What’s New in Mastercam for SolidWorks X8

July 2014
Mastercam® X8 What’s New

Date: July 2014
Copyright © 2014 CNC Software, Inc.—All rights reserved.
Software: Mastercam for SolidWorks X8

TERMS OF USE

Use of this document is subject to the Mastercam End User License Agreement. A copy of the Mastercam End User License Agreement is included with the Mastercam product package of which this document is part. The Mastercam End User License Agreement can also be found at:
http://www.mastercam.com/companyinfo/legal/LicenseAgreement.aspx

Be sure you have the latest information!

Information might have been changed or added since this document was published. The latest version of this 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.
Contents

Introduction ....................................................................................................... 1
  ▶ Mastercam Documentation ................................................................. 1
  Contact Us ............................................................................................... 1
  ▶ Mastercam Resources ........................................................................ 1

System Enhancements .................................................................................. 3
  ▶ Mastercam Simulator Improvements ................................................... 3
    Viewports .............................................................................................. 3
    Part Sectioning.................................................................................... 4
    Drill Cycle Handling............................................................................ 5
    Workpiece Tolerance.......................................................................... 6
    Exporting Support Data ...................................................................... 6
    Additional Simulator Improvements .................................................. 7
  ▶ Code Expert Editor Improvements ...................................................... 7
    Multi-Stream Navigation ..................................................................... 7
    NC Configuration Enhancements ....................................................... 8
    New Find Extents Functionality ......................................................... 8
    Additional Enhancements .................................................................. 8
  ▶ Selection Improvements .................................................................... 8
  ▶ Plane Improvements .......................................................................... 10
  ▶ System Configuration Enhancements ................................................ 12
  ▶ Migration Wizard ............................................................................... 12

Mill Enhancements ........................................................................................ 15
  ▶ 2D Milling .......................................................................................... 15
    2D HST Improvements ........................................................................ 15
    Region Chaining ................................................................................ 16
2D Contour Enhancements ................................................................. 20
Wizard Holes .................................................................................. 21
Toolpath Nesting ............................................................................. 21

► 3D Milling ....................................................................................... 22
  Z Stepdowns ................................................................................. 22
  3D HST Project Depth Cuts ............................................................. 23
  Tilt Away to Avoid Collision ......................................................... 23

► Multiaxis Milling ........................................................................... 25
  Multiaxis Roughing ....................................................................... 25
  Safety Zone Improvements ........................................................... 25
  Tool Inspection .............................................................................. 26
  Additional Improvements .............................................................. 27

► Mill Tooling .................................................................................. 27
  Tool Creation and Editing ............................................................... 27
  Edit Holder .................................................................................... 28
  Edit Assembly Names ................................................................... 29
  Calculate Feeds and Speeds .......................................................... 31
  Standalone Tool Manager ............................................................. 32

Lathe Enhancements ........................................................................ 33
  Lathe Stock Model Support ........................................................... 33
  New Contour Roughing Toolpath ................................................... 34
  Additional Improvements .............................................................. 34
Introduction

Welcome to Mastercam for SolidWorks X8! Mastercam for SolidWorks X8 features new products and new functionality focused on delivering speed and efficiency for your machining jobs. We are sure that you will benefit from what Mastercam for SolidWorks X8 has to offer you and your shop.

Mastercam Documentation
Mastercam installs the following documents in the \Documentation folder of your Mastercam installation:

- What’s New in Mastercam for SolidWorks X8
- Mastercam for SolidWorks X8 Installation Guide
- Mastercam for SolidWorks X8 Administrator Guide
- Mastercam for SolidWorks X8 Transition Guide
- Mastercam for SolidWorks X8 Quick Reference Card
- Mastercam for SolidWorks X8 Post Debugger User’s Guide
- Mastercam for SolidWorks Tutorial (Mill)
- Mastercam for SolidWorks X8 ReadMe

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

Mastercam Resources
Enhance your Mastercam experience by using the following resources:

- Mastercam Help—Access Mastercam Help by selecting MastercamX8, Contents from the SolidWorks Help menu, or by clicking the Help Topics link in the Mastercam Info Center on the SolidWorks Task Pane. Also, most dialog boxes and ribbon bars feature a Help button that opens Mastercam Help directly to related information.
- Mastercam Reseller—Your local Mastercam Reseller can help with most questions about Mastercam.
2 MASTERCAM FOR SOLIDWORKS X8/ Introduction

- Technical Support—CNC Software’s Technical Support department (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 University—CNC Software sponsors Mastercam University, an affordable online learning platform that gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master your skills at your own pace and help prepare yourself 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, including many videos, at www.mastercam.com. For tech tips and the latest Mastercam news, you can join us on Facebook (www.facebook.com/mastercam), follow us on Twitter (www.twitter.com/mastercam), and subscribe to our blog, Mastercam Xtras (http://blog.mastercam.com). Visit our YouTube channel to see Mastercam in action (www.youtube.com/user/MastercamCadCam)!

Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the knowledge base at kb.mastercam.com. To register, select Help, Link on Mastercam.com from the Mastercam menu and follow the instructions.
System Enhancements

Mastercam Simulator Improvements

Viewports
A new View ribbon bar has been added for both Backplot and Verify. The ribbon bar contains some items from the Home page (Show and 3D View) and adds two new sections called Window and Viewports. The Viewport tools are a new feature to both Backplot and Verify that split the graphics window into one of four configurations. Each viewport offers separate view orientation and control. Right-click options only work in the current view (outlined with a red border). Visibility options affect all viewports (workpiece display, tool display, etc.). The Restore default view option resets the configuration back to the original size. You can save the viewport configuration to your default settings by choosing Home, Save to Defaults.
Part Sectioning

The view sectioning tool on the Verify ribbon bar has been expanded to add three additional plane orientations. In an improvement targeted toward Mill users, when the planes are initially displayed, they are oriented around the center of gravity of your part. Even if your part is not near the system origin, it will still section efficiently. Selecting one of the ¼ view orientations displays two shaded planes - these clipping planes can be moved by left-click/hold and dragging the cursor to a new location.
You can also use just one clipping plane at a time to create the part view you need.

### Drill Cycle Handling

Drill cycles in both Backplot and Verify now respect the settings in the parameter pages, so you see the tool moving in and out as defined. This applies to all cycles for Mill and Lathe.
Workpiece Tolerance
The Backplot/Verify Options dialog box includes a new Workpiece tolerance option. Targeted towards users with very small parts, it gives you better control over the workpiece quality. The Verify/Compare feature provides better results, since you can now match the tolerance between the part and the workpiece.

Exporting Support Data
To improve customer support, you can now save your Mastercam Simulator session data to a file to send to CNC Software, Inc. for analysis. Choose File, Help, Export.
Support Data. Enter a file name and all the data for your Simulator session is saved.

Additional Simulator Improvements

- Improved image quality of custom (STL or solid) stock.
- Display correct feed rate for each move.
- Corrected displayed values in the Move List and measuring tools, which are now relative to the current toolpath’s WCS instead of world coordinates.
- Improved lighting source and direction.
- Added message when Stop Condition is triggered.
- Added support for stock origin with WCS offsets.
- Improved quality and reduced size of mesh saved in STL file.

Code Expert Editor Improvements

Multi-Stream Navigation

The editor in Mastercam Code Expert now properly supports the display of any number of streams. The editor uses the stream definition in NC Configuration and the starting and ending characters for each stream to determine if single or multiple files are required. You can toggle the display of each stream on and off in a new Navigation
pane. Sync codes are now graphically linked between visible streams, and the Navigator shows which streams contain associated sync points when you hover over the code.

**NC Configuration Enhancements**

NC Configuration allows you to tailor Code Expert display for NC files by defining the number of streams, file extensions, sync characters, tool callouts, comments, and other features for individual machine output.

You can access the new interface for creating and modifying the NC Configurations in the Application Options dialog box and directly using the NC Document ribbon bar.

Default NC Configurations are provided for several machine and control types, providing easy starting points for user customization.

**New Find Extents Functionality**

NC File document types now include an Extents pane which lists the extents for address characters in the NC file. The address characters are easily defined for reuse in the NC Configuration and can be customized in each NC document type. Double-clicking on an extent moves the cursor in the NC document to the location where the value appears.

**Additional Enhancements**

- Fonts and Colors are now fully customizable for each document type.
- **Go To Bookmark** functionality added, which allows you to add a bookmark to a document and easily return to that bookmark as desired.

**Selection Improvements**

- Better tool plane selection allows you to select an existing tool plane from within a Mill toolpath's Geometry selection dialog box. This improvement
makes it easier to accommodate changing tool planes during axis positioning or tombstone work.

- New **Propagate along tool plane** option reduces the need for manual selection in cases where **Select Loop** or **Propagate along tangencies** may not be an option.
New manual selection options for defining the start and end position in the Chain Manager. This is available in both Mill and Lathe toolpaths.

Selection filters are added to both the Mill Stock Setup Bounding Box and the Create Boundary dialog boxes.

Helpful prompts using the SolidWorks yellow message boxes have been added to selection dialog boxes throughout the product.

**Plane Improvements**

Throughout Mastercam for SolidWorks X8, the term “view” has been replaced with the term “plane” (except for graphics views). A Mastercam Plane contains a view and an origin, and a View is a matrix defining an orientation. In keeping with this change, the View Manager is now the Plane.
New options are also available in the right-click menu of this dialog box.
The plus sign button on the Plane Manager creates a SolidWorks sketch point at the origin of the currently selected tool plane. You can also access this function by choosing Toolpath Utilities, Create point at Tplane origin.

Planes are marked “dirty” if their regeneration fails. Any toolpaths that use that plane will also be marked dirty.

You cannot make changes to the default planes. You also cannot delete a plane that fits any of the following criteria:

- Plane is locked.
- Plane is being used as the active Tplane, WCS, or Gview.
- Plane is used in a machine group.

Plane names are no longer limited to 40 characters.

Selecting geometry to define a plane will always offer eight possible plane options.

You can now set your graphics view equal to the WCS (in the menu) or to a Mastercam Plane.

Automatic work offset assignment has been improved, with better performance and handling of conflicting assignments. In addition, the Work offset already used warning can now be turned off.

System Configuration Enhancements

“Mastercam” communication functionality has been removed. CIMCO 7 is now the default and only distributed option.

Migration Wizard

Mastercam for SolidWorks's Update Folder and Migration Utility features have been combined into a new Migration Wizard. You can access this new feature by choosing Toolpath Utilities, Migration Wizard.
The Migration Wizard offers a **Basic** mode and an **Advanced** mode. The Basic mode will automatically update all Mastercam file types of the previous version (for example, Mastercam for SolidWorks X7) from a given location, while the Advanced mode provides additional control over file types and previous versions to migrate.

The file update process has also been improved to provide a seamless, more efficient experience. Migration results are now displayed in the Mastercam for SolidWorks Event Log as well.
Mill Enhancements

2D Milling

2D HST Improvements

- 2D HST now supports multi-core processing on toolpaths with multiple machining regions.
- Consolidated Dynamic Area, Core, and Rest toolpaths down to one single Dynamic Mill toolpath. Also consolidated Area, Core, and Rest Mill toolpaths down to one single Area Mill toolpath.

- 2D HST Dynamic Mill includes first pass feed reduction and offset on the Cut Parameters page. This reduces the feed rate for the first pass of a dynamic mill toolpath entering stock from outside. It also expands the machining region to ensure the tool does not engage a lot of material on the very first pass due to an unknown exact block size.

<table>
<thead>
<tr>
<th>X7</th>
<th>X8</th>
</tr>
</thead>
<tbody>
<tr>
<td>Core Mill</td>
<td>Dynamic Mill</td>
</tr>
<tr>
<td>First Mill</td>
<td>Area Mill</td>
</tr>
<tr>
<td>Blend Mill</td>
<td>Dynamic Contour</td>
</tr>
<tr>
<td>Area Mill</td>
<td>First Mill</td>
</tr>
<tr>
<td>Dynamic Area Mill</td>
<td>Dynamic Core</td>
</tr>
</tbody>
</table>

- Cutting method: Climb
- Tip comp: Tip
- Approach distance: 0.0
- First pass offset: 0.0
- First pass Feed reduction: 0.0
Region Chaining

- Simplified region selection.
- New Air Region chains define safe air the tool can move within via closed chains.

- Open Air Region chain support defines an open side of a closed machining region or closes an open machining region chain.
- New Containment chains constrain machining region motion.
- Job setup stock shape support used when no machining region is selected or open machining region chains are selected. Open machining region chains are extended to block tangent edges or shortest distance.

- When no stock is defined and no open air is defined, open machining chains are closed with a straight line.
2D Contour Enhancements

You can now adjust feeds and speeds on the Multi Passes page, making it easier to control your tool motion. In addition, Mastercam for SolidWorks X8 adds corner rounding support in Ramp and Oscillate modes.

Manual Tabs

Support for Manual tabs is now included in Contour toolpaths. Until now, Mastercam for SolidWorks only had Automatic tabs. Any single position (sketch point, line midpoint, vertex, etc.) can be used to indicate the location of a tab along the chained geometry.
Wizard Holes
This new toolpath lets you manually select SolidWorks Hole Wizard Holes features and create drill toolpaths on just those features. You can access this toolpath by choosing Feature-Based Toolpaths, Wizard Holes.

Toolpath Nesting
Mastercam for SolidWorks X8 includes new toolpath nesting capabilities. You can now toolpath your parts in "assembly" space and then nest them flat to a sheet in the same assembly file.
3D Milling

Z Stepdowns
3D high speed toolpaths based on Z-level cuts now process Z stepdowns across multiple cores.
3D HST Project Depth Cuts

Mastercam for SolidWorks X8 includes new support on Cut Parameters page for multiple depth cuts. The **Order by depth** checkbox controls the cut order by depth or by input entity.

![3D HST Project Depth Cuts](image)

Tilt Away to Avoid Collision

These new options in 3D HST finishing toolpaths provide an easy-to-use introduction to 5-axis technology. Additional options on the Holder page use tilting to avoid shank and holder collisions during multiaxis motion. This toolpath requires a spherical tool.
NOTE: These options require a Mastercam Multiaxis license.
**Multiaxis Milling**

**Multiaxis Roughing**
Multiaxis Roughing is a new strategy added for X8. It allows the easy creation of pocketing toolpaths to rough out pockets that have a curved floor surface. It supports offset or Dynamic style strategies, and multiple depth floors.

**Safety Zone Improvements**
Improvements to the Safety Zone feature in multiaxis toolpaths makes defining and editing the safety zone much easier. On the dialog box, seeing a representation of the shape while defining makes it much easier and more precise to enclose the targeted volume.
Tool Inspection

Tool inspection was added to many of the multiaxis toolpaths, including Blade Expert and Port Expert. After you enter an inspection parameter, either in distance or time,
the tool will retract and trigger an inspection code at the next linking move. Your post processor must be modified to support this new feature.

**Additional Improvements**

- Enhancements to oscillation in Curve 5-axis.
- Ability to finish blades in Blade Expert with a SWARF algorithm.

*Mill Tooling*

**Tool Creation and Editing**

The old style Define Tool dialog box has been replaced with the new interface throughout the product. This provides a more consistent user experience for creating and editing tools. Some of the existing tool types are also updated so they no longer require custom profiles.
In preparation for future updates, the group boxes include expanders for advanced properties.

Collapsed state

Expanded state

**Edit Holder**

The Edit Holder interface has been updated to support additional properties. Prior to X8, all non-geometric attributes had to be edited using the stand-alone Tool Manager. With the X8 release, you can edit properties such as the upper and lower connection type and size from within Mastercam for SolidWorks.

The right-click menu in the Mastercam Tool Manager includes a new **Edit Holder** option. Selecting this option displays the edit holder interface.
The **Edit Holder** option is still available in the Holder parameters page for tree-style toolpath dialogs, but making it available in the Mastercam Tool Manager is useful when working with older toolpath types that do not offer the Holder page.

In addition, the .TOOLDB format is now the standard for holder libraries. This replaces the .HOLDERS file format.

**Edit Assembly Names**

Changing tool, holder, and assembly names is now available through a right-click menu option in the Mill operation tool parameters page.
The Assembly Name field has also been added to the Holder page for convenience.
Calculate Feeds and Speeds

The calculate feed/speed button has been added to the Create and Edit tool dialog boxes. This option is useful when you are working on tools inside of Mastercam for SolidWorks because it can query the workpiece material of the active machine group.
Standalone Tool Manager

Geometry properties have been removed from the property grid and replaced with a button that will launch the Edit Tool dialog box. You can also double-click on an item in the data grid to edit the tool.
Lathe Enhancements

**Lathe Stock Model Support**
Lathe toolpaths can now be included in stock model operations, which can be used for subsequent milling operations.
New Contour Roughing Toolpath

This new roughing toolpath is designed to take passes parallel to chained geometry. This toolpath is useful for parts where the initial stock shape is similar to the final part shape.

Additional Improvements

- **Added Incremental Depth Cuts** to Lathe Rough toolpath that allow incremented and decremented depth cuts.
- Disable the **Tool Display** option to see significant processing speed improvements for Dynamic Lathe and other toolpaths.
- Lathe stock and tailstock are now associated with any solid body used to define them.
- New upper and lower profile options in Create Boundary are very helpful for Lathe users.
Attention! Updates may be available.
Go to Mastercam.com/Support for the latest downloads.