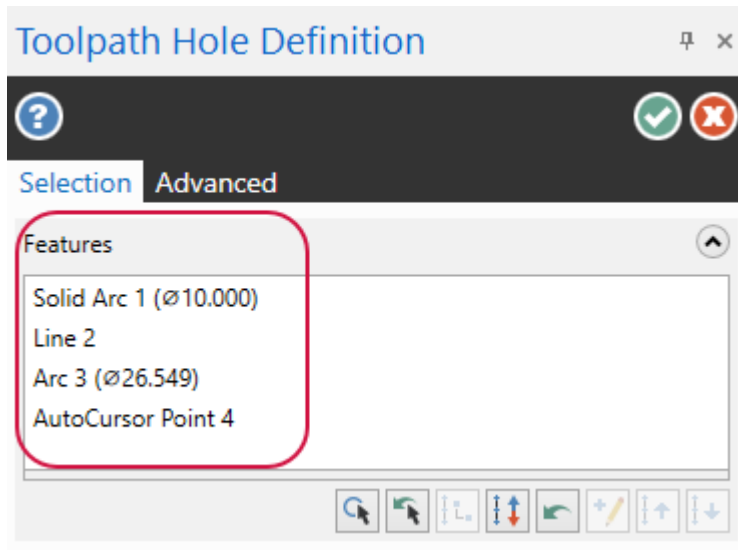
Toolpath Hole Definition

Selection for the **Toolpath Hole Definition** function panel (displayed when selecting point geometry for Mill, Router, and Canned Wire toolpaths) is improved for Mastercam 2020.

The **Features** list now displays more information about the selected points. In Mastercam 2019, the **Features** list only displayed the word **Point** and an incremental number, for example, **Point 4**:

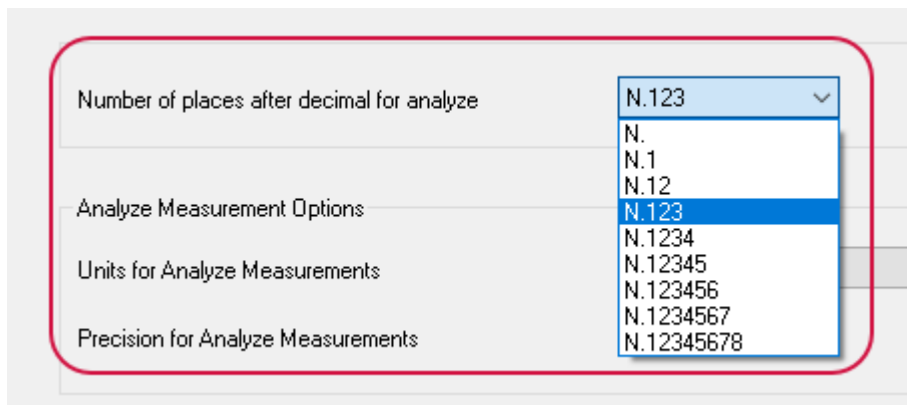


In Mastercam 2020 the **Features** list now displays the type of selection you have made and the diameter of the selected point (if applicable):



Toolpath Hole Definition now supports the selection of solid features and lines, in addition to arcs, points, and AutoCursor positions. When selecting features, you can [**Ctrl+click**] a diameter to select all matching diameters. Use [**Ctrl+Shift+click**] to select only matching diameters and vectors.

The **Analyze** page of the **System Configuration** dialog box allows you to set the number of decimal places displayed for the diameters.

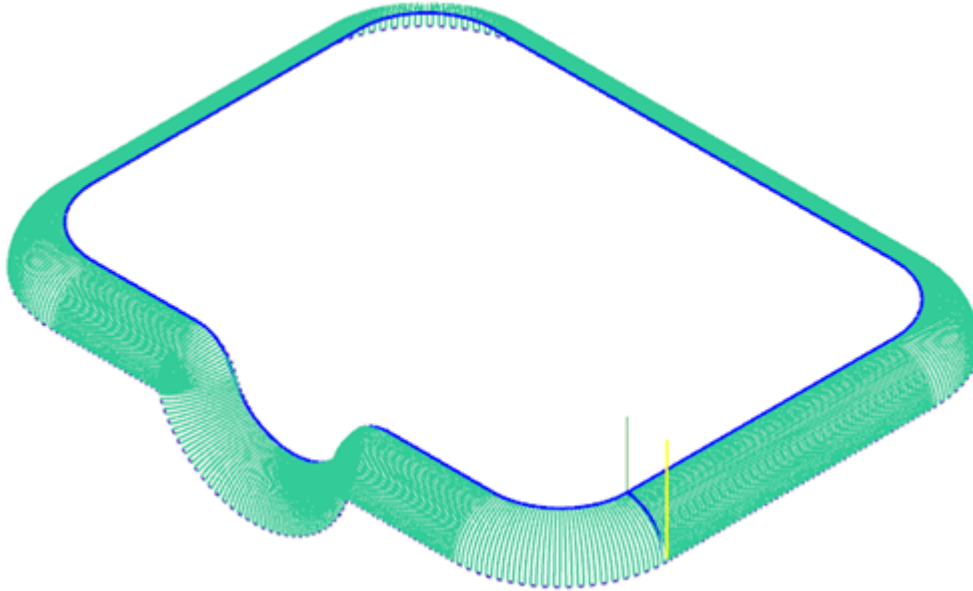


2D Enhancements

Listed below are enhancements made to 2D toolpath types.

Advanced Toolpath Display

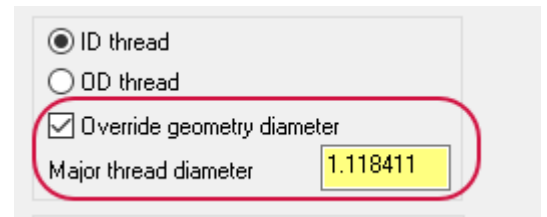
Advanced Toolpath Display, located on the **View** tab, now supports the following wireframe toolpaths:



- Ruled
- Revolved
- Swept 2D
- Swept 3D
- Coons Patch
- Lofted

Override Geometry Diameter

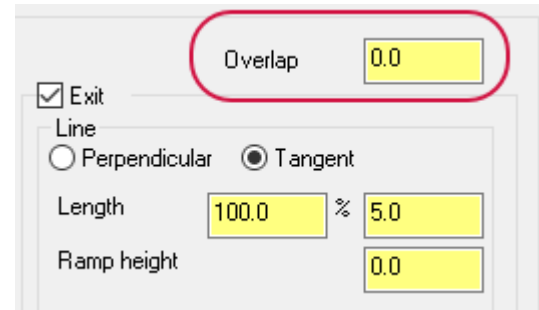
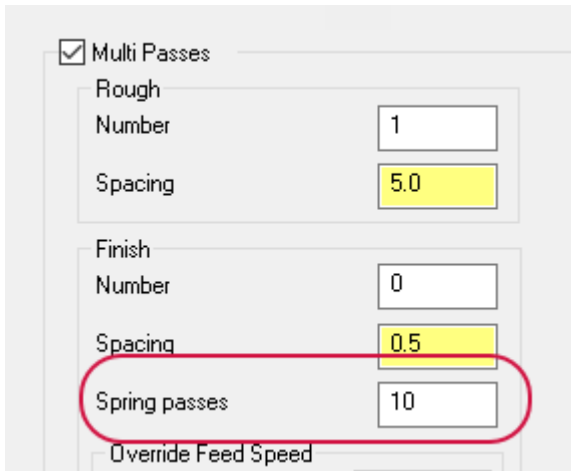
A new option, **Override geometry diameter**, has been added for Helix Bore, Thread Mill, and Circle Mill toolpaths. This new option, located on the **Cut Parameters** page, allows you to override all of the selected geometry's diameter information and forces the output to the specified **Circle diameter** (Helix Bore and Circle Mill) or **Major/Minor thread diameter** (Thread Mill).



This option is only enabled when arcs, solid arcs, or cylindrical features are selected.

Spring Passes and Overlap

Spring passes (set on the **Multi Passes** page) and Overlap (set on the **Lead In/Out** page) now correctly work together in Mastercam 2020 for Contour, Pocket, Model Chamfer, and Dynamic Contour toolpaths. Overlap is applied to the final spring pass.



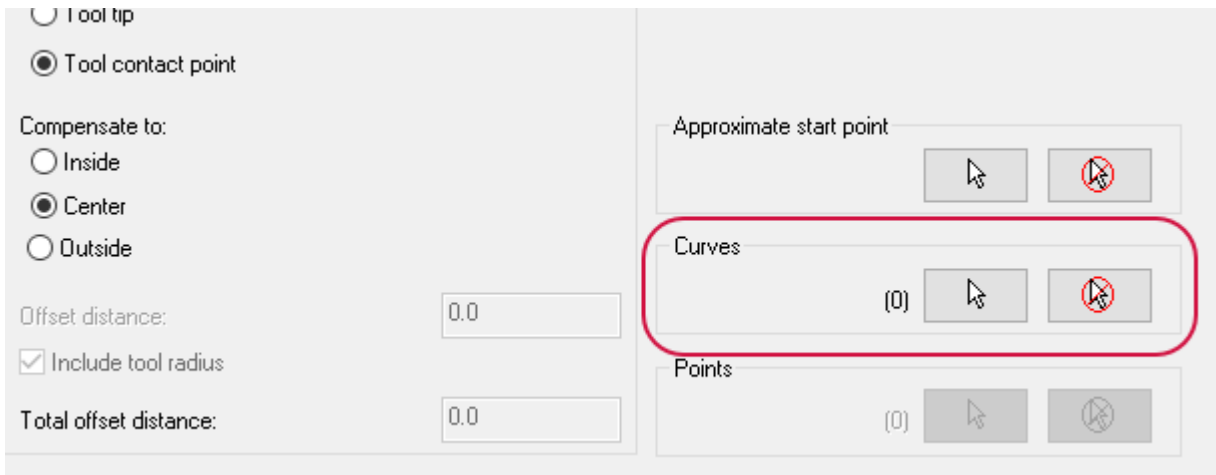
In previous Mastercam releases, Spring passes did not support Overlap and you would receive an error when generating the toolpath.

3D Enhancements

Listed below are enhancements made to 3D toolpath types.

Curve Support

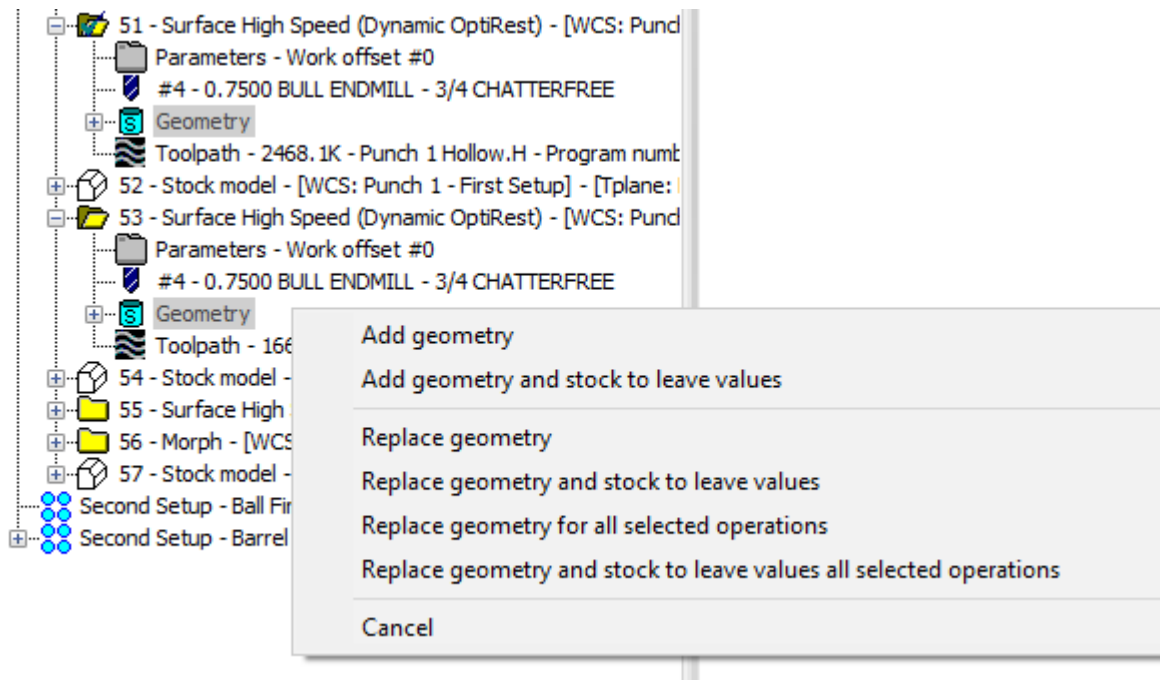
Equal Scallop and Scallop toolpaths now include the ability to select **Curves** on the **Toolpath Control** page.



When selected, the toolpath is created outward using the curves as a starting point.

Dragging and Dropping Toolpath Geometry

When dragging geometry from one 3D HST operation to another in the Toolpaths Manager, the pop-up menu now displays more options for copied geometry.

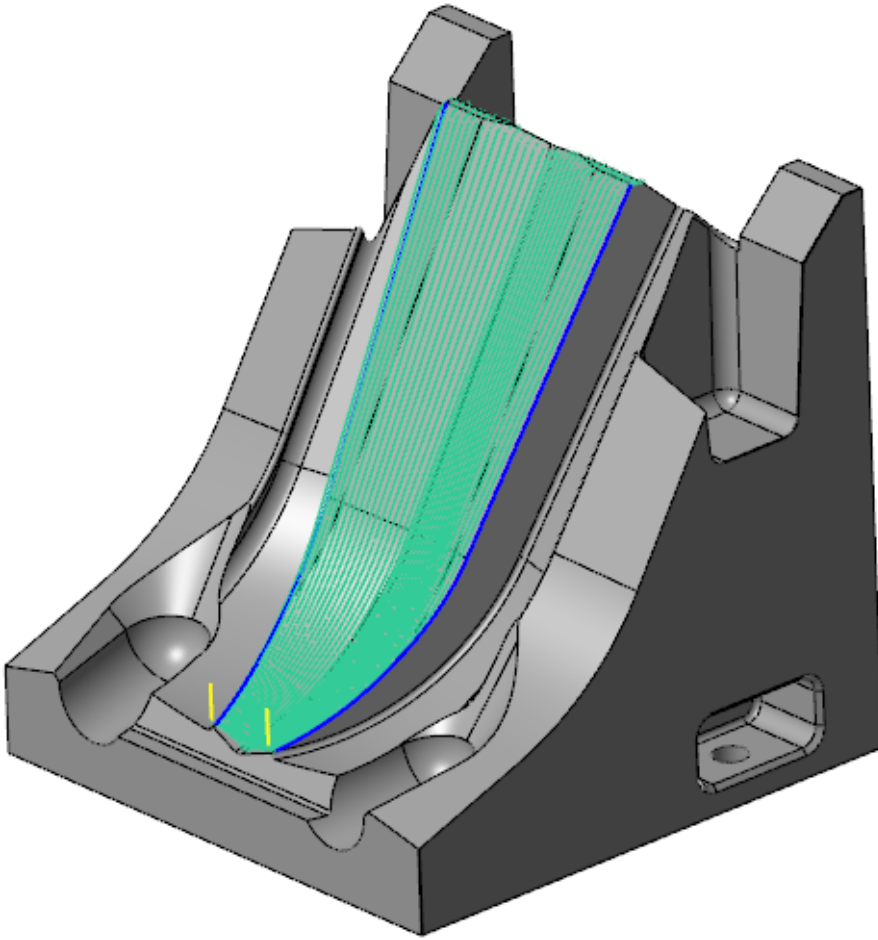


After dragging the geometry, select one of the following options:

- **Add geometry:** Adds the copied geometry to the geometry that exists in the selected operations.
- **Add geometry and stock to leave values:** Adds the copied geometry and their stock to leave values to the geometry that exists in the selected operations.
- **Replace geometry:** Deletes the existing geometry for the selected toolpath, replacing it with the copied geometry.
- **Replace geometry and stock to leave values:** Deletes the existing geometry for the selected toolpath, replacing it with the copied geometry and its stock to leave values.
- **Replace geometry for all selected operations:** Deletes the existing geometry for all selected toolpaths, replacing it with the copied geometry.
- **Replace geometry and stock to leave values all selected operations:** Deletes the existing geometry for all selected toolpaths, replacing it with the copied geometry and its stock to leave values.

New 3D Blend Toolpath

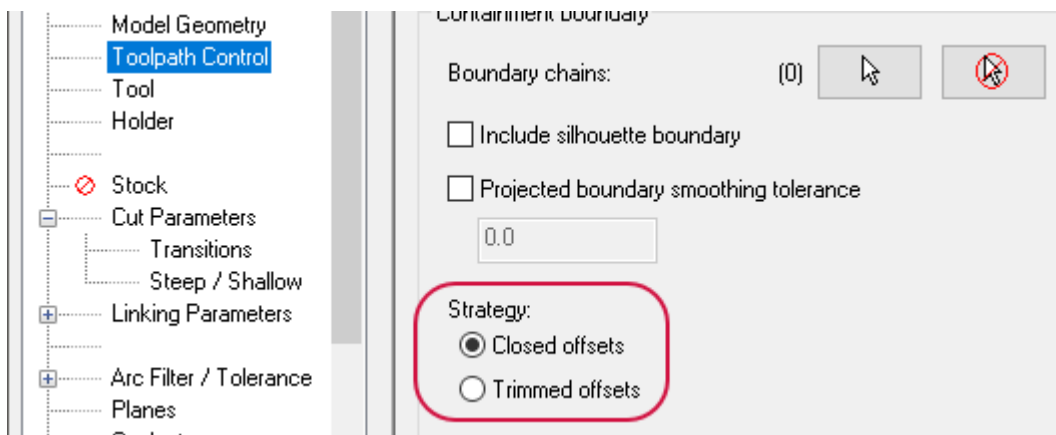
A new toolpath, Blend, has been added to the 3D High Speed suite of toolpaths. Select **Blend** from the **3D** gallery on the **Mill Toolpaths** contextual tab to access this new toolpath.



Blend is based on the Surface Finish Blend toolpath; however it is faster and has a superior quality. Blend also includes benefits such as the use of stock and holder checking. This also brings a blend-style toolpath to the Mill-Turn product.

Offset Strategy

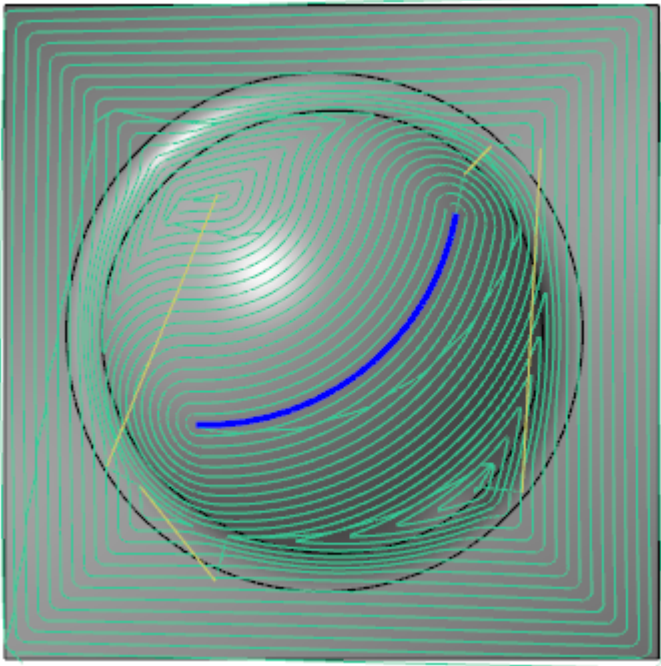
Two new options, **Closed offsets** and **Trimmed offsets**, are available in the **Toolpath Control** page for Equal Scallop and Scallop toolpaths.



Closed offsets use the following options to drive the toolpath:

- **Boundary chains**
- **Curves**
- **Machining geometry** (outer shape only)
- **Z depth** (on the **Steep/Shallow** page)
- **Angle** (on the **Steep/Shallow** page)

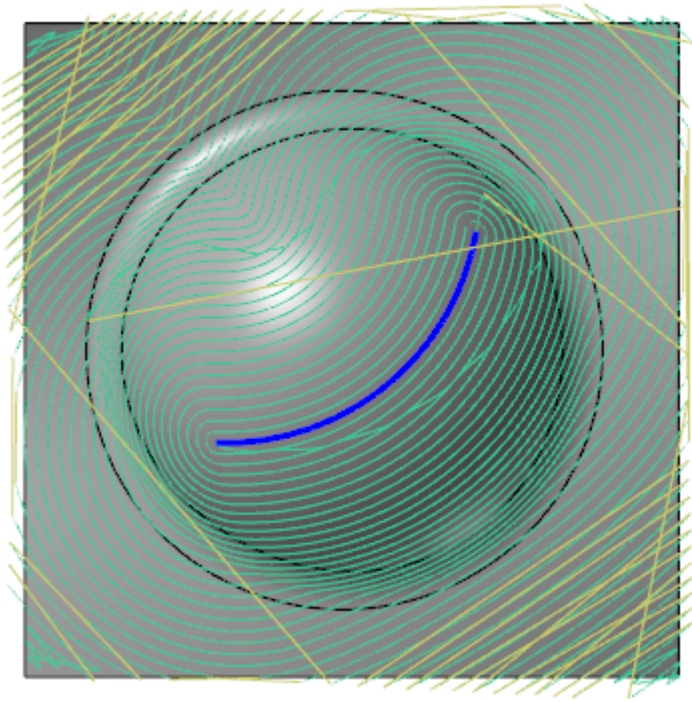
The toolpath is created with closed passes. This yields a traditional Scallop toolpath. The following image shows a toolpath with **Closed offsets** enabled and the blue spline as the selected **Curves**.



Trimmed offsets enables Mastercam to drive the toolpath from the selected **Curves** or the outer shape of the **Machining geometry**. The toolpath is then generated across the entire **Machining geometry** shape, and the resulting motion is trimmed to the following:

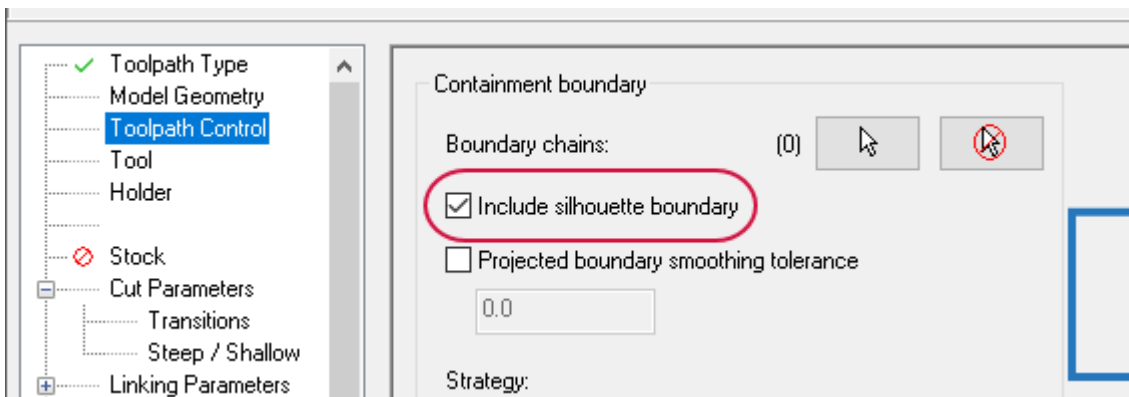
- **Boundary chains**
- **Angle** (on the **Steep/Shallow** page)
- **Z depth** (on the **Steep/Shallow** page)

The following image shows a toolpath with **Trimmed offset** enabled and the blue spline as the selected **Curves**.



Silhouette Boundary

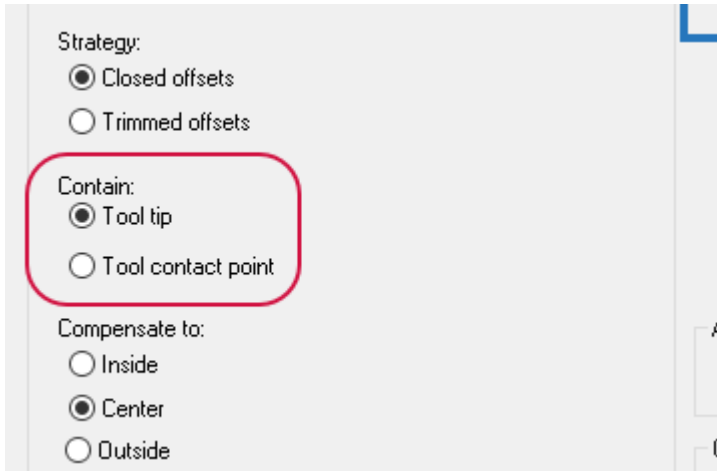
A new option, **Include silhouette boundary**, has been added to the **Toolpath Control** page.



When you select this option, Mastercam will create a silhouette boundary around the selected **Machining geometry**. This will be used as a containment boundary in addition to any **Boundary chains** that you have selected. Mastercam uses the **Projected boundary smoothing tolerance** option, the **Contain** options, and the **Compensate to** options when generating the silhouette boundaries. These options are also found on the **Toolpath Control** page.

Tool Containment

Tool containment options are now available for Equal Scallop and Hybrid toolpaths on the **Toolpath Control** page. Select one of the following:

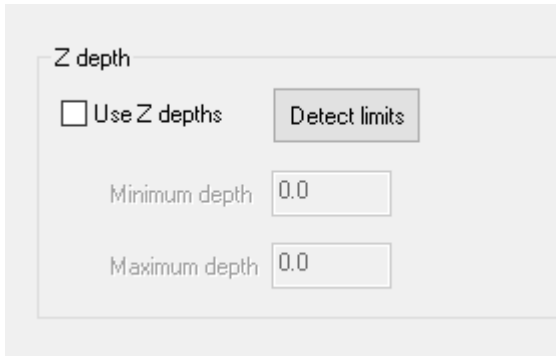


- **Tool tip:** Confines the tool center inside the containment boundary.
- **Tool contact point:** Confines the tool contact points inside the containment boundary. The tool center may run outside the selected boundary, but the contact point of the tool will not.

Z Depth Enhancements

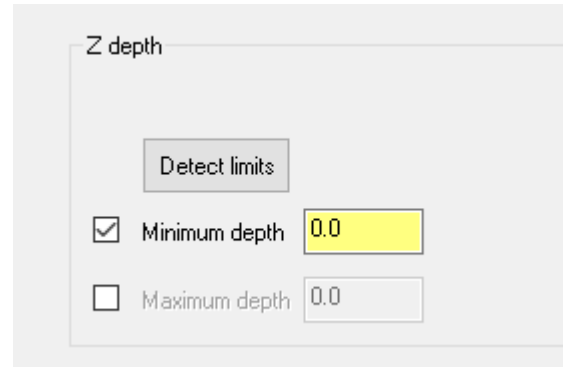
The **Steep/Shallow** page has new Z depth options. The **Use Z depths** checkbox has been replaced by separate checkboxes for **Minimum depth** and **Maximum depth**. This allows you to turn them on separately so you can make adjustments to only one option.

Mastercam 2019



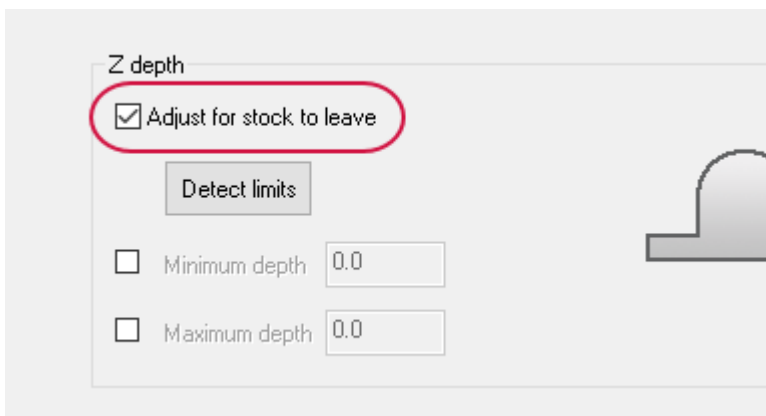
The Mastercam 2019 Z depth dialog box shows a single checkbox labeled "Use Z depths" which is currently unchecked. To the right of this checkbox is a "Detect limits" button. Below the checkbox are two input fields: "Minimum depth" and "Maximum depth", both containing the value "0.0".

Mastercam 2020



The Mastercam 2020 Z depth dialog box has been redesigned. It features a "Detect limits" button at the top. Below it are two separate checkboxes: "Minimum depth" (checked) and "Maximum depth" (unchecked). Each checkbox is followed by an input field containing "0.0". The "Minimum depth" input field is highlighted in yellow.

The **Steep/Shallow** page for Waterline, Hybrid, Area Roughing, and Dynamic OptiRough toolpaths has a new option, **Adjust for stock to leave**.



This image shows the Mastercam 2020 Z depth dialog box with an additional option. A red oval highlights a new checkbox labeled "Adjust for stock to leave", which is checked. Below this checkbox is a "Detect limits" button. At the bottom are the "Minimum depth" and "Maximum depth" checkboxes, both unchecked, with input fields containing "0.0". A small 3D model of a part is visible on the right side of the dialog.

When you select **Adjust for stock to leave**, Mastercam adjusts the toolpath based on the values entered in the **Floor stock** column for **Machining geometry** in the **Model Geometry** page by the following:

- All positive values: Mastercam uses the largest (most positive) value to adjust both **Minimum depth** and **Maximum depth**.
- All negative values: Mastercam uses the largest (most negative) value to adjust both **Minimum depth** and **Maximum depth**.
- Both positive and negative values: Mastercam uses the most positive Z value to adjust the **Minimum depth** and the most negative Z value to adjust the **Maximum depth**.

Multiaxis Enhancements

Listed below are enhancements made to Multiaxis toolpath types.

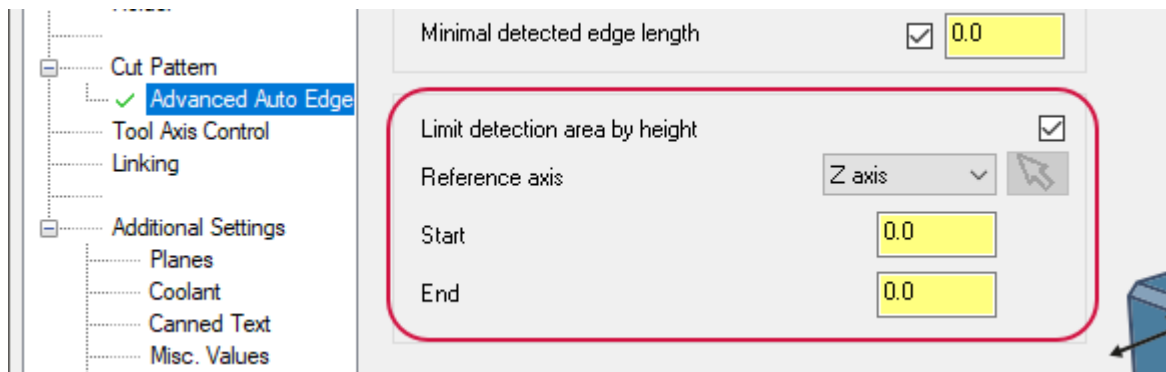
Deburr

The Deburr toolpath includes the following enhancements:

On the **Cut Pattern** page, you now have an option to set a **Climb** or **Conventional** cutting direction.

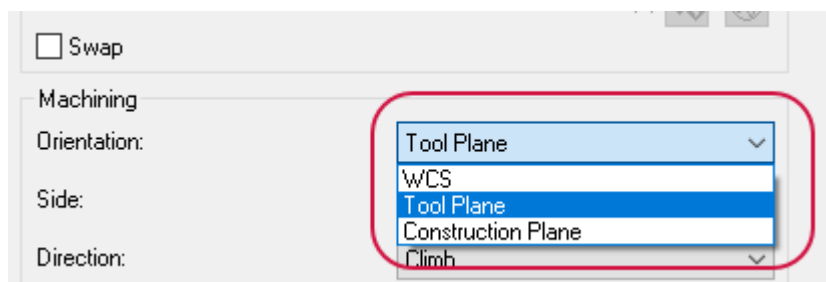
- **Climb** cuts in one direction with the tool rotating in the opposite direction of the tool motion.
- **Conventional** cuts in one direction with the tool rotating in the same direction as the tool motion.

On the **Advanced Auto Edge Detection** page, you can now limit the detection area by a set height or by selecting the **Limit detection area by height** option and setting the necessary parameters.



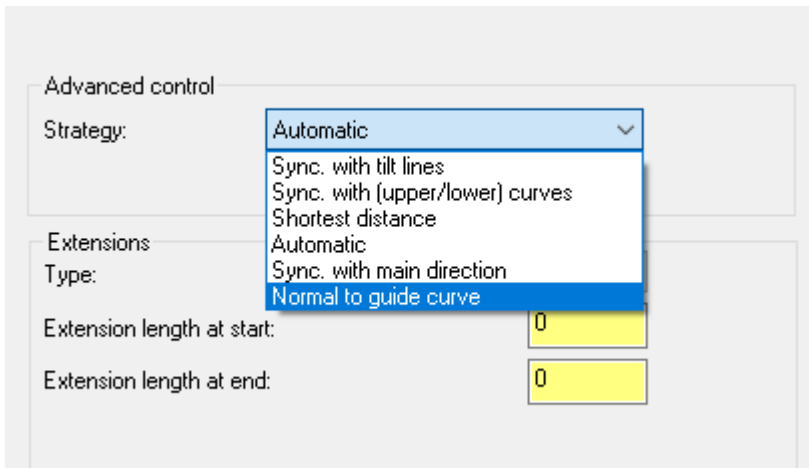
Swarf Milling

A new drop-down, **Orientation**, is now available on the **Cut Pattern** page for Swarf Milling toolpaths. This option is helpful in the cases where you need to realign the operation's orientation from world Z to the **WCS**, **Tool Plane**, or **Construction Plane**.



This new option allows you to choose the plane that you want to orient the operation to. Previously, you modified the orientation by using the **Planes** page in the **Multiaxis Toolpath - Swarf Milling** dialog box.

The **Normal to guide curve** option is also available under the **Strategy** drop-down on the **Cut Pattern** page. By selecting this option, all edges are automatically obtained from the selected part surface when cutting.



Additional Enhancements

- Mastercam 2020 includes processing speed improvements to the Dynamic Mill toolpath, 3D high speed toolpath linking steps, and the 5-axis Curve toolpath.
- Mastercam 2020 includes improvements to the **Projected boundary smoothing tolerance** option on the **Toolpath Control** page.
- You can now have multiple **Air region** chains, instead of only one per **Machining region** for Dynamic Mill and Area Mill toolpaths.

LATHE ENHANCEMENTS

Listed below are Lathe enhancements made to Mastercam 2020.

3D Tooling

3D tooling, which was introduced in Mastercam 2019, has many usability improvements.

- New stop-on-contact functionality when mating insert to holder.
- New 3D tool icons in tool lists to help distinguish between 3D and non-3D tools in the Lathe Tool Manager.



- Auto-populating compensation point when defining 3D tools.
- New tabs to display library inserts and holders in the Lathe Tool Manager.



Insert Designer

Mastercam 2020 allows you to build inserts using the new **Insert Designer** function panel. Insert Designer supports ANSI and ISO parametric definitions or custom inserts based on solid models.

Insert Designer

? [Close] [Maximize]

Basic | Advanced

Insert

Name: CNMG432

Type: ☒ General turning
☐ Threading
☐ Grooving and parting

Units

☒ Inch
☐ Metric

Definition

☒ ANSI
☐ ISO
☐ Custom

Dimensions

Code:

Shape: C (80 degree diamond)

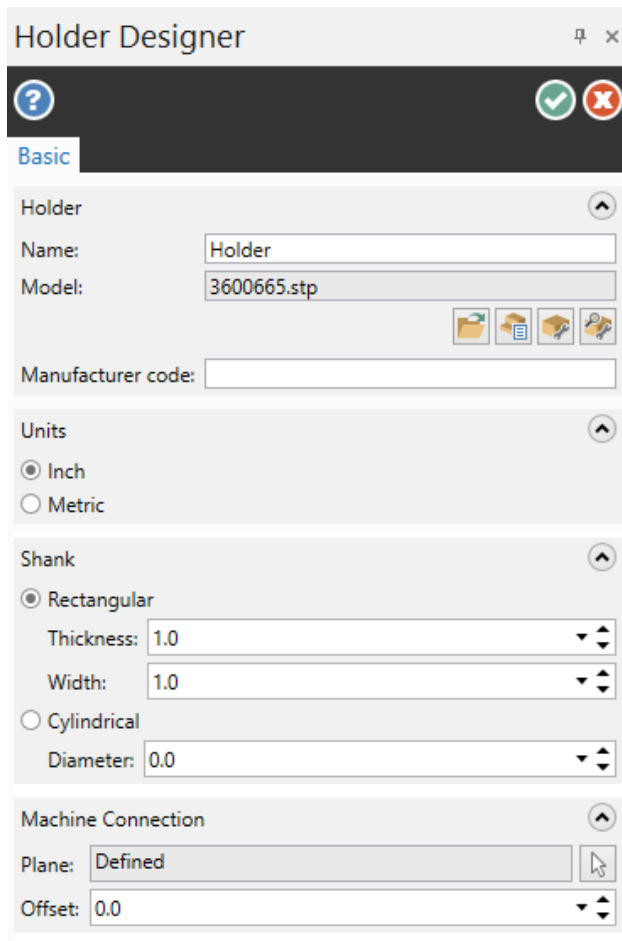
Relief angle: 0.0

Cross section: G

Inscribed circle diameter:	0.5
Thickness:	0.1875
Corner radius:	0.03125

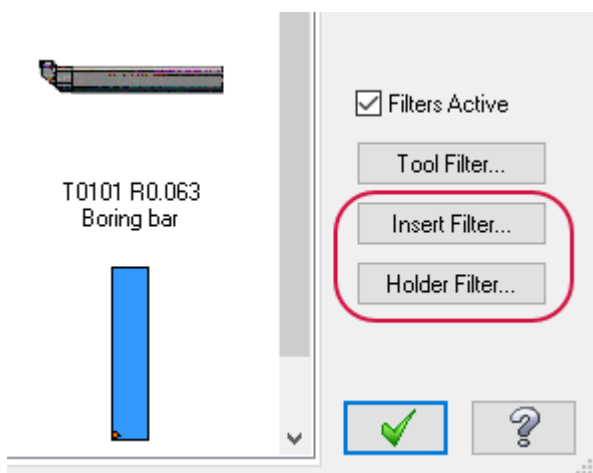
Holder Designer

Also in Mastercam 2020, you can build holders using the new **Holder Designer** function panel. Holder Designer supports custom holders from solid models.



Filters for Inserts and Holders

You can now filter inserts and holders by attributes. In the Lathe Tool Manager, select either **Insert Filter** or **Holder Filter**.

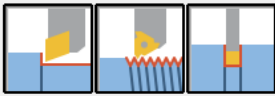


Use the dialog box to filter your selections by selecting attributes you are looking for. After clicking **OK**, only items that match that criteria will display in the Tool Manager.

Lathe Insert Filter dialog box:

Lathe Insert Filter [X]

Insert Type



All None

Parameters

Units:

☒ Inch
☐ Metric

☐ Corner radius:

Minimum: 0.0 inch
Maximum: 0.0 inch

Insert Materials

☒ Carbide ☒ CBN
☒ Cermet ☒ Diamond
☒ Ceramic ☒ Unknown

All None

Copy job material

Cutting Side

☒ Right hand inserts
☒ Left hand inserts

General Turning

Type:

☒ ANSI
☒ ISO
☒ Custom

Shape:

☐ T (triangle)
☐ R (round)
☒ C (80 degree diamond)
☐ D (55 degree diamond)
☐ E (75 degree diamond)
☐ M (86 degree diamond)
☐ V (35 degree diamond)
☐ A (85 degree parallelogram)
☐ B (82 degree parallelogram)
☐ K (55 degree parallelogram)
☐ W (80 degree trigon)
☐ S (square)
☐ H (hexagon)
☐ L (rectangle)
☐ O (octagon)
☐ P (pentagon)

All None

Threading

Type:

☒ Parametric
☒ Custom

Style:

☒ 60 degree Top Notch
☒ Acme Top Notch
☒ Buttress Top Notch
☒ 60 degree Laydown
☒ Acme Laydown

All None

Grooving and Parting

☐ Width:

Minimum: 0.0 inch
Maximum: 0.0 inch

Type:

☒ Parametric
☒ Custom

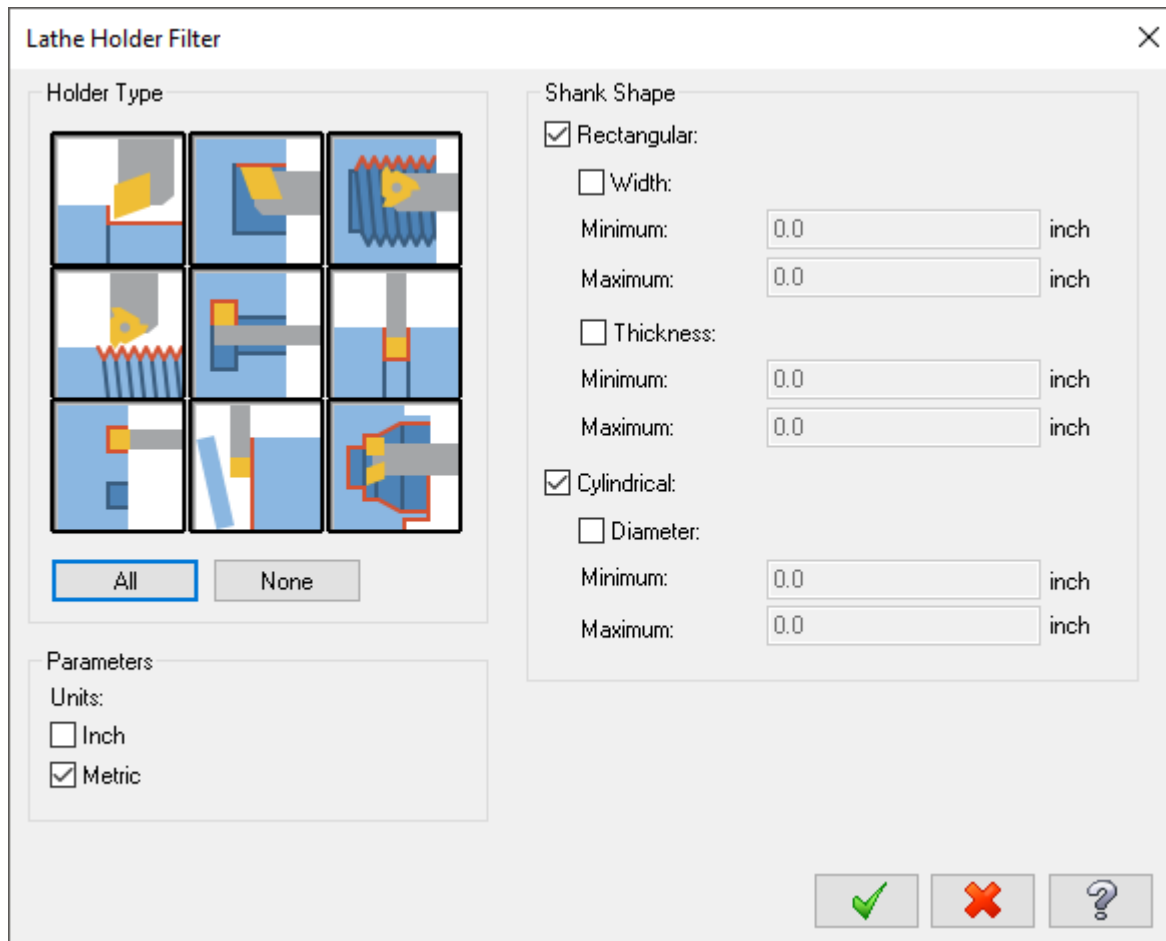
Style:

☒ Double End (square)
☒ Double End (radius)
☐ Double End (40 degree V)
☐ Double End (45 degree chamfer)
☐ Single End (square)
☐ Single End (chamfer)
☐ Single End (round)
☐ Single End (radius)

All None

[OK] [Cancel] [Help]

Lathe Holder Filter dialog box:



Improved Chuck Jaw Definitions

Mastercam 2020 includes improvements to chuck jaw definitions. Parametric jaw definitions are more powerful and flexible. The following enhancements have been added:

- Support for pie-shaped jaws in addition to rectangular-shaped jaws.
- Define as many steps as required, set the dimensions of each jaw individually, and add a radius to the step.
- Change the mounting orientation with easily reversible steps.

- Live preview showing the overall shape of the jaw and its total dimensions.

Parameters
Geometry

Geometry
Parametric

Color
109

Chord tolerance
0.025

Translucency
Solid
Transparent

Shape

☒ Rectangular

Thickness
1.1024

Outer edge radius
0.0

☐ Pie

Sweep angle (degrees)
0.0

Minimum radius
0.0

Wedge offset
0.0

Summary

Total height
2.5197

Total width
1.6929

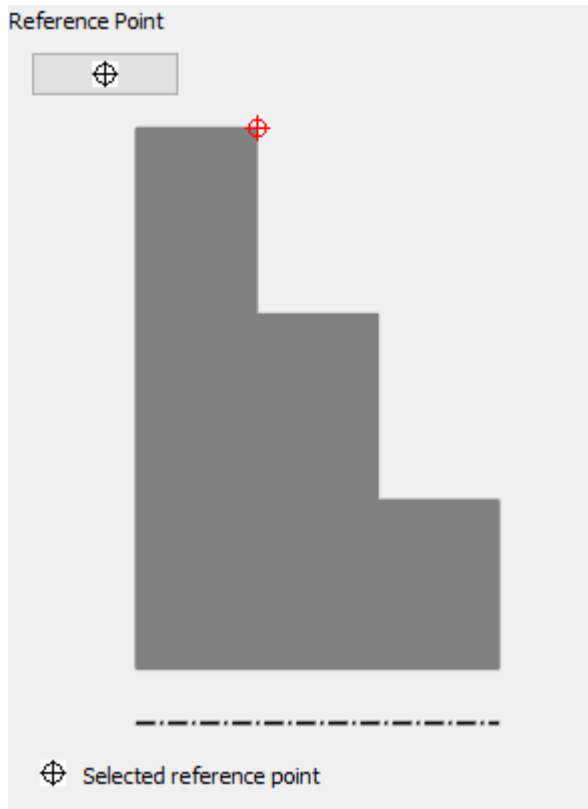
Number of steps
3

Preview

Steps

#	Width	Height	Radius
1	1.6929	0.7874	0.0000
2	1.1286	0.8661	0.0000
3	0.5643	0.8661	0.0000

Previous versions of Mastercam limited you to a fixed set of seven possible reference positions (four outside diameter, three inside diameter) to identify the location where the jaws clamp the part. In Mastercam 2020, you can select any point on the profile. The reference point is included in the live preview of the jaws.



MILL-TURN ENHANCEMENTS

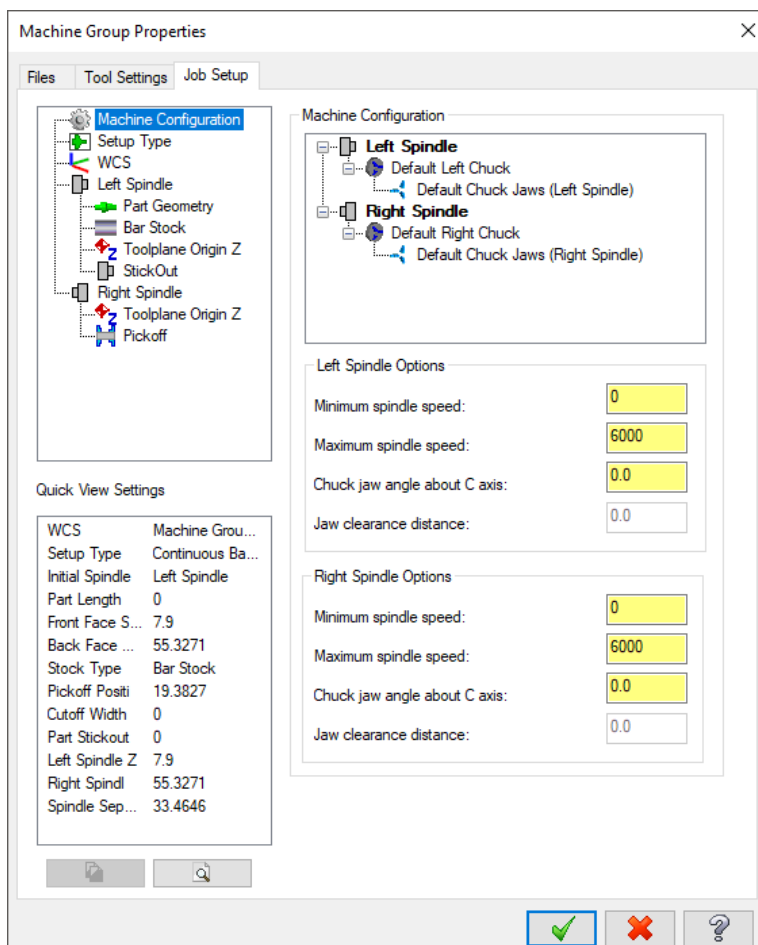
Listed below are Mill-Turn enhancements made to Mastercam 2020.

Improved Machine Configuration Workflow

Mastercam 2020 gives you more power for setting up machining jobs and configuring **.machine** files for Mill-Turn. A suite of new tools works together to significantly improve and streamline the Mill-Turn workflow.

New Machine Configuration Page

The Job Setup workflow has been enhanced with the addition of the new **Machine Configuration** page. Use this page to select the chucks and jaws to be loaded on the machine.



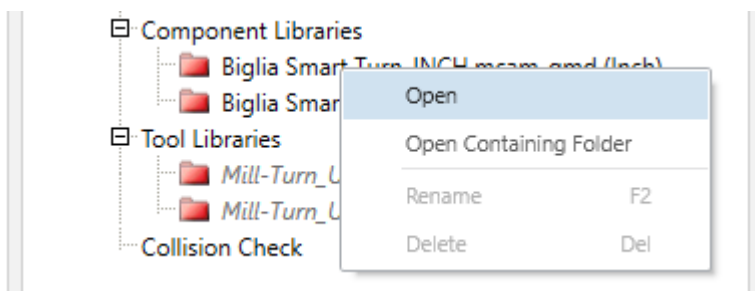
- Previous versions of Mastercam required you define chuck jaws for each part. In Mastercam 2020, you can save chuck jaws to a component library and re-use them in multiple parts. You can define as many sets of chuck jaws as necessary.
- You can also define different chucks and save them to a component library so they can be managed together with jaws. In previous versions of Mastercam, you were limited to the chuck that was built into the **.machine** file

and only machine developers could make changes to the chuck. Mastercam 2020 allows you to swap out chucks for each part by selecting them from their library.

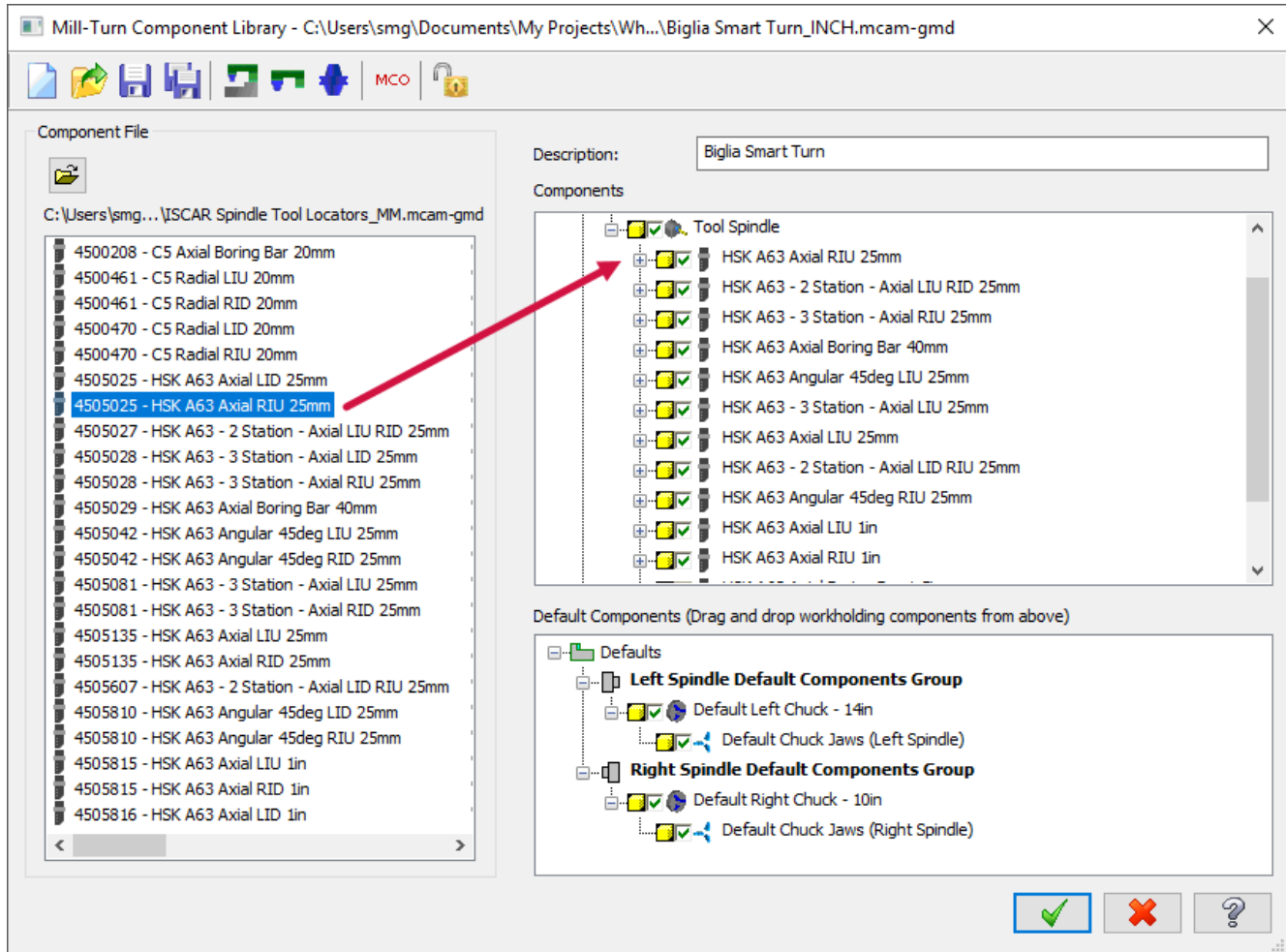
- You can select which chucks and jaws to use as defaults.
- The **Machine Configuration** page also lets you enter minimum and maximum spindle speeds directly as part of the Job Setup. Previously, these values were saved in the machine definition and needed to be edited there. Moving these settings to the **Job Setup** tab makes it more convenient for you to enter values.

New Tool Locator and Component Library Functionality

The workflow for managing tool locators and component libraries has been reworked and is now available to all Mill-Turn users. Now any Mill-Turn user can create or edit tool locators and manage their tool locator libraries, instead of being dependent on your Reseller or machine developer. You can open the component library by right-clicking on the **.mcam-gmd** file in the Machine Explorer in Code Expert and selecting **Open** or by double-clicking it.



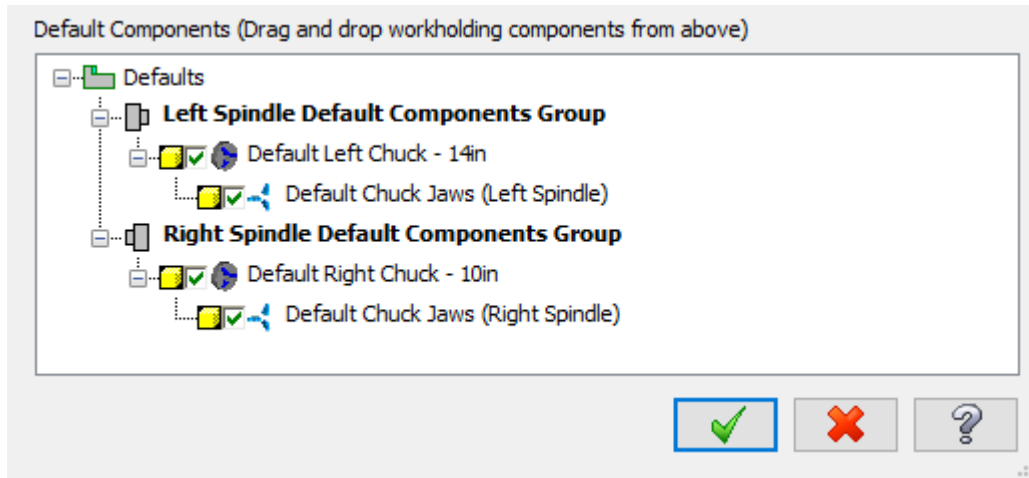
Mastercam opens the library and displays it in the new **Mill-Turn Component Library** dialog box:



You can use the new interface to easily to do any of the following:

- Import tool locators from other libraries.
- Create new or custom libraries.
- Create new tool stations—including half-index tool stations—or edit existing ones.
- Edit or shift the tool mounting locations on the block.
- Edit or replace the solid model used to simulate the locator.
- Create completely new tool locators as required.

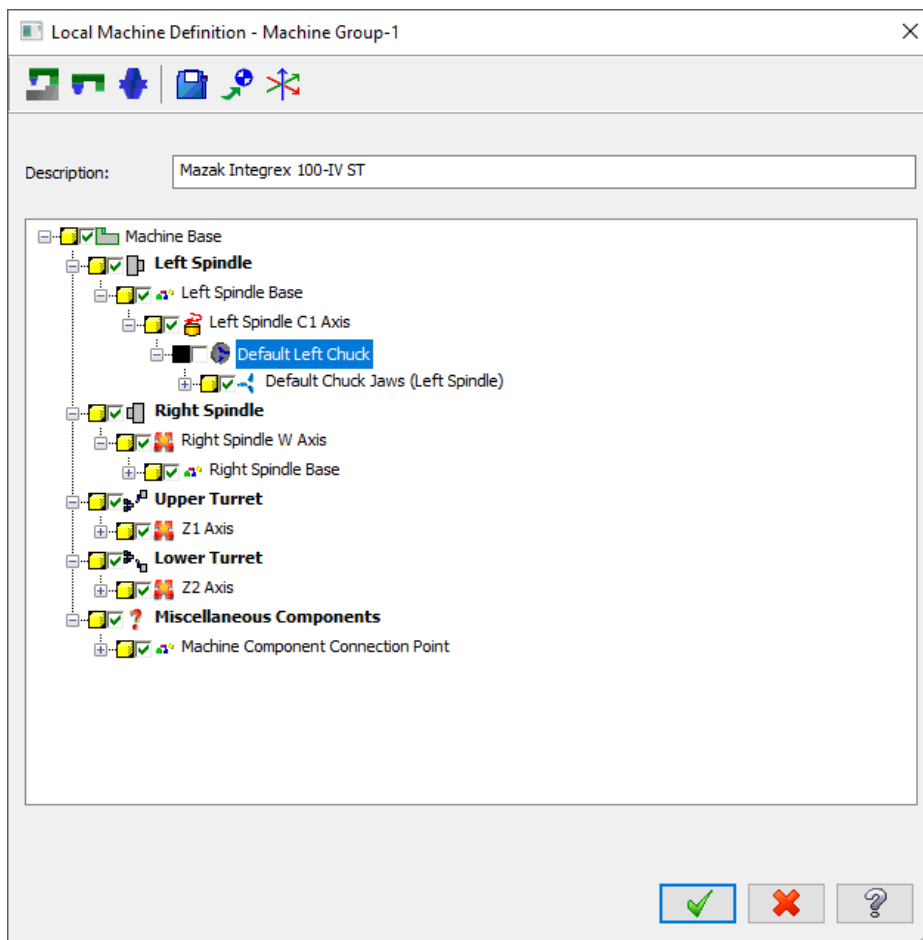
Many tool locator libraries are already available on Mastercam's Tech Exchange (community.mastercam.com/TechExchange) and are available to any Mill-Turn users. The component libraries themselves have been enhanced so that users can store chucks and chuck jaws in them. In addition, you can select the chuck and jaws that will be used as the default for each spindle.



This means you can easily store turret tool locators, spindle tool locators, chucks, and chuck jaws in the same component library. The new **Machine Configuration** page ("[New Machine Configuration Page](#)" on page 73) exposes these components to you during Job Setup, allowing for flexibility in configuring the machines for each job.

New Interface for Local Machine Definitions

Mastercam 2020 includes a new simplified interface for working with the local copy of a Mill-Turn machine definition.



The new interface removes functions that are not relevant to editing the local copy of the machine definition, and focuses your attention only on tasks that are appropriate for job-specific changes. It also indicates that you are working on a local copy of the machine definition instead of the master copy.

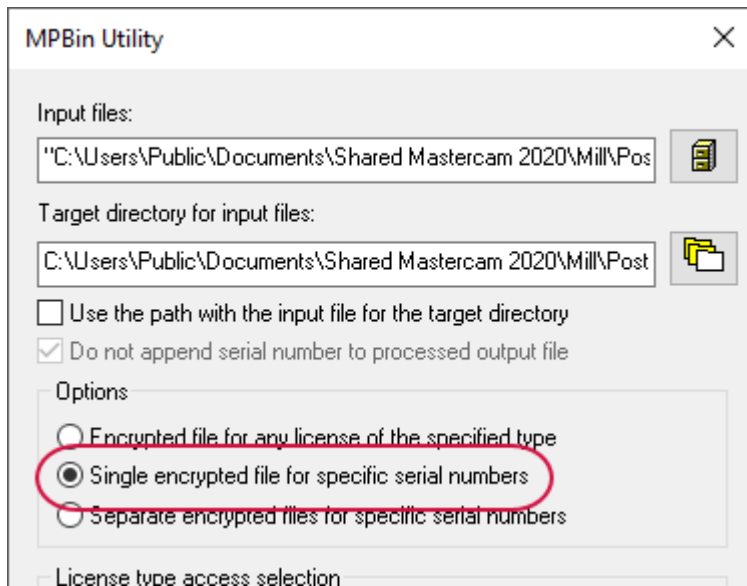
POST ENHANCEMENTS

Listed below are enhancements made to posts.

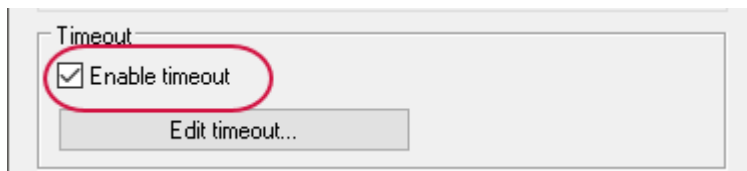
MPBin Enhancements

For Mastercam 2020, the MPBINX utility has been renamed MPBin. In addition, it includes several new features.

MPBin now lets you create a single PSB file that can be used for multiple SIMs or license numbers. Previously, MPBin would output a separate PSB for each SIM. You can still choose to output individual PSB files.



MPBin now includes the ability to enter an expiration date for the binned (encrypted) post.



MP Post Enhancements

Improved language support for post text

Mastercam 2018 introduced the ability to localize post text by associating each section of post text with a Language-Culture code (for example, en-US for United States English). The Language-Culture code is encapsulated in a `<language>` tag (for example, `<language>en-US</language>`). However, this has created confusion when the post does not include a post text section with a Language-Culture code that matches the user's workstation. In these cases, Mastercam 2018 and Mastercam 2019 display system default post text instead of the intended post text.

Mastercam 2020 includes more robust error handling so that it is more likely that a user will see useful post text if the proper language tag is not found. When a user loads a machine and control definition in Mastercam 2020, Mastercam searches for post text in this order:

- A post text section that matches the current control definition name plus the Language Culture Name from the Mastercam workstation.
- A DEFAULT post text section of the proper product type, with the proper Language Culture Name.
- An en-US post text section.
- The first post text section of any language.
- System post text.

For example, if a user in France loads a Generic Haas 4-axis Mill post, Mastercam will look for the following post text section:

```
<control>
  <control_label>CTRL_MILL|GENERIC HAAS 4X MILL</control_label>
  <language>fr-FR</language>
  <misc_integers>
    <misc_1>
```

If this section is not found, Mastercam looks for a DEFAULT post text section:

```
<control>
  <control_label>CTRL_MILL|DEFAULT</control_label>
  <language>fr-FR</language>
  <misc_integers>
    <misc_1>
      <text>Work Coords [0-1=G92, 2=G54'S, 3=G52, 4=both]</text>
      <value>2</value>
    </misc_1>
```

If no DEFAULT post text section exists, Mastercam will use this post text section:

```
<control>
  <control_label>CTRL_MILL|GENERIC HAAS 4X MILL</control_label>
  <language>en-US</language>
  <misc_integers>
    <misc_1>
```

If no en-US section exists, Mastercam will use the first post text section it finds. If no post text section of any language exists in the post, Mastercam will use default system post text.

Improved support for UTF-8 encoding for MP posts

Beginning in Mastercam 2019, MP posts (*.pst, *.mcpost, and *.set files) are encoded as UTF-8 text format. This improves support for post text in languages other than US English. However, Code Expert in Mastercam 2019 did not enforce this. This means that in Mastercam 2019, if your Code Expert was set to use a different character encoding scheme and you opened an MP post, it would save the post with the non-UTF-8 encoding.

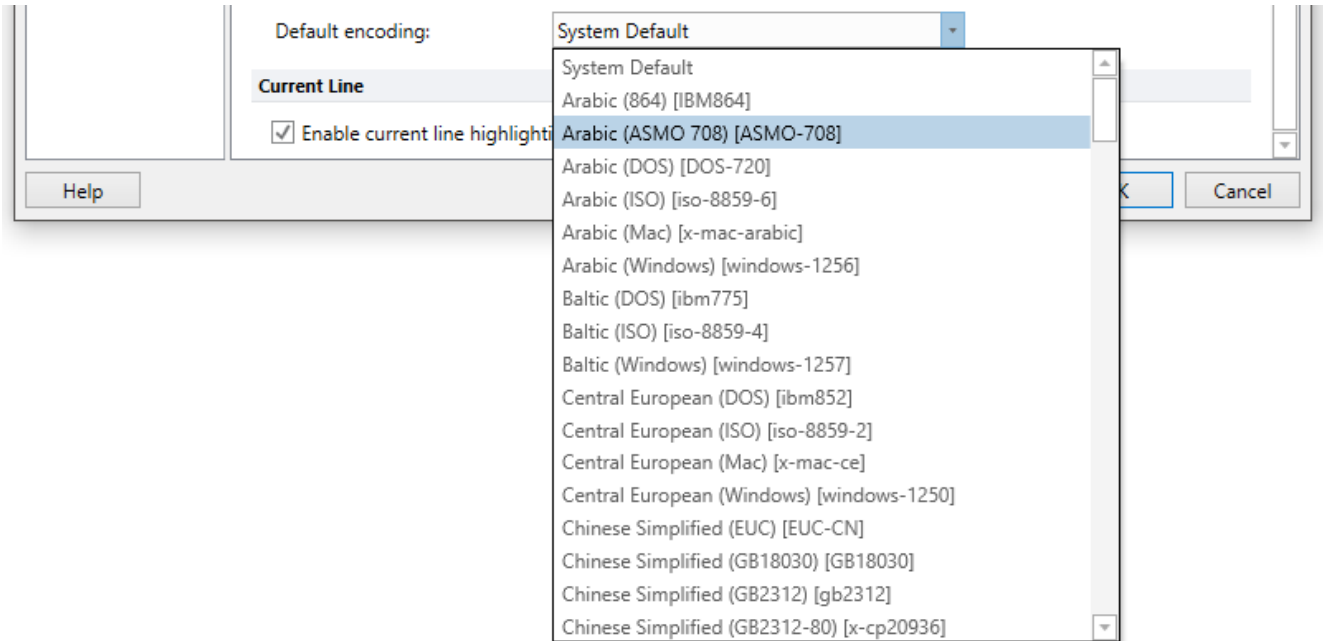
For Mastercam 2020, Code Expert will now respect and enforce the UTF-8 requirement for MP posts.

- If you open a Mastercam 2020 post in Code Expert and the post is not in UTF-8 format, Code Expert will open and save it as a UTF-8 file.
- If Code Expert is set to use a different character encoding method than UTF-8 and you open a Mastercam 2020 post, it will still open and save it as a UTF-8 file, instead of the encoding scheme selected in Application Options as shown above.

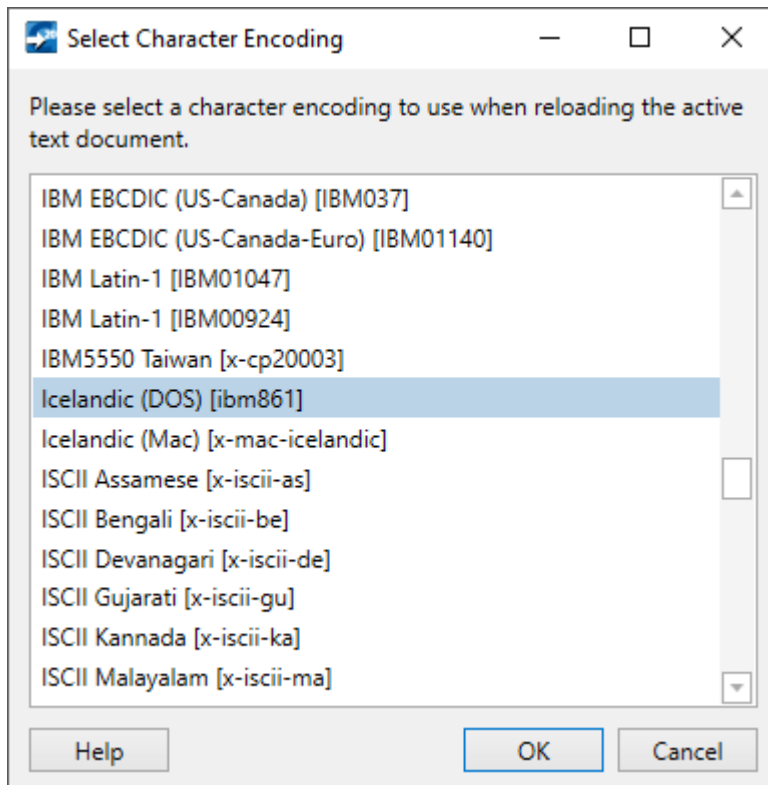
- If you open a Mastercam 2019 or earlier post in Code Expert 2020, Code Expert will use the active character encoding scheme that has been selected in Code Expert.

Mastercam 2019 and Mastercam 2020 automatically convert posts to UTF-8 when you migrate or update them from earlier versions, but you should be aware of this distinction if you work with or support posts from earlier Mastercam versions.

There are two places in Code Expert where you can select a character encoding scheme. The default method is set in the **Application Options** dialog box, on the **Document Settings** page.



The second is the **Reload with Encoding** command on the **Home** tab.





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

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



www.mastercam.com

Mastercam®

Mastercam® is a registered trademark of CNC Software, Inc. ©1983-2019. All rights reserved.
All other trademarks are property of their respective owners.