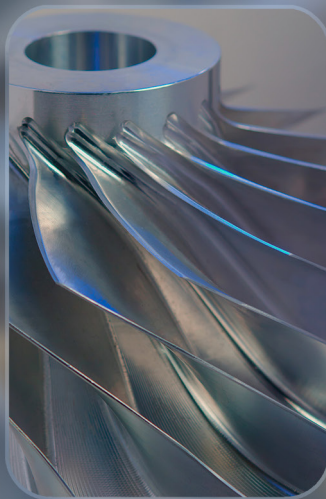


Mastercam X⁷

What's New

N662 X13.4145 Y-3.246 Z1.6613 A-263.456 F941.26
N664 X13.4145 Y-3.246 Z1.6613 A-263.456 F941.26
N666 X13.4095 Y-3.2339 Z1.8235 A-264.288 F922.43
N668 X13.4045 Y-3.2136 Z1.9851 A-261.416 F902.58
N670 X13.3995 Y-3.1852 Z2.1456 A-258.543 F882.56
N672 X13.3945 Y-3.1488 Z2.3048 A-255.662 F862.8
N674 X13.3912 Y-3.1202 Z2.4095 A-253.75 F849.28
N676 X13.3874 Y-3.0826 Z2.5295 A-251.532 F834.7
N678 X13.3835 Y-3.0408 Z2.6471 A-249.331 F819.91
N680 X13.3785 Y-2.9791 Z2.799 A-246.442 F801.23
N682 X13.3735 Y-2.9098 Z2.9475 A-243.552 F783.53
N684 X13.3685 Y-2.8333 Z3.0922 A-240.665 F766.24



What's New in Mastercam X7

April 2013

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. The ReadMe file (ReadMe.pdf) includes the latest information about Mastercam features and enhancements.



Mastercam® X7 What's New in Mastercam X7

Date: April 2013

Copyright © 2013 CNC Software, Inc.— All rights reserved.

First Printing: April 2013

Software: Mastercam X7

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://dnn.mastercam.com/companyinfo/legal/LicenseAgreement.aspx>

Contents

.....

Introduction	1
▶ Mastercam Documentation	1
Contact Us	1
▶ Mastercam Resources	1
General Enhancements	3
▶ Mastercam Mill-Turn	3
Sync Manager	3
New .MACHINE File	4
New Posting Language	4
New Job Setup Interface	5
Part Handling Operations	5
Enhanced Plane and Axis Combination Selection	6
Pinch Turn Toolpath	7
▶ Mastercam Simulator	8
▶ New File Editor	11
▶ Mastercam Advanced Configuration Utility	12
▶ Customer Satisfaction Survey	13
▶ New Mastercam Art Install	14
▶ File Translator Updates	14
▶ Post Processor Changes	14
▶ Automatic Updates	14
Design Enhancements	17
▶ Modify Solid Feature	17
▶ Modify Solid Fillet	18

- ▶ Remove Solid Fillet 19
- ▶ Solid Hole-Axis..... 20
- ▶ Xform Project Improvements 21
- ▶ 2D Fillet Clearance Offset 22
- ▶ Surface Fillet Improvements..... 23

General Mill Enhancements 25

- ▶ Renishaw Productivity+ 25
- ▶ Stock Model Improvements 26
- ▶ Improved Dynamic Mill and OptiRough Motion 26
- ▶ Hide Button on Tree-Style Dialogs..... 26
- ▶ Redesigned Arc Filter/Tolerance Page 26

Mill Tooling Enhancements 29

- ▶ Workflow Improvements 29
 - New Tool Manager..... 29
- ▶ New Tool Wizard 32
 - New Holder Wizard 35
- ▶ Mill Tool Assemblies 36
 - Multiple Holder Component Support 36
 - Saving to a TOOLDB Library 37
- ▶ Custom Tool Improvements 38
 - Store Profile Data in Document File or (TOOLDB) Library..... 38
 - Export to a Mastercam Level..... 39

2D Mill Enhancements 41

- ▶ Multi-threading Support for 2D HST Toolpaths 41
- ▶ Dynamic Motion for 2D HST Peel Mill..... 41
- ▶ Blend Mill Improvements..... 43
 - Finish Passes 43
 - Synchronized Chain Support 44
 - Extend Entry/Exit 46
- ▶ Relaxed Chaining Restriction for Slot Mill Toolpaths..... 48

▶ New Finish Pass Directions for Helix Bore Toolpaths.....	48
3D Mill Enhancements	49
▶ General 3D Mill Improvements	49
Multi-Core Support.....	49
Check Surface Support.....	49
▶ 3D HST Hybrid.....	49
Optimized Cut Order	49
Open/Closed Passes.....	52
Automatic Offset Method	53
Improved Motion Near Boundaries	54
Smarter Regeneration	55
▶ New 3D HST Toolpath Fillet Page	55
▶ Containment Boundary Improvements.....	56
▶ 3D HST OptiCore and OptiArea Toolpaths.....	56
Optimized Cut Order for Step Ups.....	57
Safer Retract Motion Over Material.....	58
▶ New 3D HST Project Toolpath.....	58
Multiaxis Mill Enhancements	61
▶ Oscillate Motion for Curve and Swarf 5-axis	61
▶ Port Expert Enhancements.....	62
▶ Blade Expert Enhancements	62
▶ Independent Safety Zone for Classic Multiaxis Toolpaths	63
Lathe Enhancements	65
▶ New Dynamic Rough Toolpath.....	65
▶ Rough Zigzag Motion Options.....	65
▶ Face Grooving Start Range Option.....	66
▶ Full Radius Tool Support for Plunge Turn	67

VI • WHAT'S NEW IN MASTERCAM X7

Introduction

Welcome to Mastercam X7! Mastercam X7 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 X7 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 X7*
- *Mastercam X7 Installation Guide*
- *Mastercam X7 Administrator Guide*
- *Mastercam X7 Transition Guide*
- *Mastercam X7 Quick Reference Card*
- *Mastercam X7 Post Debugger User's Guide*
- *Getting Started with Renishaw Productivity+™*
- *Mastercam X7 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 **Help, Contents** from Mastercam's menu bar or by pressing [**Alt+H**] on your keyboard. 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.
- *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 and www.mastercamedu.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)!



General Enhancements

Mastercam Mill-Turn

Mastercam X7 also includes Mastercam Mill-Turn, designed to simplify programming for complex Mill-Turn machines.



IMPORTANT: If you are interested in purchasing this product, please contact your local Mastercam reseller.

The centerpiece of Mill-Turn is the *Code Expert* application. Like the new Tool Manager, the Code Expert runs in a separate window. Code Expert incorporates an entire suite of tools for managing multitasking applications.

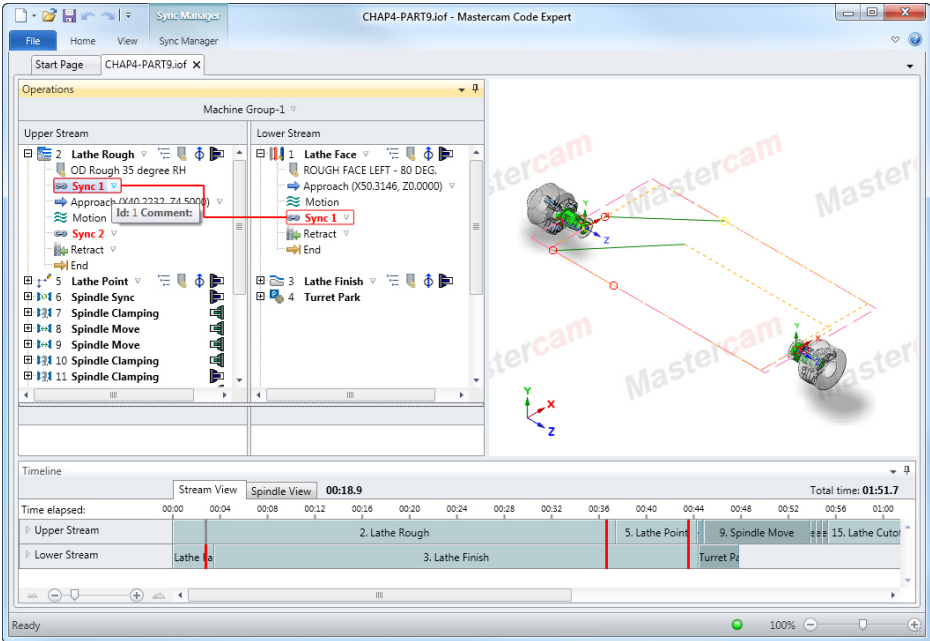
Notes:

- *To access Help throughout Code Expert, click the Help button or press [F1].*
 - *You can hover over the ribbon bar buttons for detailed function explanations and shortcut keys.*
-

Sync Manager

The Sync Manager creates syncs between operations and allows you to enter other machine-specific programming information. It includes:

- A multi-streamed operations tree, similar to the Operations Manager in Mastercam
- A GANTT-style timeline view of your operations.
- A graphic display that shows your part, fixtures, toolpaths, and syncs.



New .MACHINE File

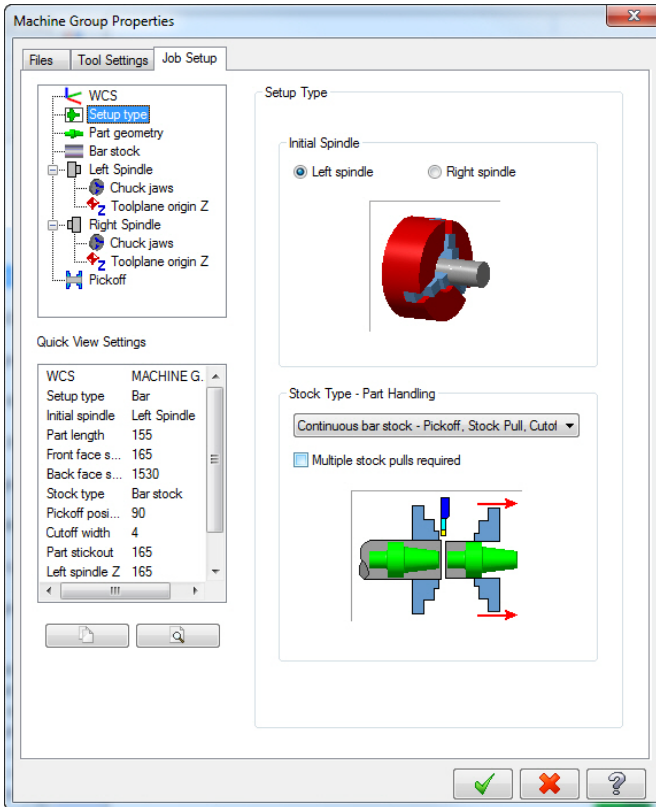
In Mill-Turn, separate machine definitions, control definitions, and post files are combined into a single .MACHINE file that can live anywhere on your system. The .MACHINE file also includes other resources and configuration data that enable it to be much more closely tied to Mastercam workflow and programming.

New Posting Language

Mill-Turn also includes a new posting language called MP.NET. This is a modern object-oriented language that replaces legacy MP for Mill-Turn applications. In conjunction with MP.NET, a new data backbone called the Intermediate Object Format (IOF) has been developed, replacing the ASCII NCI file.

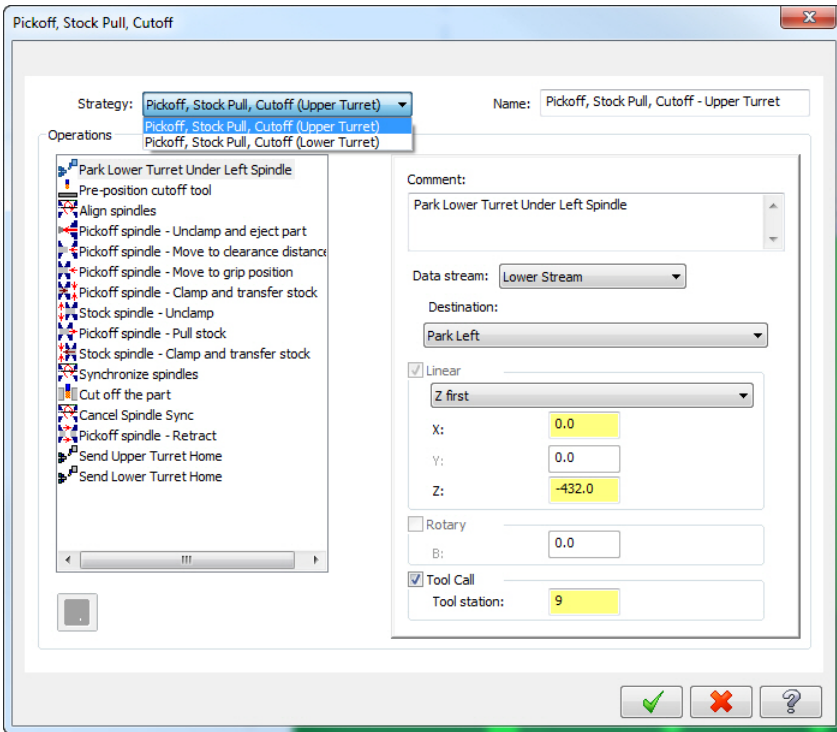
New Job Setup Interface

Mill-Turn features a re-engineered workflow that begins with a new Job Setup interface inside Mastercam. This leverages the .MACHINE file to automatically create planes for programming your part, subspindle setups, and part handling applications.



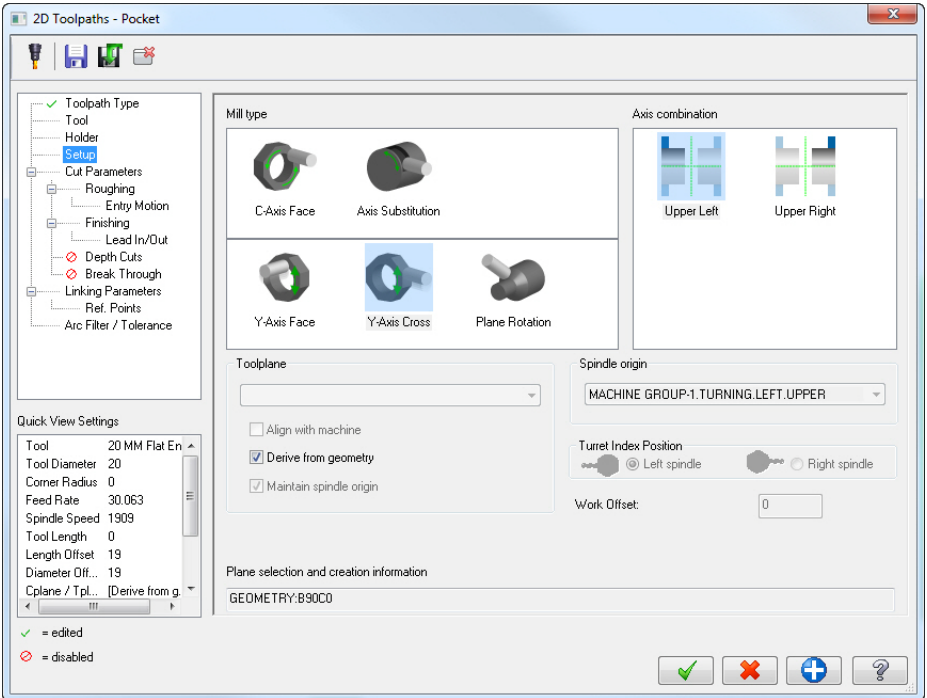
Part Handling Operations

The set of Misc Ops from Mastercam Lathe have been replaced in Mill-Turn by a new set of part handling operations. These are event-based operations that are organized into machine-specific sequences and application strategies.



Enhanced Plane and Axis Combination Selection

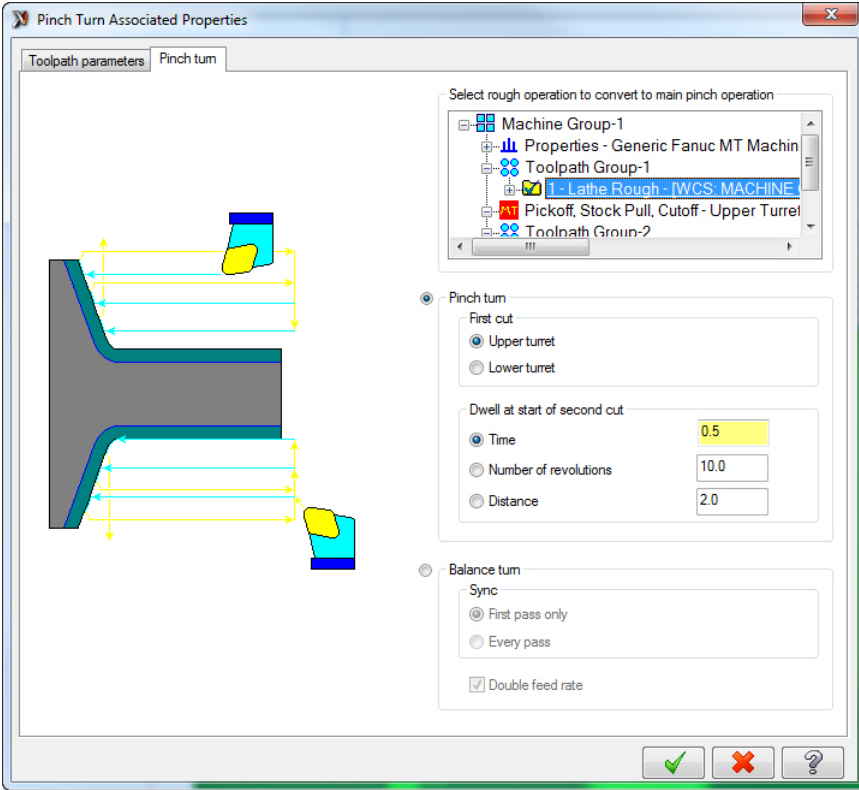
New interfaces have been developed for working with tool planes and axis combinations. These are based on Mastercam's knowledge of your machining and part-handling application, so that a great many unnecessary or redundant options have been removed. For example, the new Setup page for milling operations relies on your chosen application orientation and axis combination to present only those options that make sense.



A similar approach is also used in turning operations, selecting axis combinations, and creating transform operations.

Pinch Turn Toolpath

A new pinch turn operation is now available. This takes an existing roughing operation, creates new cutting passes on the opposite turret, and automatically creates the necessary syncs.

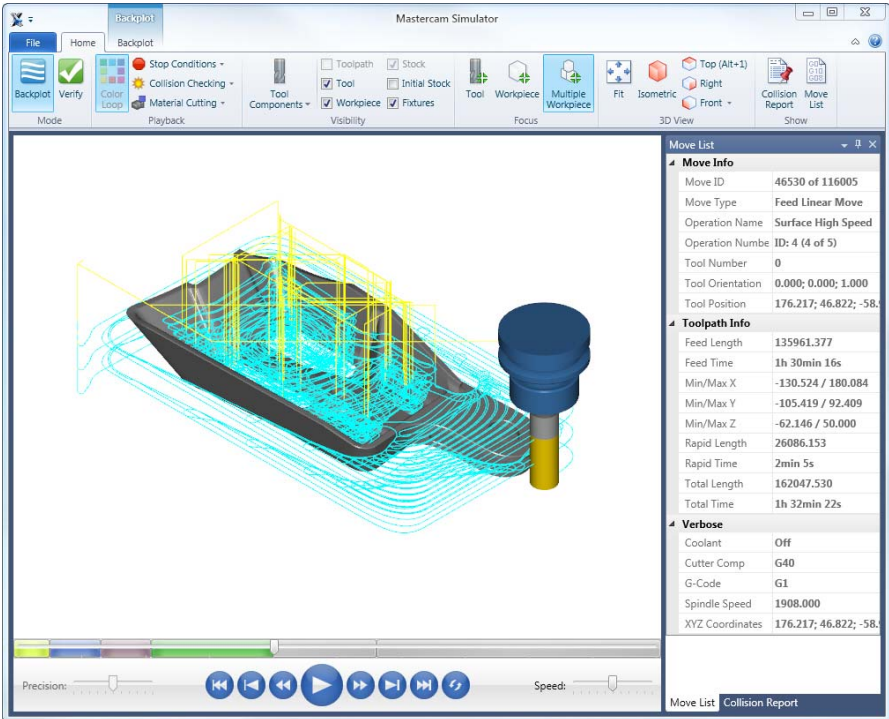


Mastercam Simulator

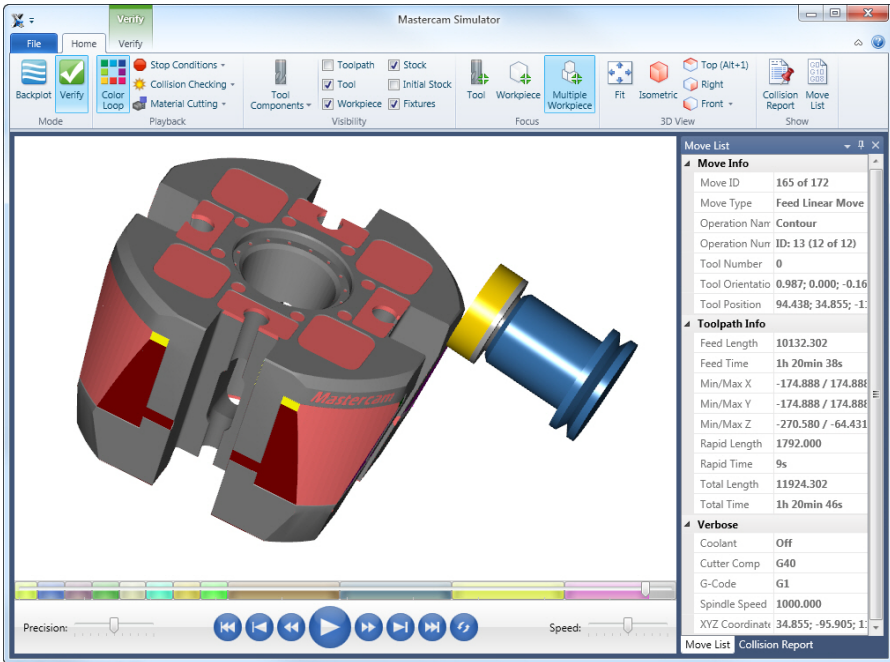
The Backplot and Verify utilities that you use to check all your Mastercam tool motion have now been combined to offer a more efficient workflow, better analysis tools, and more comprehensive toolpath support in Mastercam Simulator.

You still access Backplot and Verify from buttons at the top of the Operations Manager, but these utilities now display their results in a separate window from Mastercam. This allows you to keep making adjustments to your toolpaths even as your tool motion display continues.

*Note: The Backplot button in the Operations Manager still opens the classic Backplot application from previous Mastercam versions. To open the new Backplot function in Mastercam Simulator by default, choose **Mastercam X7, Backplot, Disable** in the **Mastercam Advanced Configuration Utility**.*

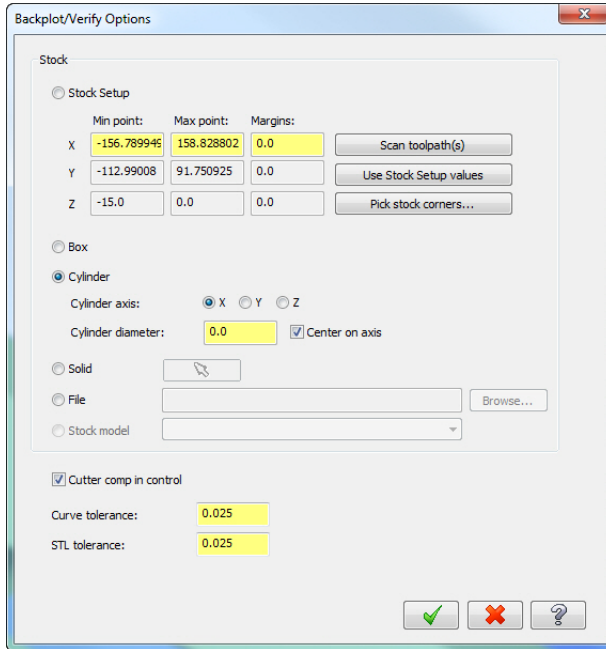


This new Mastercam Simulator window, similar to the window in Machine Simulation, also includes many more analysis tools, such as an improved move list and more color coding options. It also supports more toolpath types, including Transform and Rotary axis.



In addition, both Backplot and Verify in Mastercam Simulator are now using the same NCI data to verify your tool motion. This creates more consistent results between the two verification options.

The run-time settings that used to be displayed as a dialog box on the side of the screen have been replaced with a dialog box you can access from the Operations Manager toolbar. You can adjust these settings and reload your part file by clicking Backplot or Verify again.



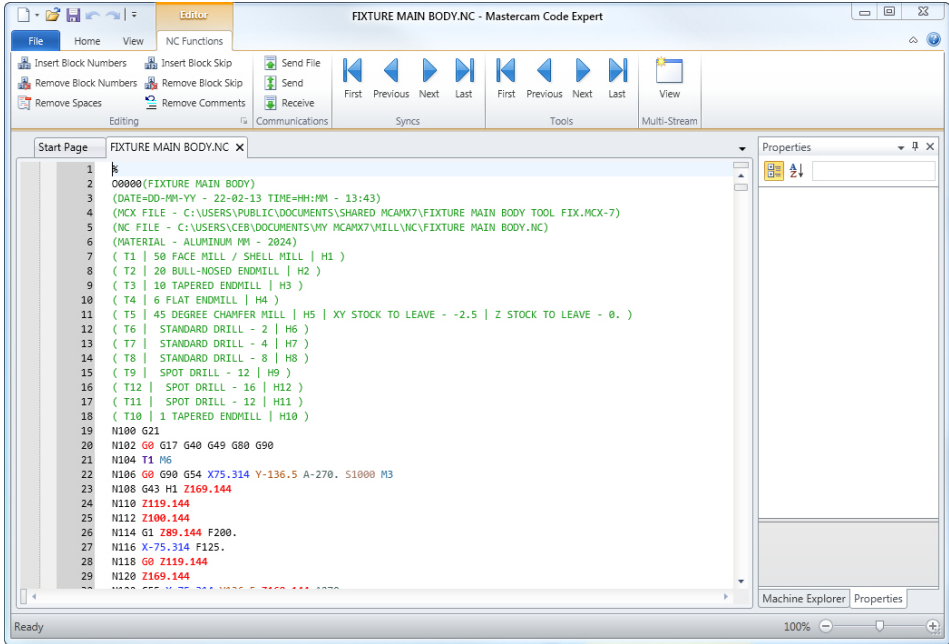
Notes:

- You can only have one Mastercam Simulator window open at a time.
 - You can hover over the ribbon bar buttons for detailed function explanations and shortcut keys.
-

New File Editor

A new editor replaces the editor formerly supplied with Mastercam. In addition to the new MP.NET posting language, it supports legacy MP, NC code, and VB script. It

includes completely configurable keyword highlighting, AutoComplete, communications, prebuilt code snippets, outlining, and bookmarks.

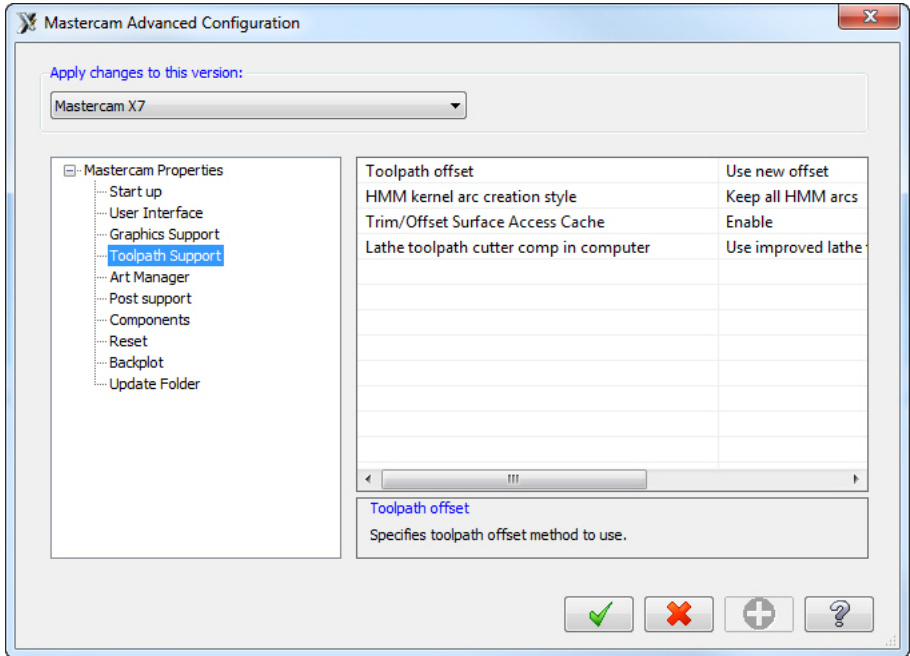


Note: For Mastercam Mill-Turn, the new file editor is fully integrated with the other Code Expert tools.

Mastercam Advanced Configuration Utility

You can use the Mastercam Advanced Configuration dialog box to manipulate features of Mastercam and Mastercam for SolidWorks. Access the dialog box by selecting **Mastercam X7, Utilities, Advanced Configuration** from the Start menu. Click the **Help** button to learn more about the options available in this dialog box.

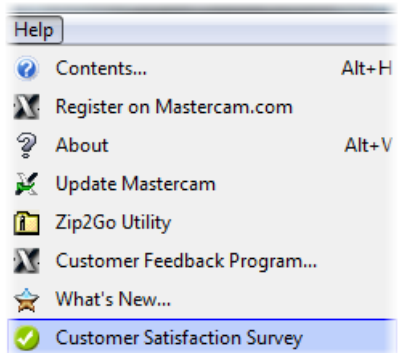
Note: This utility replaces the Mastercam Control Panel Applet from previous versions.



Customer Satisfaction Survey

The Mastercam X7 Help menu now provides quick access to the new [Mastercam Customer Satisfaction Survey](#). Please use this short survey to tell us about your satisfaction with Mastercam, its usefulness to you in the workplace, and its ease-of-use. This information will help us to improve your Mastercam experience.

- The survey has no end date, because neither does our interest in listening to our Mastercam customers.
- All Mastercam users can participate and should use the survey regularly to check in.



- You can win a \$50 USD iMastercamstore.com gift card - winners are selected each quarter.
- The survey is available in English, German, French, Italian, Spanish, Chinese (Simplified), and Japanese.

If you have any questions or comments regarding this survey, please contact userexperience@mastercam.com.

New Mastercam Art Install

Mastercam Art is now available as a 64-bit application, which lets you take full advantage of your 64-bit hardware and operating systems.

File Translator Updates

AutoCAD: 2013

Catia: V5 R22

Parasolids: Version 25

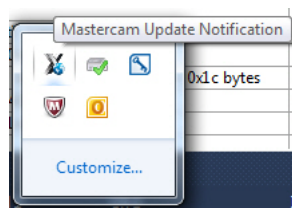
Rhino: V5

Post Processor Changes

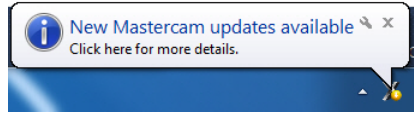
The only change to NCI data for Mastercam X7 is that the 20700 line is no longer written to the NCI file. For information on any additional changes, please contact your local Mastercam Reseller for a copy of the *Mastercam X7 NCI & Parameter Reference*.

Automatic Updates

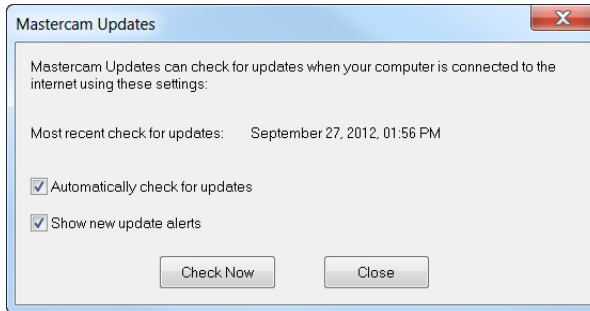
Mastercam X7 features a tray notification program that will automatically check for updates to Mastercam.



If an update is available, a notification message displays. If you click on the message, you can read about what has changed, and then decide whether or not to install the update. If you close the message, the notification stays in the tray and you can select it later.



You can also disable the automatic check. A manual check is always available from the **Help, Update Mastercam** menu option.



Design Enhancements

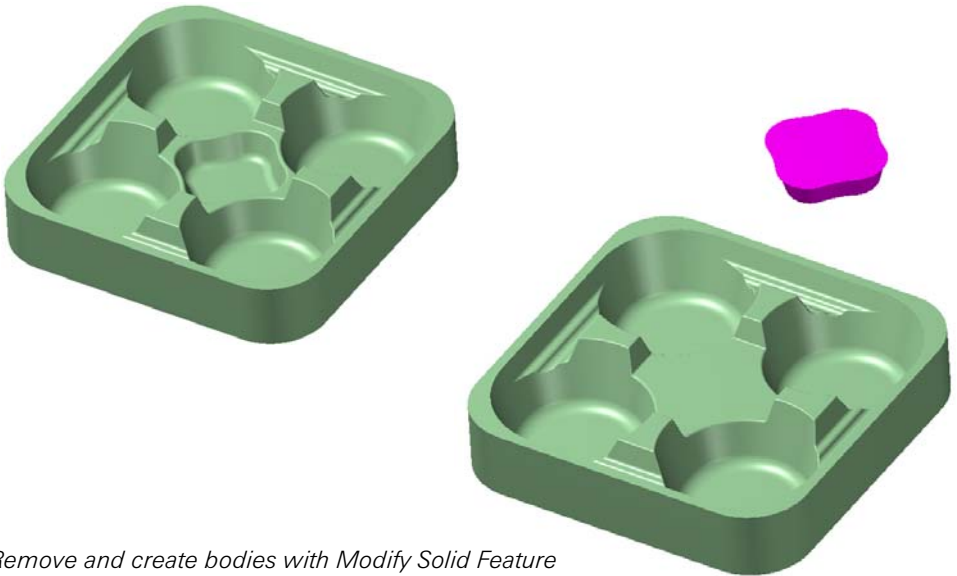
Modify Solid Feature

The new Modify Solid Feature function in Mastercam X7 allows you to create bodies and/or remove features from solid models that do not have any operation history. To create or remove a body, select the face of any feature, including bosses and open pockets, on the solid model. Mastercam recreates the feature in a new, independent solid body and leaves the original solid unchanged, or eliminates the feature by modifying the original body.

The **Remove and create body** option allows simultaneous model creation and modification. Upon creation, bodies generated by the Modify Solid Feature function occupy the same space as the original selected feature.



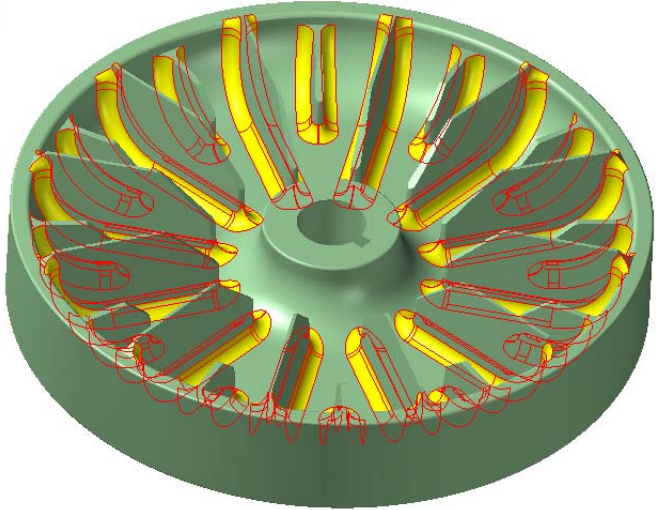
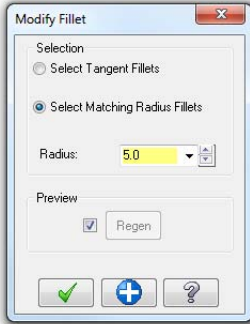
IMPORTANT: Modifications to a solid model using this function cannot be undone. Work incrementally and be aware of all selections before applying changes.



*Remove and create bodies with Modify Solid Feature
(Newly created bodies moved for demonstration purposes)*

Modify Solid Fillet

Use the new Modify Solid Fillet function to quickly change the radius of fillets in solid models that do not have any operation history. You can apply your changes to sets of tangential fillets or fillets with matching radii.



The radius value in the dialog box is populated by the first fillet you select. When you select fillets with different radii, they are edited to match the value of the first selected fillet.

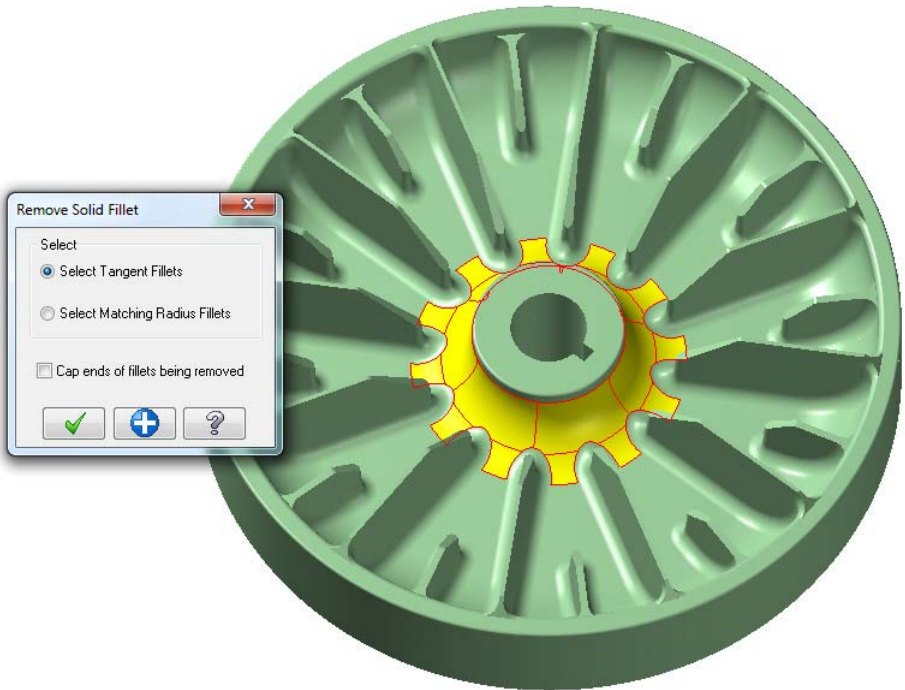
Mastercam displays an error message if the function is unable to apply your modifications to the fillets you select. If you encounter any errors, try narrowing your selection. Working incrementally with a smaller selection of fillets should yield better results.



IMPORTANT: Modifications to fillets using this function cannot be undone. Work incrementally by editing fewer fillets per application, and be aware of all selections before applying changes.

Remove Solid Fillet

The Remove Solid Fillet function removes fillets from solid models that do not have any operation history. You can select and remove fillets individually, or in groups of tangential fillets or fillets with matching radii.



The function is unable to remove overlapping fillets (PK Error Code: 525). If you encounter this error, try narrowing your selection to remove only the top-most fillets. If you still cannot remove the fillets, select the **Cap ends of fillets being removed** option to remove the fillet in pieces.



IMPORTANT: Modifications to fillets using this function cannot be undone. Work incrementally by editing fewer fillets per application, and be aware of all selections before applying changes.

Solid Hole-Axis

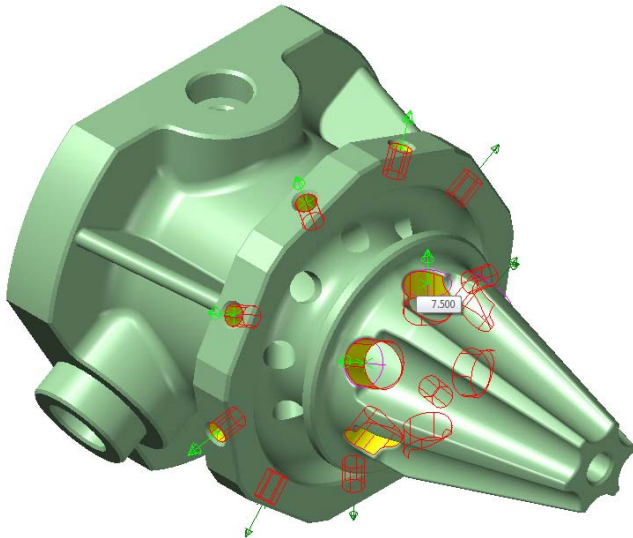
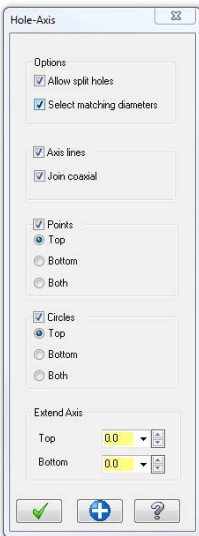
The new Solid Hole-Axis function creates axis lines in circular holes in solid bodies, with or without history. Hole axis lines are created using your current system attribute settings (line size, color, etc.). You can also create related geometry (points and circles) using the options in the dialog box.

You can only use this function on cylindrical holes, both open and closed. You can create hole axis lines on the following types of cylindrical holes:

- Through holes
- Blind holes
- Holes completely encased by the solid
- Split holes

Choose **Solids, Hole-axis**, set your options in the Hole-axis dialog box, and select the wall (internal face) of each hole where you want to create the live axis line(s). The axis lines, points, and circles remain live until you click **Apply** or **OK**. While the lines and related geometry are live, you can work in the dialog box to make any changes. The changes you make display in “real time” as you work. When you are satisfied with your results, click the **Apply** or **OK** button to set the axis lines and geometry.

The following picture shows examples of live axis lines:



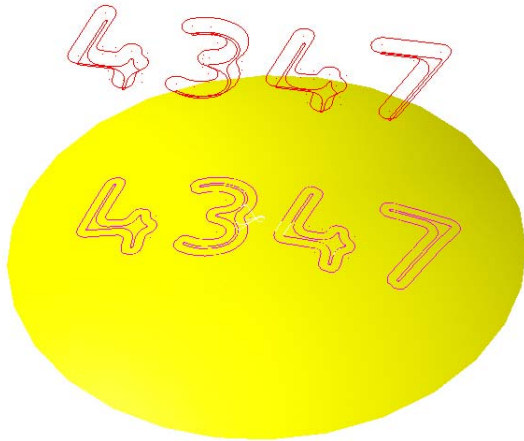
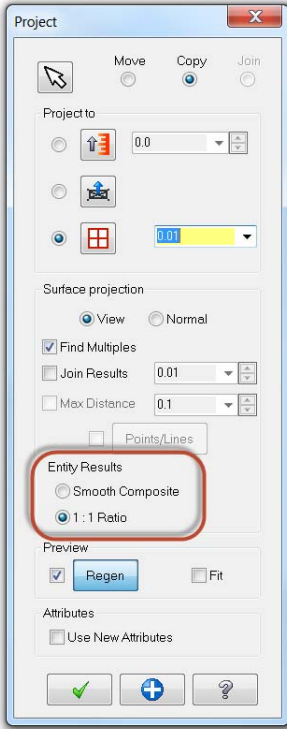
Notes:

- *You can select fillets, but you cannot apply this function to chamfers or round-overs.*
 - *Vector arrows show direction that Mastercam recognizes as the top of the holes. The direction on through holes depends on the point of your mouse selection. The direction on blind holes always points out (top). You can choose to change the vector direction by clicking the axis line.*
 - *Axis lines start at the bottom of the cylinders, regardless of the shape of the hole.*
 - *Axis lines end at the top of the cylinders, regardless of the shape of the top of the hole.*
 - *Axis lines created on holes with angled tops end at the top of the angle.*
 - *Counterbored holes require a selection for each diameter.*
 - *Interrupted holes require a selection for each side.*
-

Xform Project Improvements

New controls have been added to the Xform Project dialog specifically for projecting onto a surface. Particularly useful when working with small parts, Mastercam now prompts you to increase your default tolerance to attain the best results.

In addition, you can choose between projecting blended splines or attempting to match the number of source geometry pieces to the number of projected entities.



2D Fillet Clearance Offset

A new **Additional clearance distance** option has been added to the Fillet Entities and Fillet Chain ribbon bars. This option is only available when the fillet style is set to **Clearance**.



The maximum allowable value equals 95% of the fillet radius. If a larger value is entered, the value will automatically be changed to equal 95% of the fillet radius. The

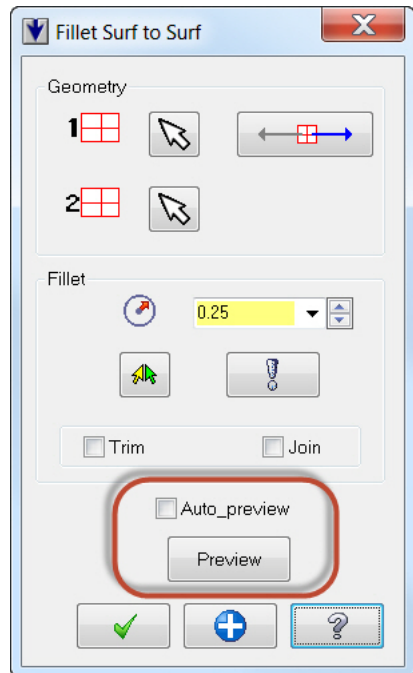
distance is measured along the fillet radius from the intersection of the entities, as shown.



Surface Fillet Improvements

To improve the workflow, the Surface Fillet dialog boxes now include a Preview button instead of a Preview checkbox. In Mastercam X7, the dialog box will open more quickly and allow you to modify the parameters without first trying to process the fillet result. You can then choose to see a preview by pressing the **Preview** button. The Preview button only provides a snapshot of the current settings. Any changes made after are not reflected until you click **Preview** again. Once you click **OK** or **Cancel**, the preview state is turned off.

You can also select the **Auto-preview** checkbox to automatically preview modifications in the graphic window. When you modify values in the dialog box fields or perform actions in the graphics window, the results are immediately updated.



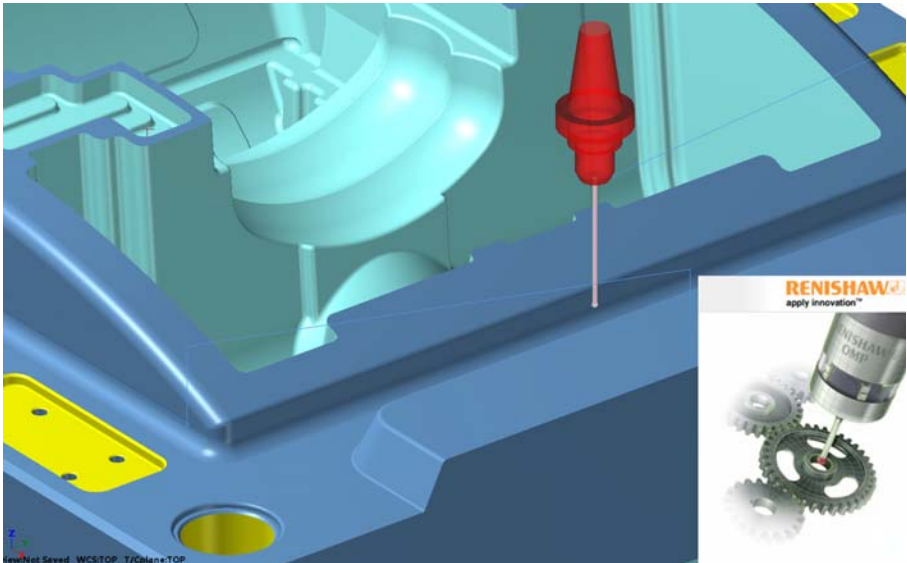
General Mill Enhancements

Renishaw Productivity+

Mastercam X7 integrates Renishaw's Productivity+ for in-process gauging. This is the ability to use a measuring probe on a machine tool to determine fixture offsets, orientation, and critical dimensions. Manual tool-setting, job set-up and inspection are time consuming, and prone to operator errors. Probing eliminates the need for tool presettlers, expensive fixtures, and manual setting with dial indicators. Probing is fast and reliable and using Productivity+ software allows for machine offsets to be automatically adjusted, even during the machining process, to further enhance the manufacturing process.



IMPORTANT: If you are interested in purchasing this product, please contact your local Mastercam reseller.



Stock Model Improvements

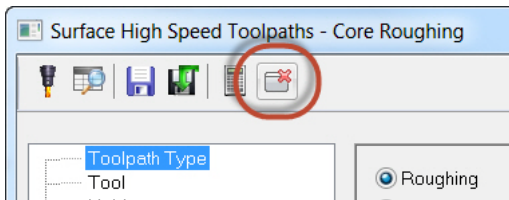
Stock model in Mastercam X7 now supports multi-core processing for all toolpaths, making it even faster to regenerate your stock models.

Improved Dynamic Mill and OptiRough Motion

The Dynamic and OptiRough toolpaths in Mastercam X7 have improved, more efficient tool motion, including enhanced cut orders and faster calculation times.

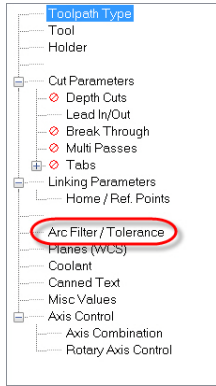
Hide Button on Tree-Style Dialogs

This button temporarily minimizes any tree-style dialog so you can more easily see your part in the graphics window.

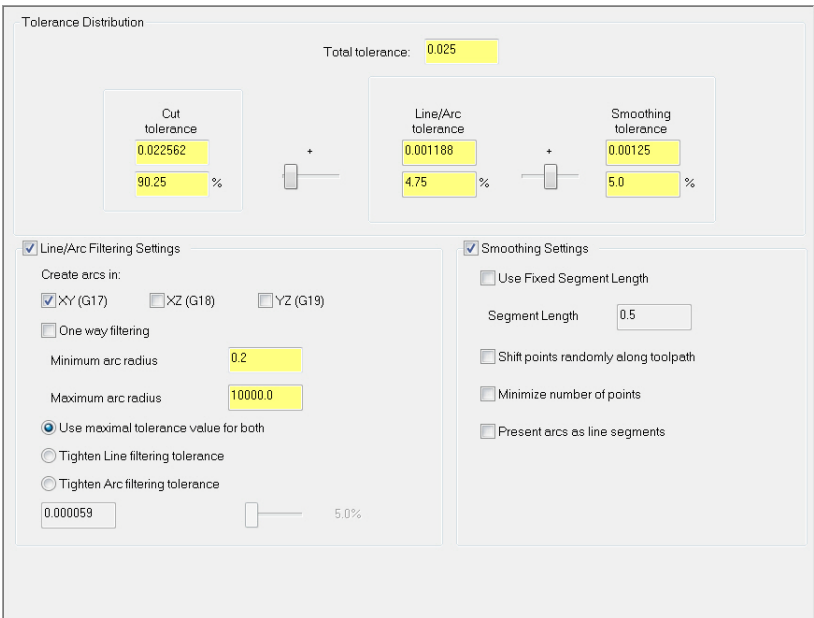


Redesigned Arc Filter/Tolerance Page

Toolpath refinement, previously available to only 3D toolpaths, is now also available to all 2D toolpaths (classic and 2D HST). In Mastercam X7, the ability to apply line/arc filtering and smoothing controls to your toolpaths has been integrated into the toolpath tree-style dialog on the Arc Filter/Tolerance page. It no longer displays as a secondary dialog box and does not need to be enabled.



The layout has been simplified, eliminating the vertical sliders in X6. Additionally, the controls have been modified to allow for the dynamic adjustment of the toolpath's total tolerance. At any time, you may enter a value in the **Total tolerance** field or change the values in the **Cut**, **Line/Arc**, and/or **Smoothing** tolerances. Mastercam will automatically calculate and update the ratios as necessary.

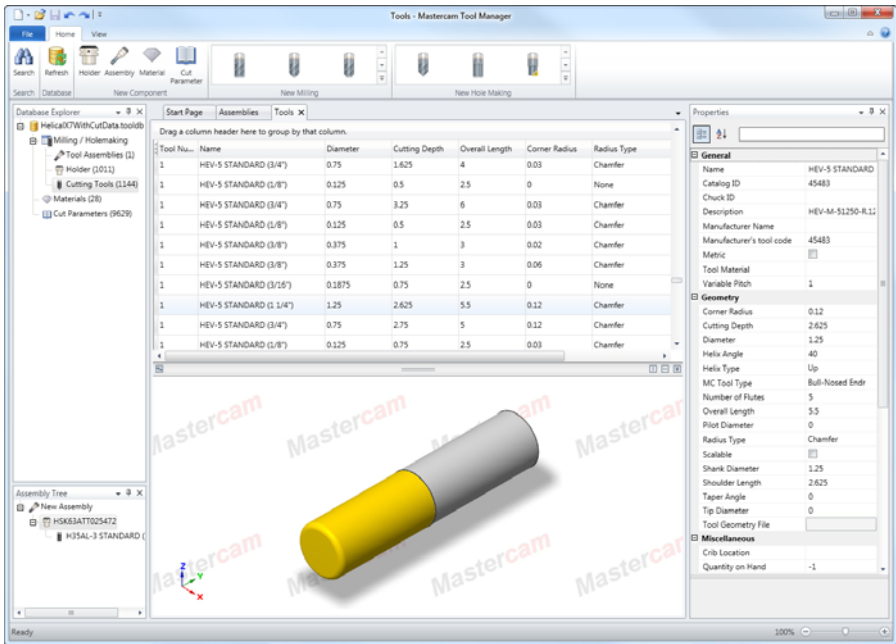


Mill Tooling Enhancements

Workflow Improvements

New Tool Manager

Mastercam's new Tool Manager provides a whole new way to organize and create your tooling. This application offers a lot of power and flexibility for all your mill and router tools. Tool Manager is launched separately from Mastercam by choosing **Start, Programs, Mastercam X7, Tool Manager**.

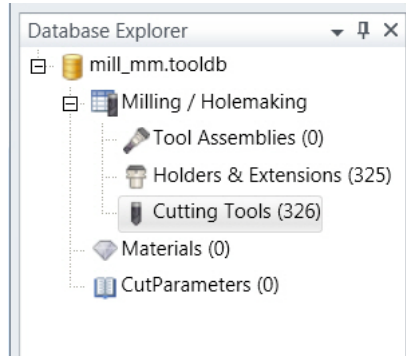


The new Tool Manager has been designed from the ground up as a flexible and efficient way to manage tools and tool holding components and to create tool assemblies that can be used in Mastercam. In addition to the tooling component support, Tool Manager also integrates work material and cut parameter data so that you can take advantage of a manufacturer's cutting recommendations.

The Tool Manager workspace is made of several key components:

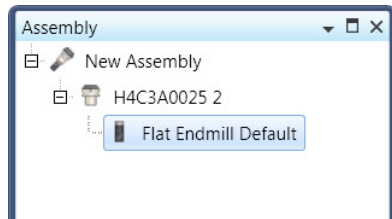
Database Explorer

The *Database Explorer* provides a view of the active database, including tools, holders, and assemblies. Each instance of Tool Manager displays only one tool database at a time. You can have multiple instances of the application and even drag and drop components between instances. Double-clicking on the tree items in the Database Explorer opens a new document or sets the active document.



Assembly Tree

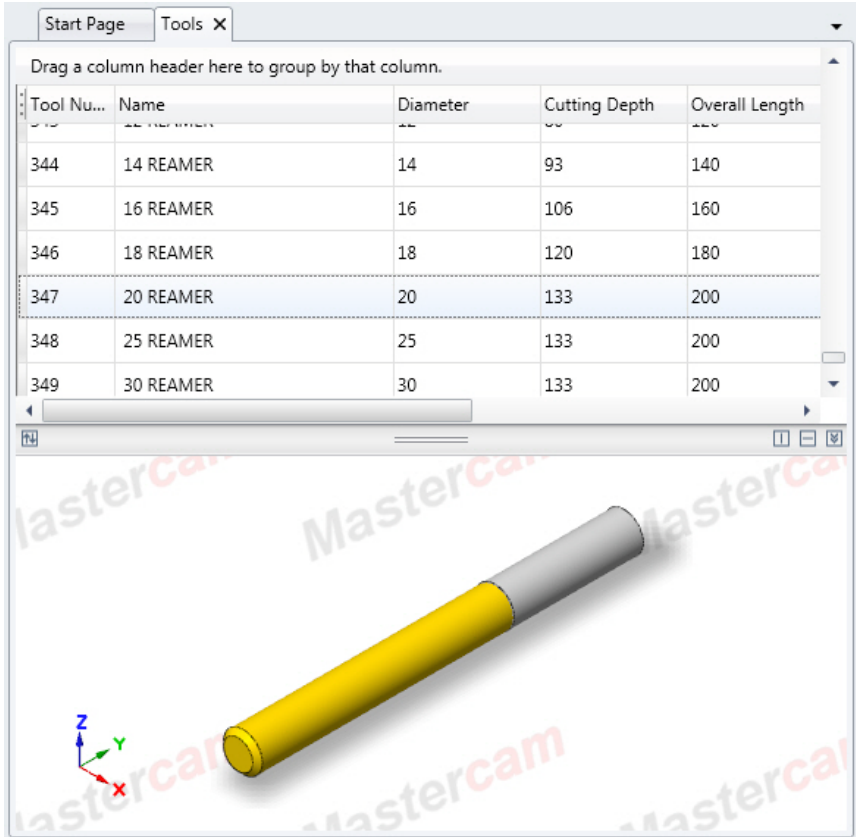
The *Assembly Tree* displays a hierarchical representation of the active tool assembly. Items in the tree start from the spindle side of a tool assembly and move progressively closer to the cutting tool.



Document Workspace

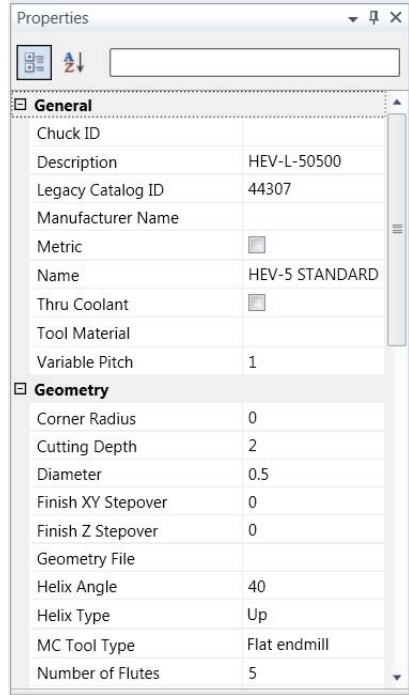
The *Document Workspace* shows each document type as a collection of items. All documents offer a data grid view which supports sorting, grouping, and selection of one or more items. Below the data grid is a 3D display of the selected tool or holder.

For tool assemblies, this display area also provides selection and adjustment of tool component positioning and orientation.



Properties Grid

All properties available for a selected item are displayed in the *Properties Grid*. Use the data grid to select one or more items for editing. When multiple items are selected, only common properties display in the properties grid. These properties can be edited for the entire group of selected items.

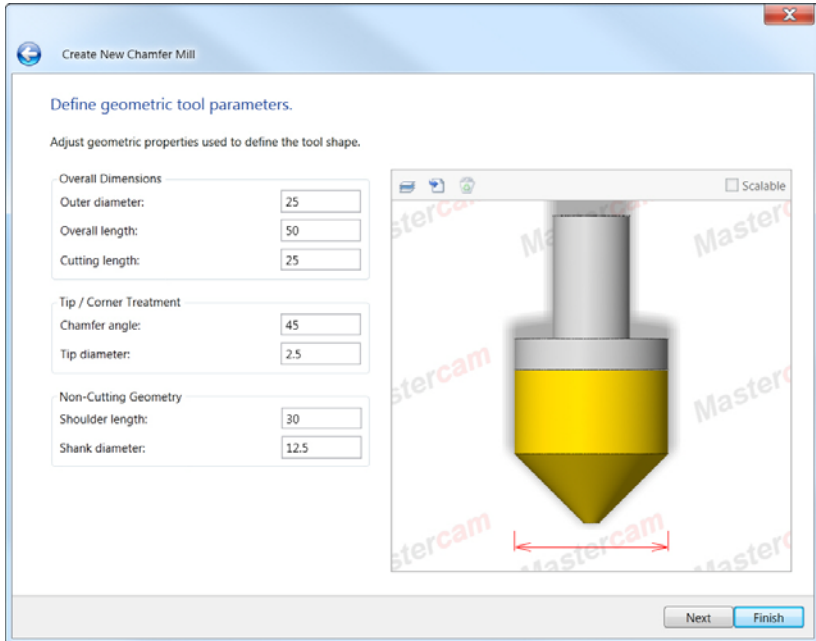


Notes:

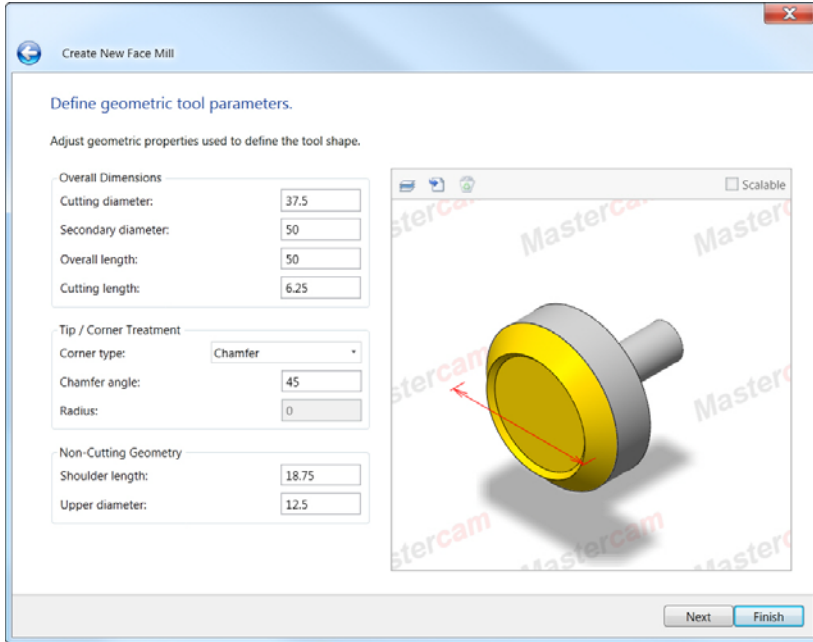
- To access *Help* throughout *Tool Manager*, click the *Help* button or press [F1].
 - You can hover over the ribbon bar buttons for detailed function explanations and shortcut keys.
-

New Tool Wizard

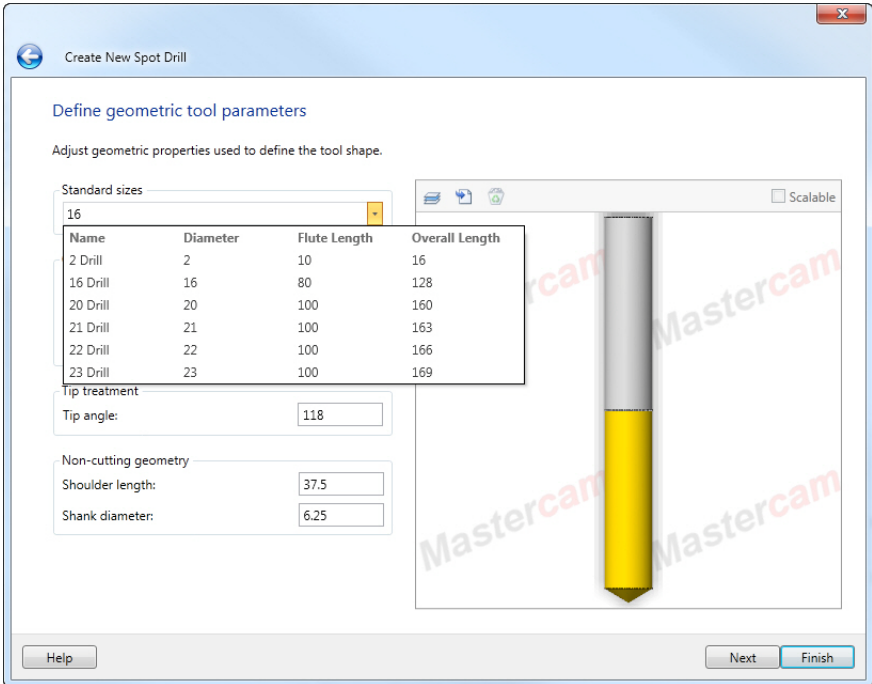
The new *Tool Wizard* makes the process of creating a milling or holmaking tool much simpler. You can access the *Tool Wizard* by right-clicking in your operation's tool parameters and choosing **Create new tool**, or by selecting the **Home** ribbon bar in *Tool Manager* and selecting one of the tool types.



The intuitive interface guides you through the process and visually represents your adjustments. You can right-click in the graphics area to view your tool from different angles. Also, the aspect that you are editing is marked with red dimension lines.



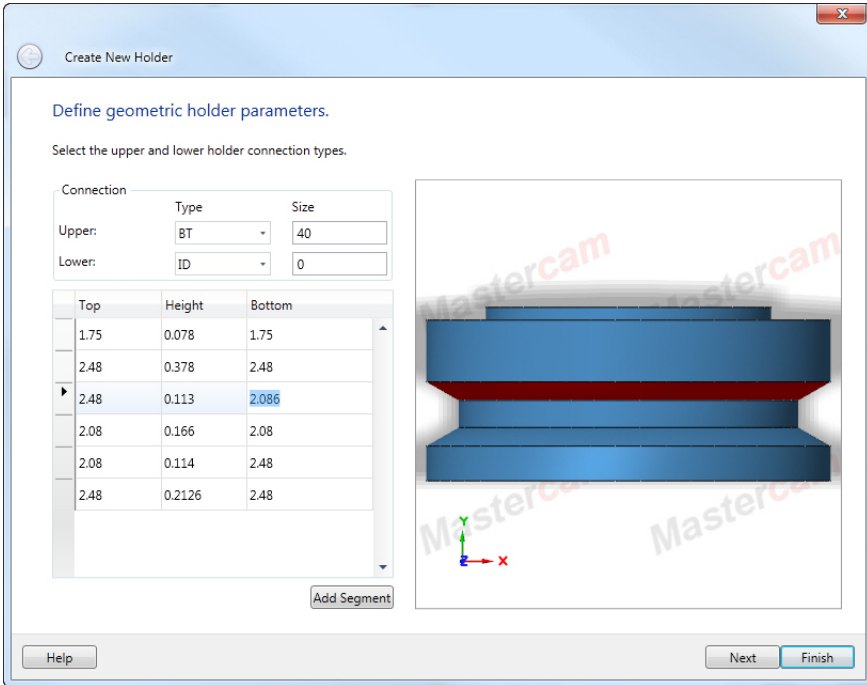
When creating a holmaking tool, you can type a standard drill size or common diameter to quickly filter through the list of possible tools.



New Holder Wizard

The new Holder Wizard looks very similar to the Tool Wizard. You can access it by going to the **Holder** page in your operation and clicking the **New Holder** button or

by selecting the **Home** ribbon bar in Tool Manager and clicking the **Holder** button in the ribbon bar.

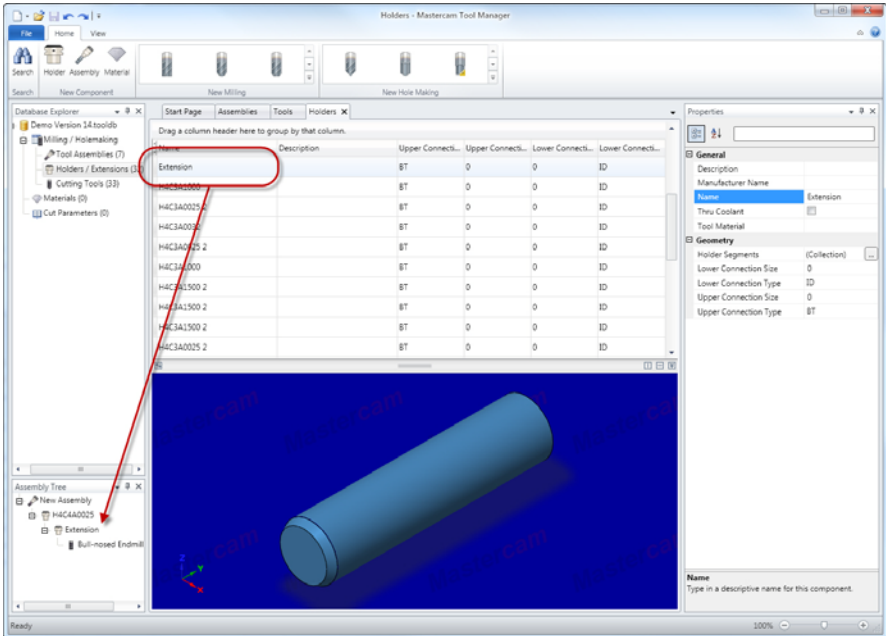


Mill Tool Assemblies

Multiple Holder Component Support

The new Tool Manager application lets you create tool assemblies that include combinations of holders and extensions. You can use extensions to stabilize a long tool or extend the tool length for certain areas of your part.

Drag an extension from your Holders page in Tool Manager and position it in the appropriate place in the Assembly Tree.

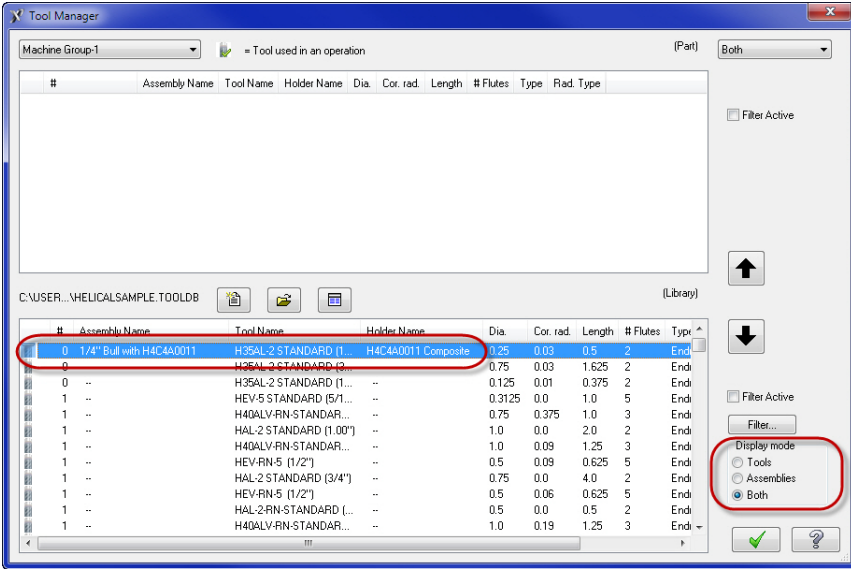


You can also drag and drop within the Assembly Tree to reposition the extension.

Saving to a TOOLDB Library

Mastercam .TOOLDB files now support mill tool assemblies within the library. When you open a .TOOLDB file from the Tool Manager within Mastercam X7 (Toolpaths,

Tool Manager), you can select to view both the tools and assemblies in the library.



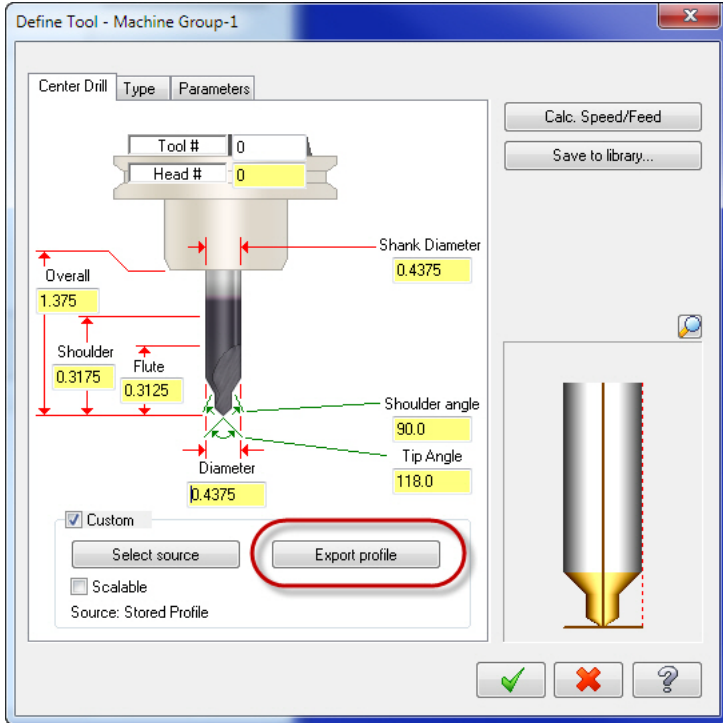
Custom Tool Improvements

Store Profile Data in Document File or (TOOLDB) Library

To allow for easier part transfer between Mastercam users, custom tool profile information can now be stored directly in your Mastercam part file. The new method also addresses the issue of someone changing a library and it silently affecting previously programmed part files. If a library is updated and you revisit a previously programmed file, you can simply re-select the updated tool and the operation will be updated (and marked dirty). Otherwise, the original profile that you programmed against is still considered valid and remains unchanged.

Export to a Mastercam Level

One method for keeping your custom tool profile with your Mastercam part file is to export the geometry to a level in Mastercam X7. In the Define Tool dialog box, click **Export profile** in the Custom section.

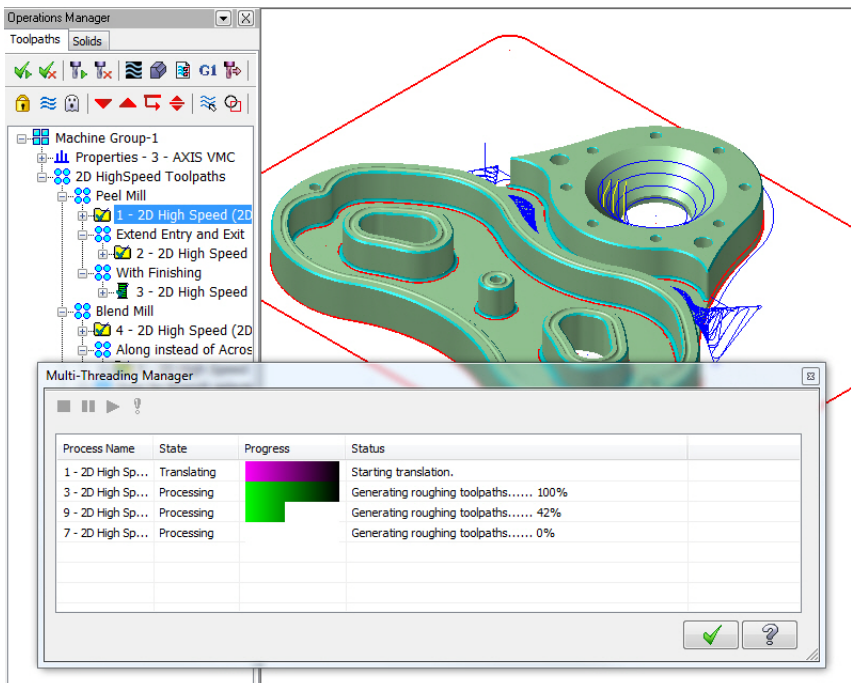


Select a level for the tool geometry and click **OK**. You can then edit the tool profile on that level.

2D Mill Enhancements

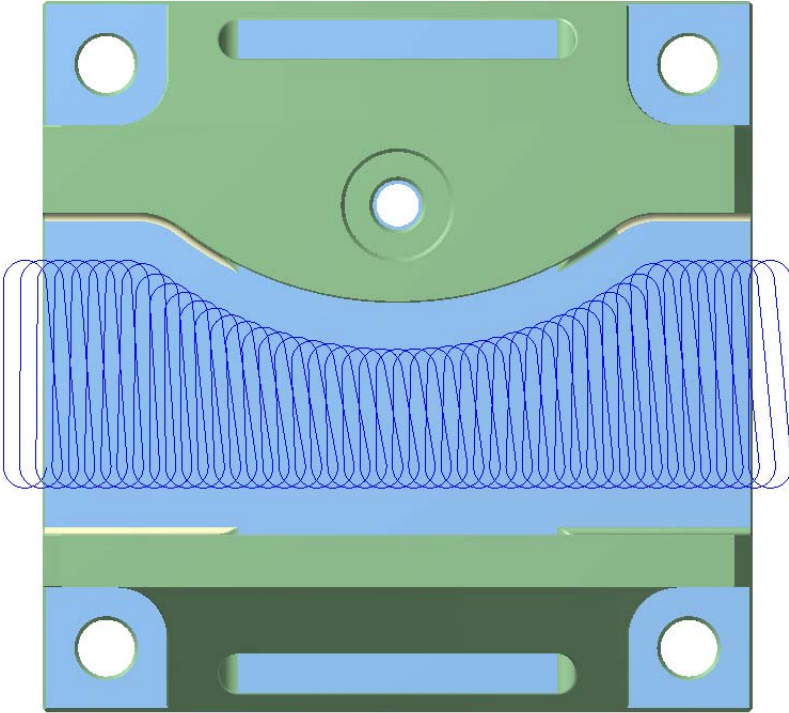
Multi-threading Support for 2D HST Toolpaths

The same multi-threading support that has been available for 3D HST and Multiaxis toolpaths is now extended to the 2D HST toolpaths, except for Blend. This processing mode allows Mastercam to split the program regeneration into smaller, separate program threads that process simultaneously. The result is faster toolpath processing that takes full advantage of today's multi-core processors.

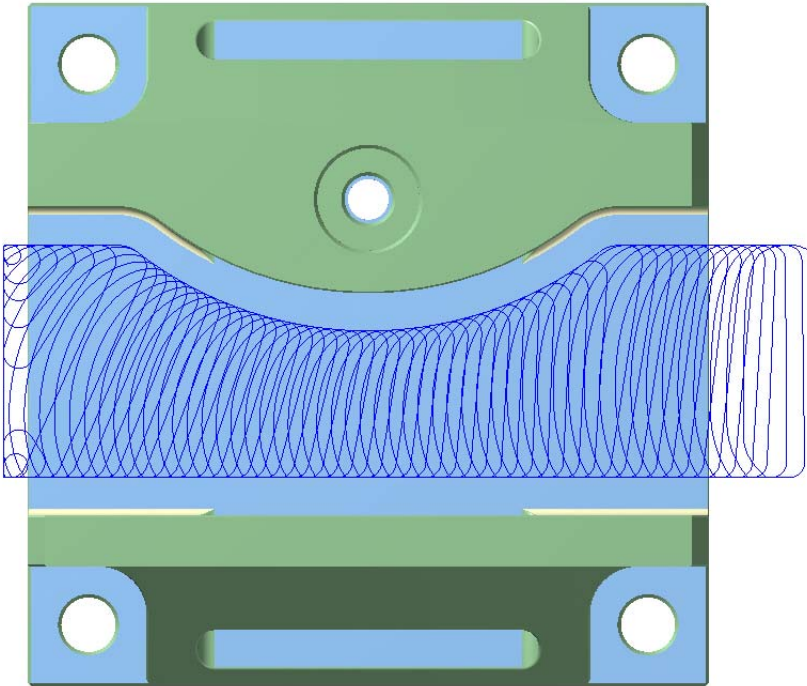


Dynamic Motion for 2D HST Peel Mill

This new tool motion in 2D HST Peel Mill toolpaths allows Mastercam to automatically adjust based on tool engagement. This makes your tool motion more efficient, especially for high speed machines.



X6 HST Peel Mill

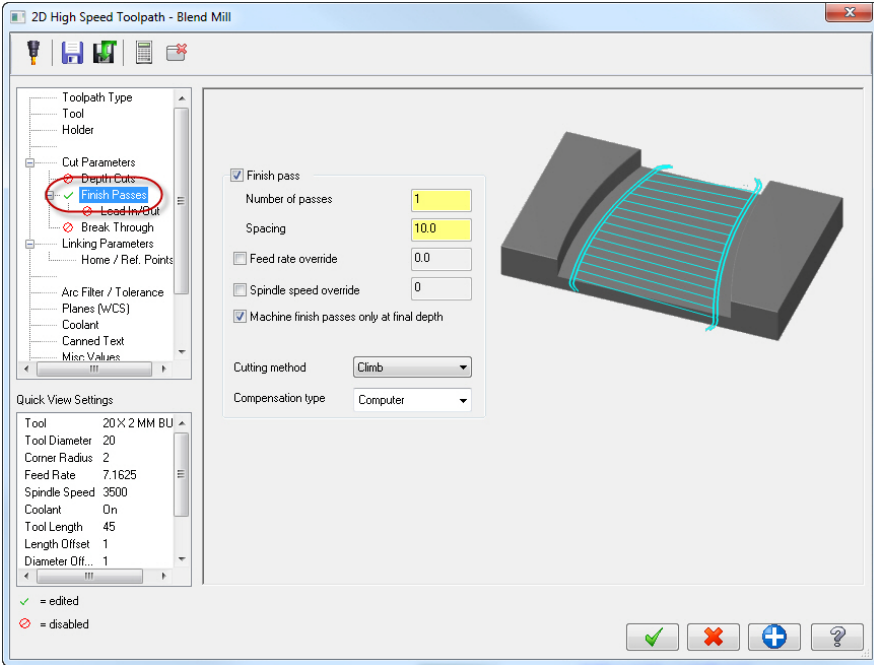


X7 HST Peel Mill

Blend Mill Improvements

Finish Passes

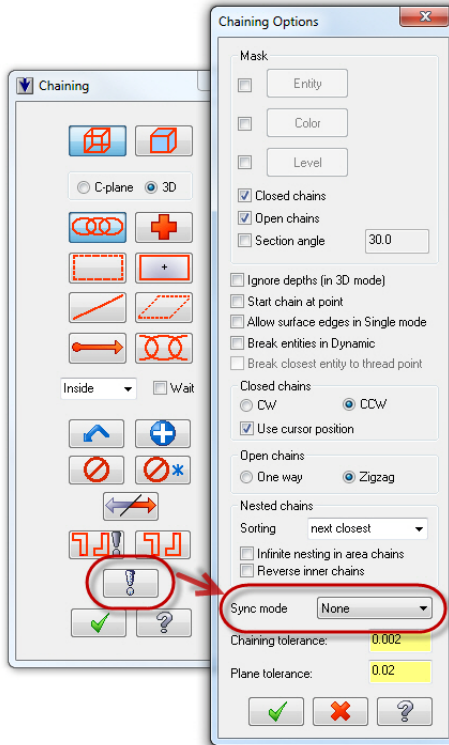
Similar to finish passes in many other Mastercam toolpaths, finish passes were added to Blend Mill for X7. Adding finishing passes along the selected boundaries of the toolpath can improve the final finish quality of your part.



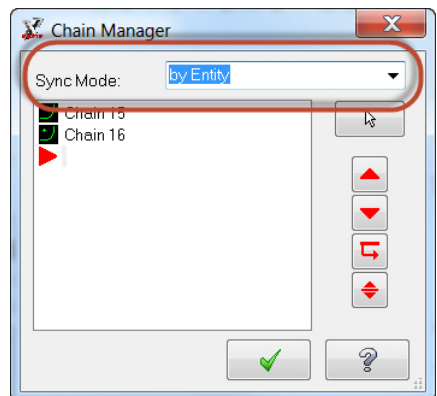
Synchronized Chain Support

Mastercam X7's Blend Mill toolpath now includes additional controls to produce exactly the tool motion you want for your part. Just like the chain synchronization options that are available in Mastercam Wire, you can now adjust your toolpath

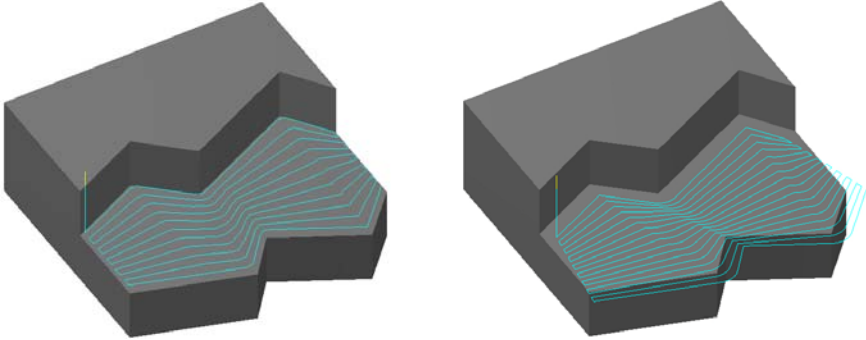
motion by synchronizing your two selected chains. You apply the synchronization method when you first select your chains (in the Chaining Options dialog box).



You can also adjust the sync mode within the Chain Manager for an existing toolpath to see how different sync options may adjust your toolpath.

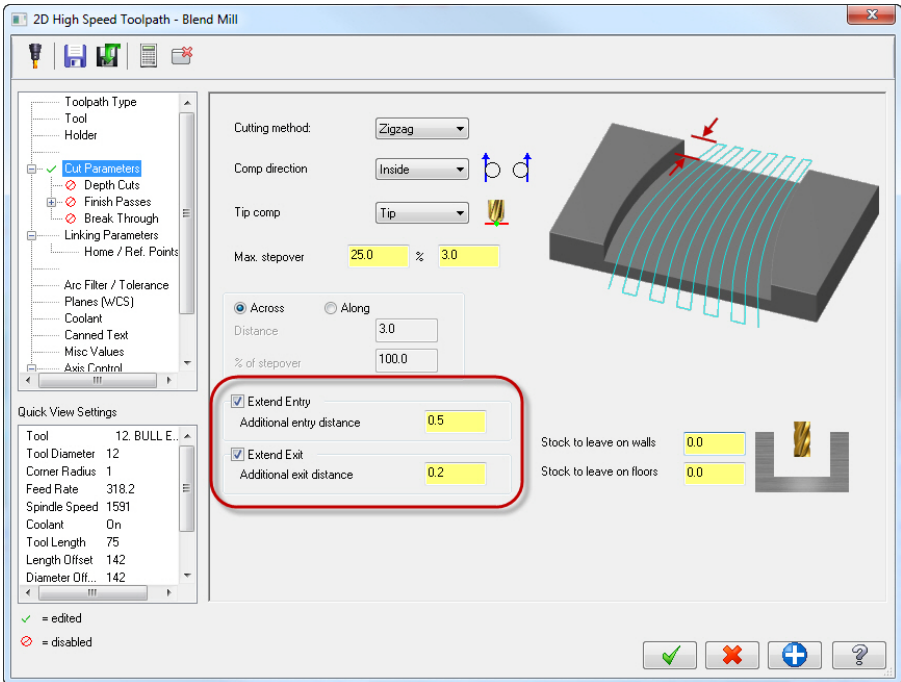


The following graphics show the same part with no sync and with syncing by entity.



Extend Entry/Exit

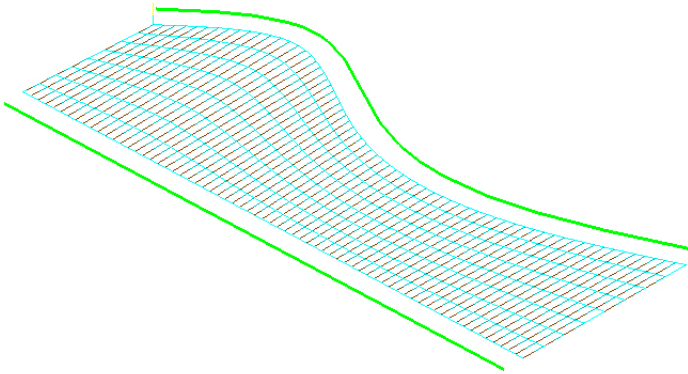
Use these Blend Mill fields to adjust the initial and final tool engagement with the material. You can select either field and enter an additional entry/exit distance.



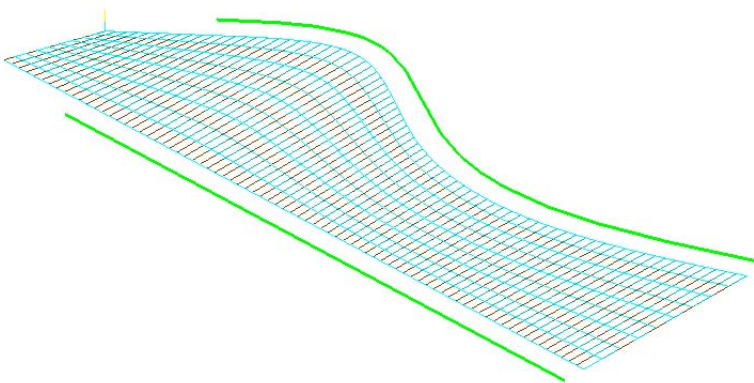
By default, the initial cut is defined by the stepover amount. When you select the Extend entry/exit fields, Mastercam adds the additional amount you specify to the beginning or end cut.

These options are useful if the part geometry between the two chains is something other than a straight line, such as a slot that spans the corner of a part.

Extend entry off:



Extend entry on:



Relaxed Chaining Restriction for Slot Mill Toolpaths

In previous versions of Mastercam, the Slot Mill toolpath required that your chain was comprised of two parallel lines connected by two 180 degree arcs. In Mastercam X7, the Slot Mill toolpath accepts any chain with two parallel sides and a closed boundary.

New Finish Pass Directions for Helix Bore Toolpaths

In previous versions of Mastercam, Helix Bore finish passes were limited to only bottom to top generation. In Mastercam X7, you now have the ability to select how finishing passes are created. On the toolpath's Rough/Finish parameters page, select a finish method from the three following options in the drop-down menu.

- **Helix upwardly:** The tool feeds to the finish bore dimension and then generates a helix from bottom to top.
- **Helix downwardly:** The tool retracts to the top of stock, moves to the finish bore dimension and then generates a helix from top to bottom.
- **Circle:** The tool feeds to the finish bore dimension and creates a single circular cut at the bottom of the hole.

3D Mill Enhancements

General 3D Mill Improvements

Multi-Core Support

3D HST Raster, Radial, and Spiral toolpaths now include multi-core processing to speed up your toolpath regeneration.

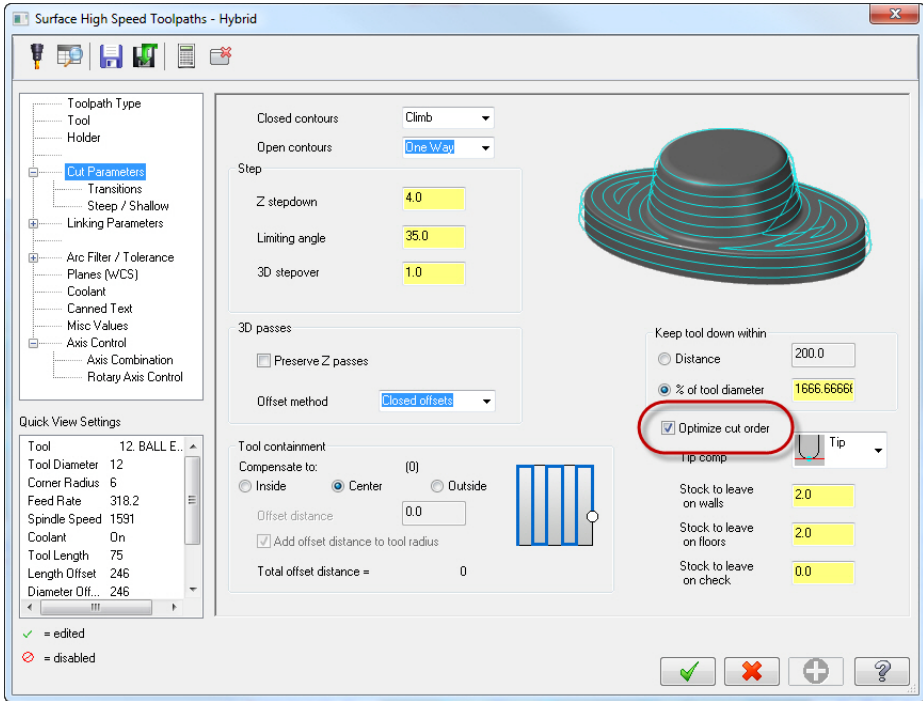
Check Surface Support

Raster, Waterline, Spiral, Radial, and Hybrid toolpaths in Mastercam X7 now offer a method for selecting surfaces that your tool motion should avoid. This improved toolpath control functions just like other surface toolpaths in Mastercam.

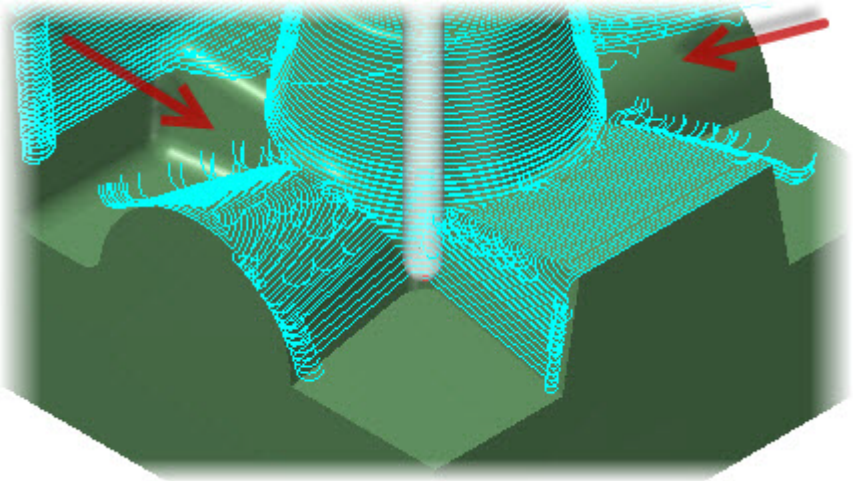
3D HST Hybrid

Optimized Cut Order

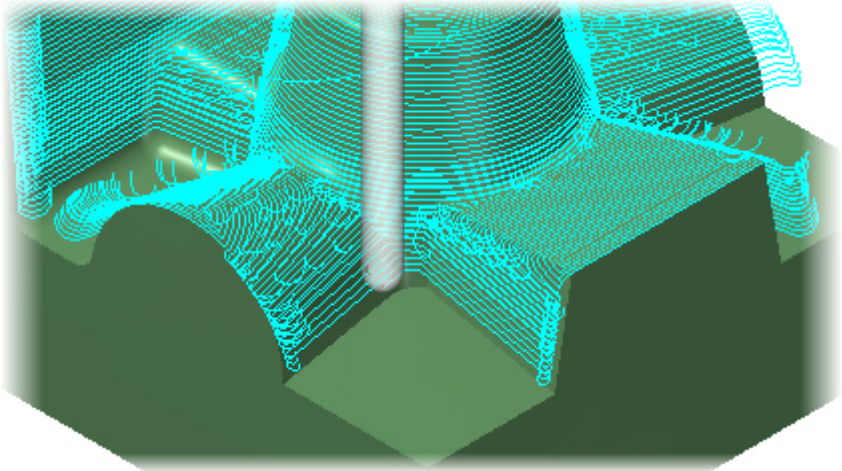
In Mastercam X7, 3D HST Hybrid toolpaths now include a new **Optimize cut order** checkbox on the Cut parameters page. This option defines the cut order Mastercam applies to different cutting passes in the toolpath.



When selected, Mastercam works in tiers, ordering the toolpath's cutting passes based on proximity and safety. Mastercam machines features by area until it reaches a common Z-level, then repeats the process until it reaches the next common ground. The tool stays in a common ground, which reduces unnecessary motion and shortens the length of the toolpath.

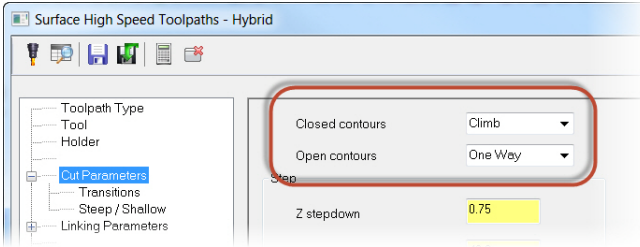


When deselected, Mastercam machines all cut passes Z-level by Z-level.

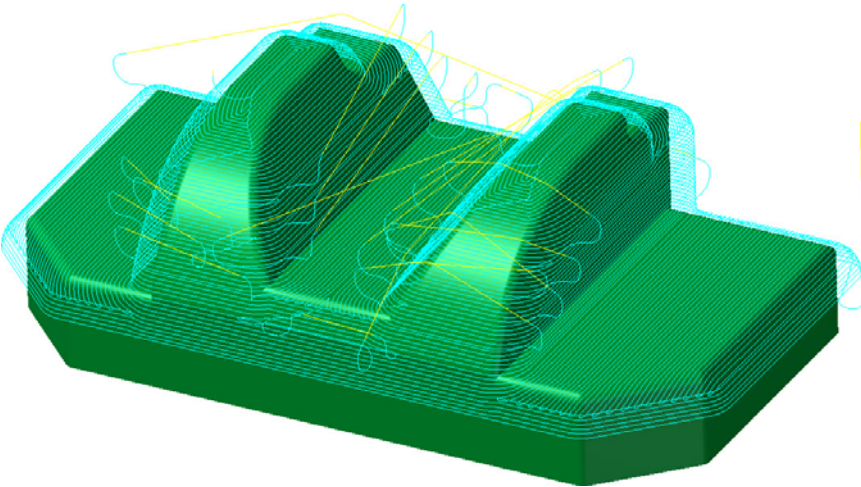


Open/Closed Passes

Hybrid toolpaths now offer additional options for handling open and closed contours.

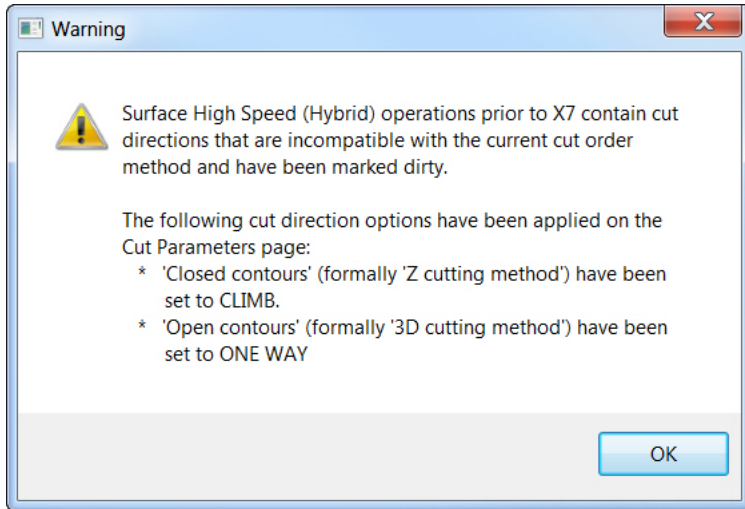


Closed contours contain continuous motion without a need for a retract or reversal of direction. You can select climb or conventional machining for these contours. Open contours offer **One Way** or **Zigzag** cutting options. Open contours set to One Way will use the cutting method selected for closed contours.





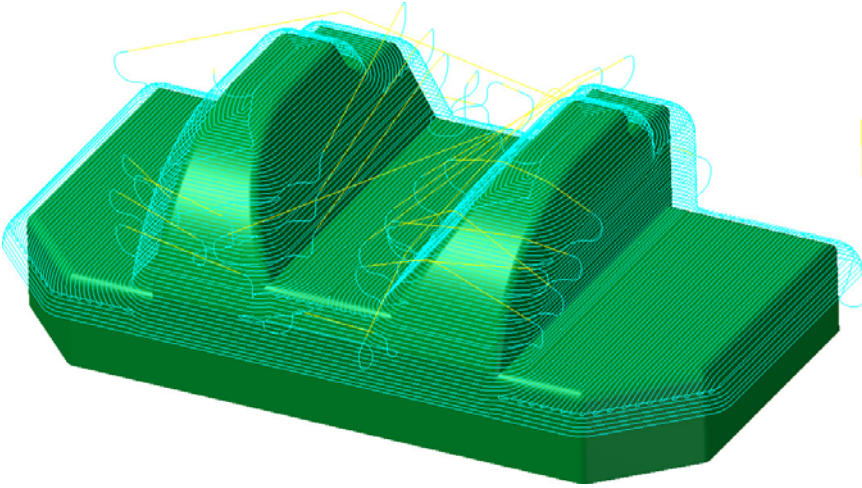
IMPORTANT: If you are opening an X5 or X6 file that contains a Hybrid toolpath, you may encounter the following message that explains how Mastercam X7 is handling these parameter changes:



Click **OK** to continue and regen your toolpath in the Operations Manager.

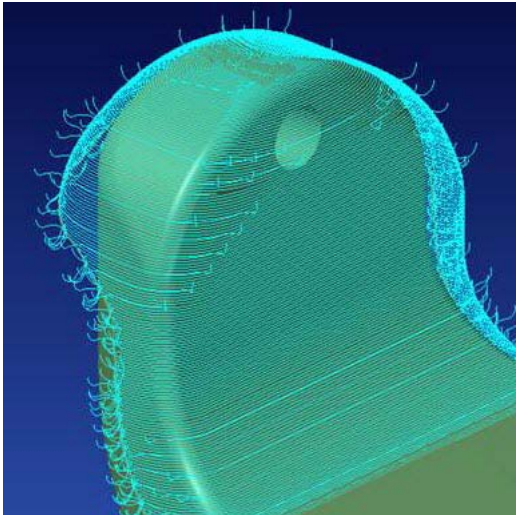
Automatic Offset Method

The new Automatic offset method controls how Mastercam handles the boundaries between steep and shallow areas. If the toolpath engages part surfaces that include core and cavity properties, Mastercam applies upper to lower offset methods to core features and lower to upper offset methods to cavity shapes.

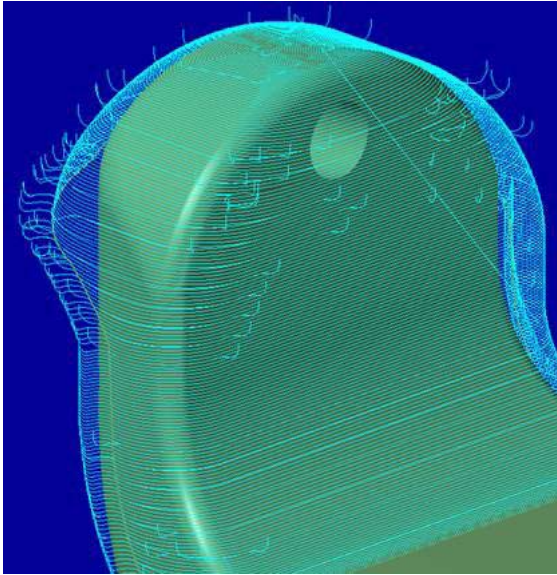


Improved Motion Near Boundaries

3D HST Hybrid toolpaths calculate more efficient high-speed motion at your part's boundaries.



X6 Hybrid motion



X7 Hybrid motion

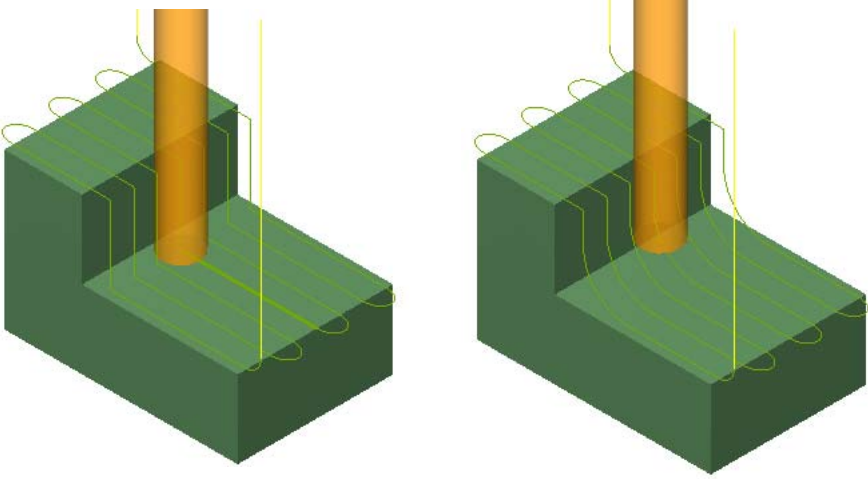
Smarter Regeneration

To improve regeneration speed, Hybrid toolpaths now only recalculate the necessary sections of the toolpath instead of processing the entire toolpath when changes are made.

New 3D HST Toolpath Fillet Page

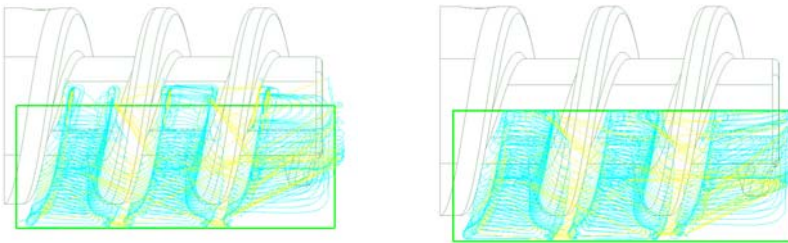
Use this page to have Mastercam automatically generate fillet motion in your high speed toolpath. Toolpath filleting allows you to create smooth toolpath motion while maintaining a high feed rate. Generate the toolpath fillet based on a simple radius value or by entering tool information to control the fillet. The fillet motion is gener-

ated only on internal corners. Your part geometry remains unaltered, however, your toolpath contains smoother motion, as shown in the example below.



Containment Boundary Improvements

When selecting chains for Area Clearance, OptiArea, Rest, and OptiRest toolpaths, you can now specify chains to restrict tool motion. The two images below show X6 versus X7 results.



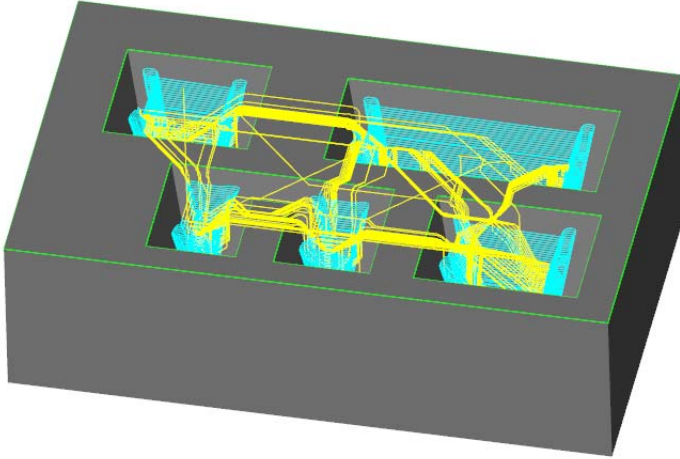
3D HST OptiCore and OptiArea Toolpaths

In addition to general improvements to the tool motion for these toolpaths, the following enhancements are included in Mastercam X7.

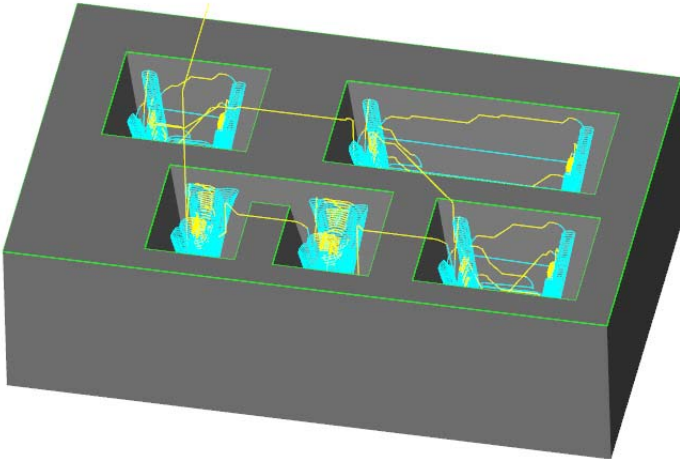
Optimized Cut Order for Step Ups

New **Optimize cut order** options on the Cut Parameters page allow you to define the cut order Mastercam applies to different cutting passes in the toolpath. From the drop-down menu, you can now choose to machine cut passes by depth, by proximity, or by pocket.

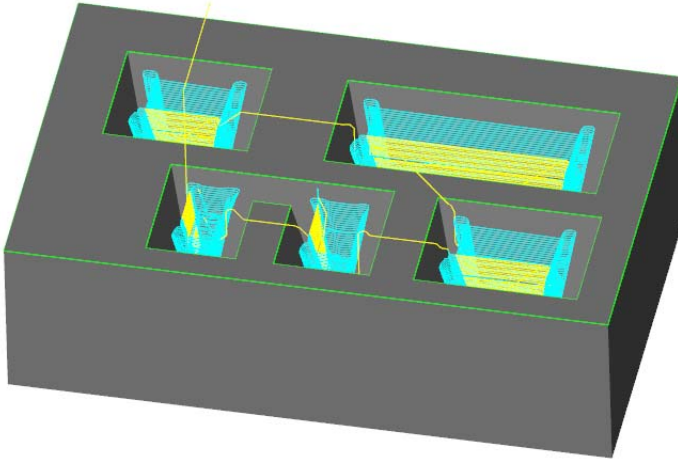
- **By Depth:** Mastercam machines all cut passes z-level by z-level.



- **Next closest:** Mastercam moves to the closest cut from where it finished the previous cut.



- **By pocket:** Mastercam first machines all the step downs, moving from pocket to pocket. After all step downs on a Z-level are machined to completion, Mastercam machines the step-ups by next closest, in the safest cut order.



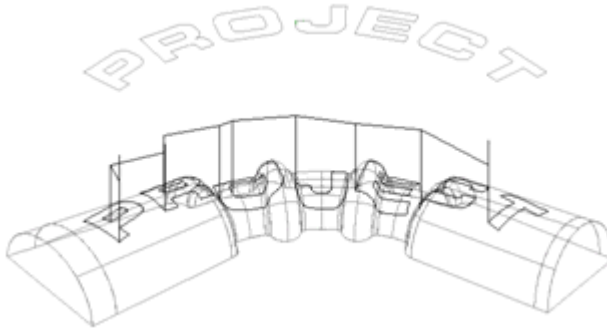
Safer Retract Motion Over Material

Retract motion over your part has been enhanced to address parts where you have not specified material.

New 3D HST Project Toolpath

Project toolpaths project either geometry or a toolpath from an earlier operation onto surfaces. In previous versions of Mastercam, the Project toolpath was only available as a surface roughing toolpath. In Mastercam X7, the full suite of high speed toolpath parameters are available to refine the tool motion.

Projecting points onto surfaces can create plunging motion to clear small areas in detailed parts. A common use of project is engraving created by projecting curves onto surfaces as shown below:



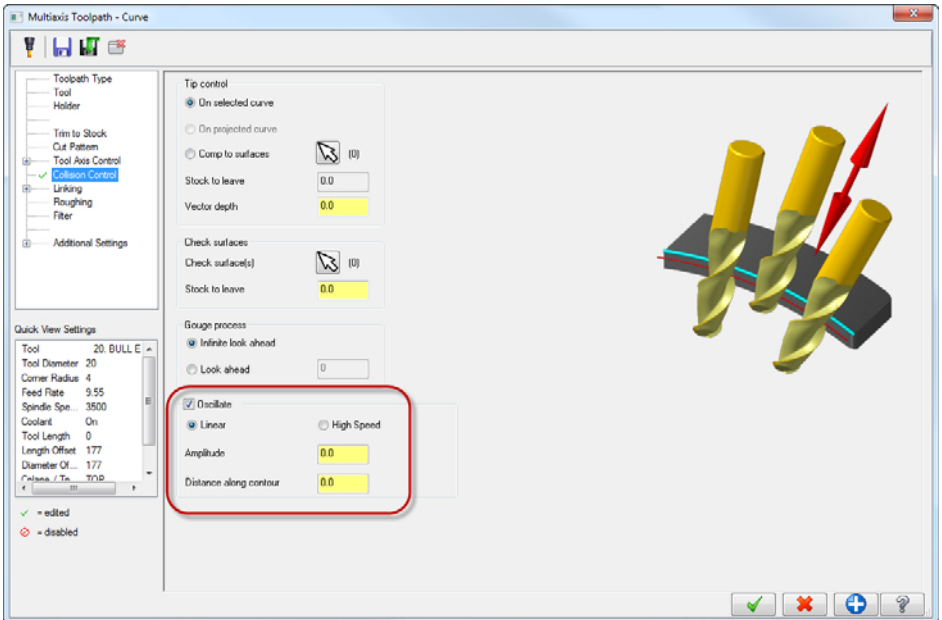
There are two ways to access the new 3D HST Project toolpath:

- Choose **Milling, 3D High Speed**, and select **Project**. (Mill-Turn machine types)
- Choose **Toolpaths, Surface High Speed**, and select **Project**. (All other machine types)

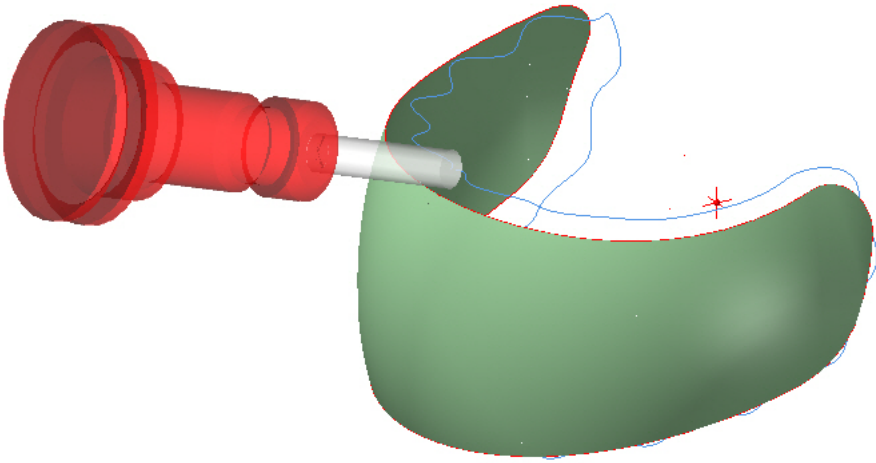
Multiaxis Mill Enhancements

Oscillate Motion for Curve and Swarf 5-axis

Oscillating tool motion improves your tool life by not always machining with the same area of the tool. This motion is particularly useful for trimming work. The Curve 5-axis and Swarf 5-axis toolpaths now include an **Oscillate** option on the Collision Control page.



You can select either linear (sharp direction changes) or high speed (smooth direction changes) motion as your tool moves through the toolpath.



Port Expert Enhancements

Mastercam's Port Expert add-on creates accurate, efficient head porting toolpaths on port surface or solid data. Port Expert uses a tapered lollipop tool, and has collision checks for the entire tool (shank, arbor, and holder). Mastercam X7 includes the following additional features:

- Toolpath generation for non-planar port openings.
- Additional controls for toolpath resolution.

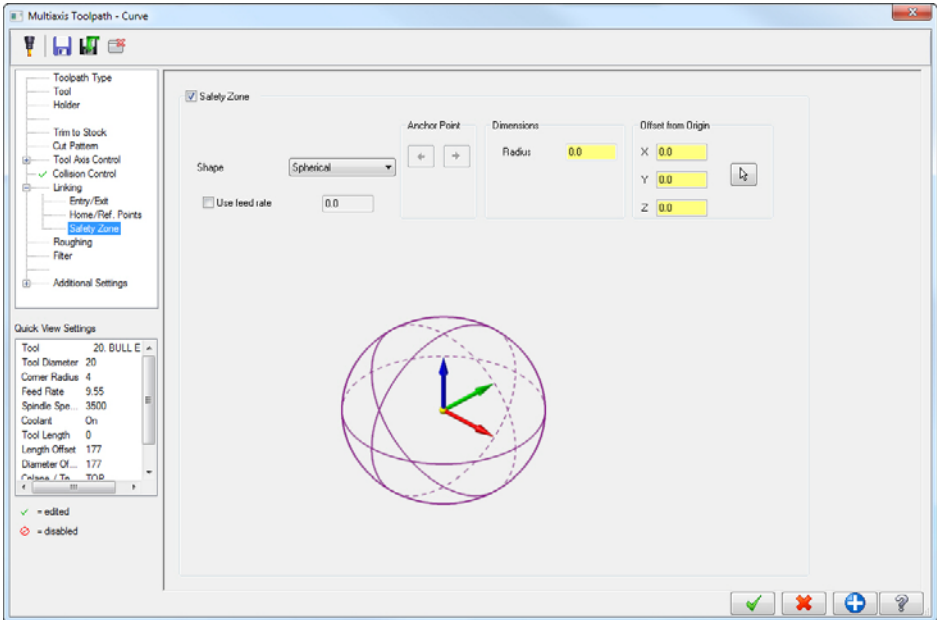
Blade Expert Enhancements

Mastercam's Blade Expert is specifically designed to generate the necessary toolpaths for many different multi-blade configurations. Mastercam X7 includes the following new features for Blade Expert:

- Support for straight and bull-nose tools.
- Support for Mastercam's Stock Model for more efficient rest machining.

Independent Safety Zone for Classic Multiaxis Toolpaths

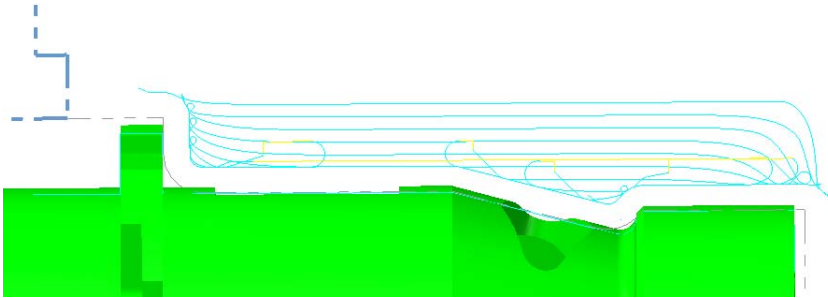
Setting a safety zone for Classic Multiaxis toolpaths has been moved from the Machine Group Properties to a new page in the toolpath dialogs. This change provides you with more control for motion in each individual toolpath instead of setting a safety zone that applies to all toolpaths in your part.



Lathe Enhancements

New Dynamic Rough Toolpath

The Lathe Dynamic Rough toolpath is designed for hard materials cut with button inserts (i.e., radius or ball). The dynamic motion allows the toolpath to cut gradually, remain engaged in the material more effectively, and use more of the surface of your insert, extending the tool life and increasing the cutting speed.

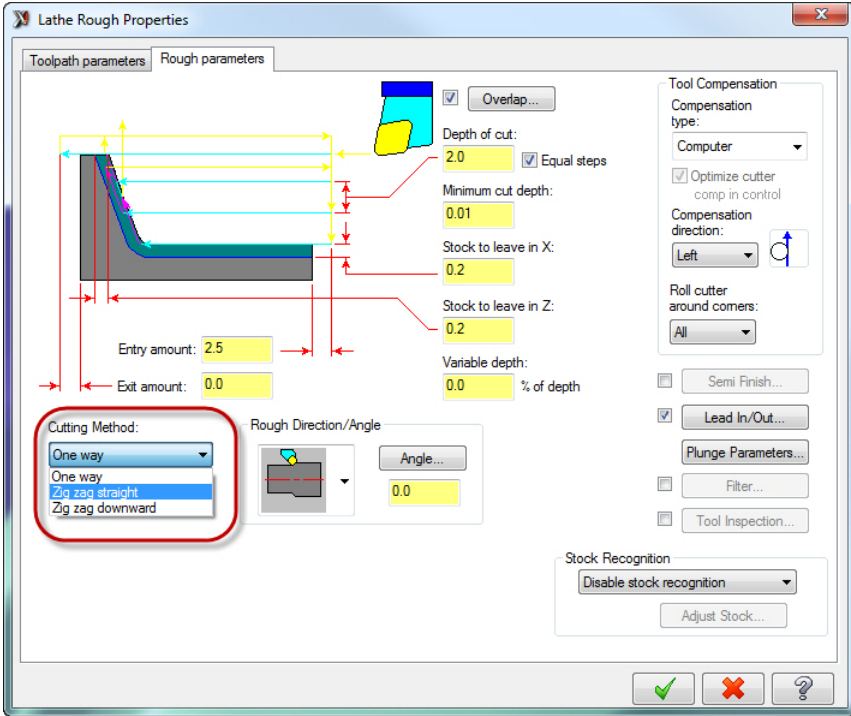


Notes:

- The lead in and lead out moves are used only on the semi finish passes.*
 - Stock must be defined prior to selecting this toolpath.*
 - Only button inserts (i.e., radius or ball) are supported.*
 - Cutter compensation type can only be set to Computer or Off. Control, Wear, and Reverse Wear are not supported.*
-

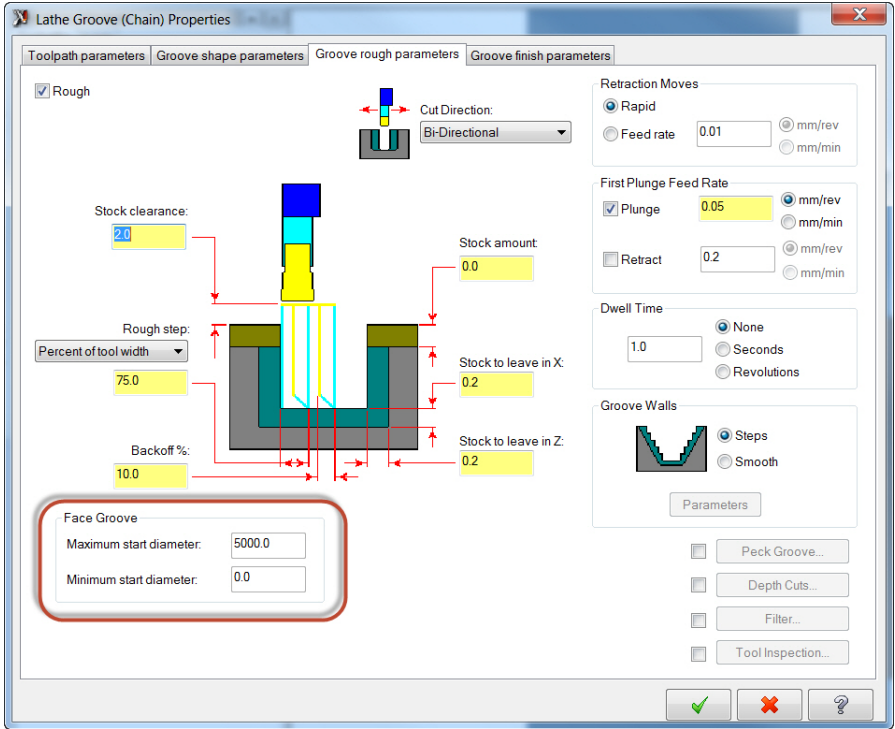
Rough Zigzag Motion Options

These new options for Lathe Rough toolpaths provide you with more control of your zigzag tool motion. You can choose to move back and forth straight across, or ramp downward with each tool pass.



Face Grooving Start Range Option

These new grooving parameters allow you to position your tool accurately for face grooving. Enter a maximum and minimum start diameter for positioning the first groove cut.



Full Radius Tool Support for Plunge Turn

In previous versions of Mastercam, the Plunge Turn toolpath only supported flat bottomed tools. In X7, full radius tool support is available.

**Attention! Updates may be available.
Go to Mastercam.com/Support for the latest downloads.**



cnr software, inc.

671 Old Post Road
Tolland, CT 06084 USA
www.mastercam.com

Printed in the USA on 100% Post Consumer Waste, FSC Stocks