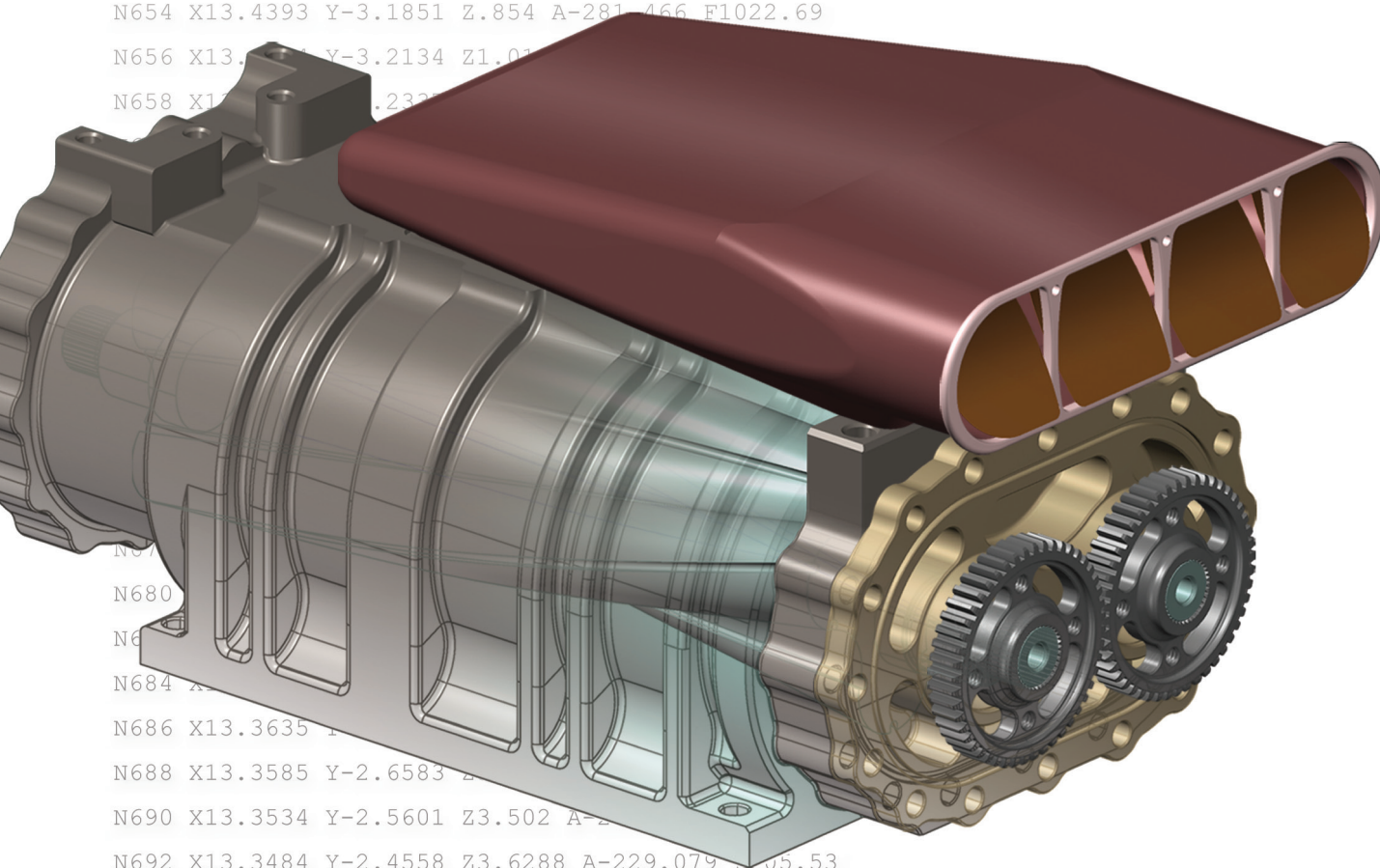
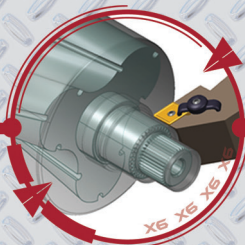
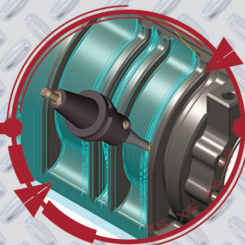
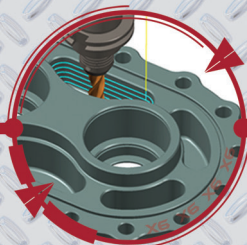


Mastercam X⁶

What's New



When Second Best Won't Cut It.



Mastercam® X6 What's New

Date: October 2011

Copyright © 2011 CNC Software, Inc.— All rights reserved.

First Printing: October 2011

Software: Mastercam X6

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:

www.mastercam.com/legal/licenseagreement/



What's New in Mastercam X6

General System Enhancements

64-bit Support

Mastercam X6 is available as a 64-bit application, which lets you take full advantage of your 64-bit hardware and operating systems.

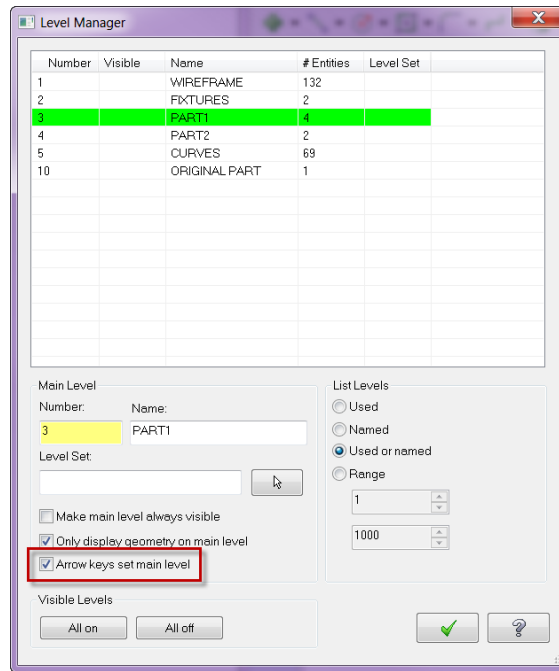
Level Manager Improvements

These improvements focus on level manipulations that are especially useful on parts imported from other systems where levels were not used. These enhancements will assist you in organizing the data so you can create toolpaths or perform other edits to the model.

On the Change Levels and Attributes dialog boxes, you now have control over the visibility of new levels you create for geometry. You can force new levels to be visible or invisible once the geometry is moved or copied.



In addition, you can control which level is the Main level using the arrow keys in the Level Manager. Select the **Arrow keys set main level** checkbox and use the arrow keys to move through the level list and change your Main level.



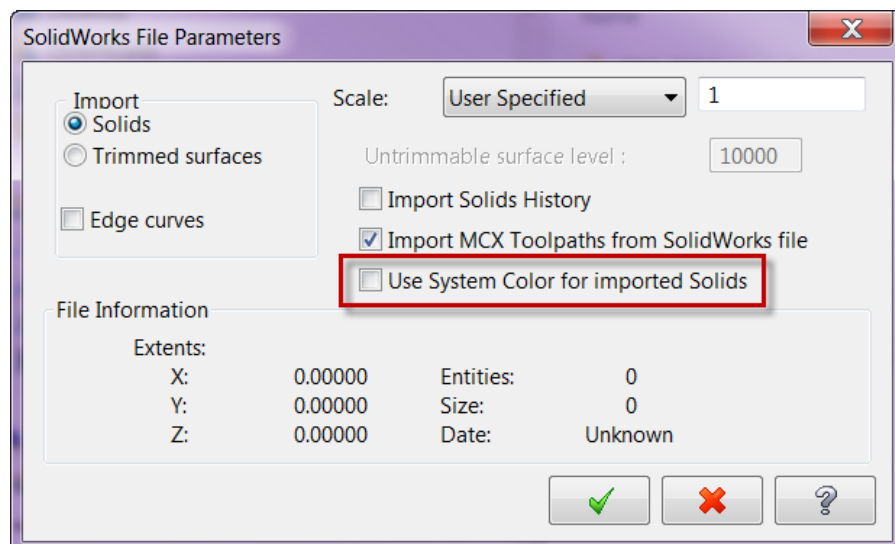
Quick Mask for Groups

This new Quick Mask provides a shortcut to the Groups Manager dialog box for geometry selection. Instead of selecting **All**, **Group Manager**, you can simply click this toolbar button and then double-click to quickly select all geometry assigned to a group.



Override Solid Face Color on Import

This new option, located in **File, Open, Options** or **Settings, Configuration, Converters**, forces the colors of imported solids to take on the current system color. This overrides any face or feature colors that have already been applied to a solid.



DWFX Import

Mastercam X6 now supports the .DWFX file format, which is an Autodesk file format based on the XML Paper Specification ([XPS\) format](#) from Microsoft. DWFX files can also be opened directly within Microsoft's XPS Viewer.

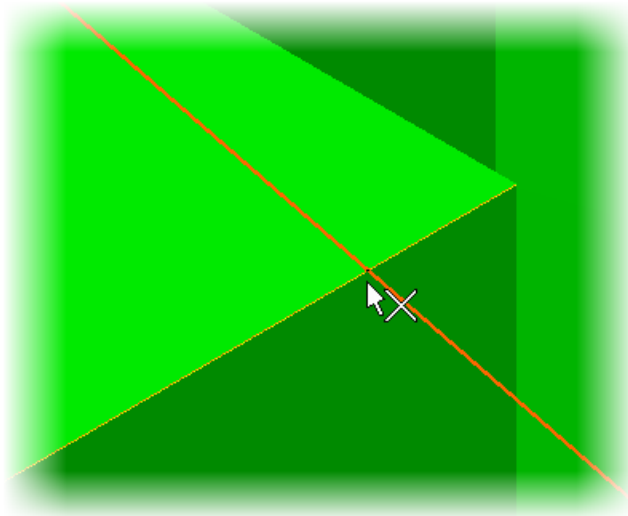
File Translator Updates

- **ACIS:** R22
- **Autocad:** Can now take .DWFX files
- **CATIA V5:** R21
- **KeyCreator:** 2011
- **Parasolids:** 23
- **ProE/Creo:** Wildfire5/Creo 1.0
- **SolidEdge:** ST4
- **SolidWorks:** 2012

Design Enhancements

Intersecting and Trimming Solid Edges

Mastercam's AutoCursor now recognizes intersections between wireframe geometry and solid edges without creating additional geometry first.

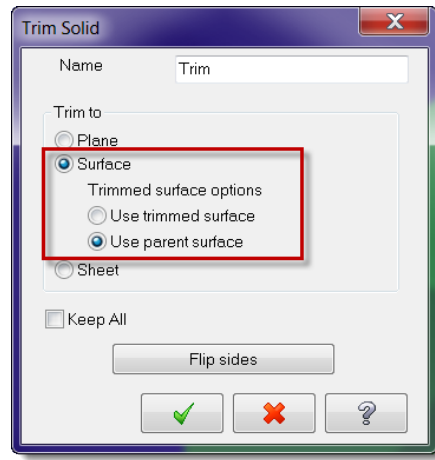


You can also trim or extend wireframe geometry to solid edges using the existing Mastercam trimming functions. Some examples include:

- Using **Edit, Trim/Break, Trim/Break/Extend** to trim the wireframe to one, two, or three solid edge entities.
- Selecting **Edit, Trim Many** to use a solid edge as the trim boundary
- Using **Edit, Trim/Break, Break at Intersection** to break wireframe where it intersects solid edges.

Solid Trim to Surface

The Trim Solid dialog box includes two new options for trimming solids to surfaces. These options give you more control over your trimming results.



Remove Solid History

The functionality from the NoHist C-Hook is now incorporated into the main Mastercam product. This function strips the list of operations or features from a solid model, leaving it as a "brick" in the Solids Manager. You can use this to prevent changes and protect your model. You can access this function by choosing Solids, Remove all solid history operations

Xform Fit

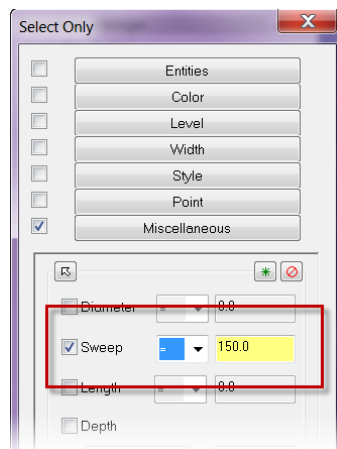
Xform Fit is a new function in Mastercam X6 that "nests" geometry along a vector, making it easier for you to make multiple copies between two defined points without knowing the exact distance between the points. This can be helpful when fitting parts between clamps and fixtures.

Trim/Divide Support for Splines

Mastercam X6 includes support for splines in Xform Divide for trimming curves or curves to be trimmed.

All and Only Mask – Arc Sweep

The All and Only masks now include the option to select arcs and circles with a sweep angle in mind. This is another way to speed up your geometry selection. For example, selecting a 0.5" diameter can be limited to only circles when diameter and arc sweep are used together. You can set these options on the **Miscellaneous** page of the Select All and Select Only dialog boxes.



Analyze Point or Area Using Feet/Inches

In the following Analyze functions, you can now see results in inches, feet/inches decimal, feet/inches fractional, millimeters, centimeters, or meters:

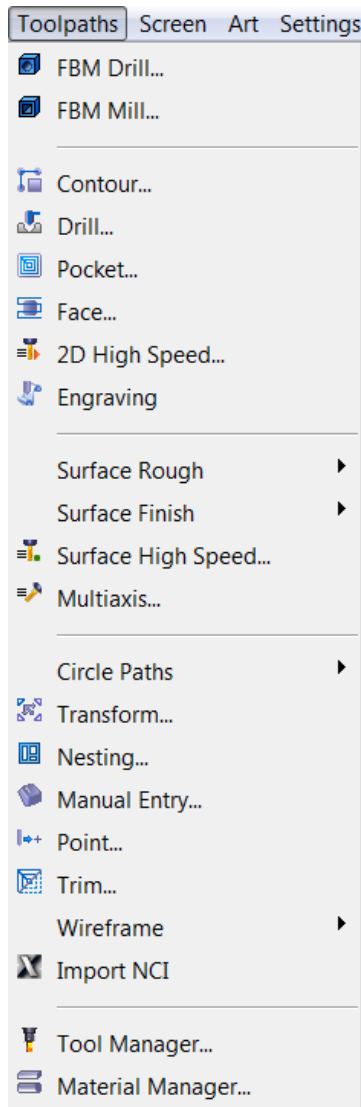
- Position
- Volume/Area, 2D Area
- Volume/Area, Surface Area
- Volume/Area, Solid Properties

General Mill/Router Enhancements

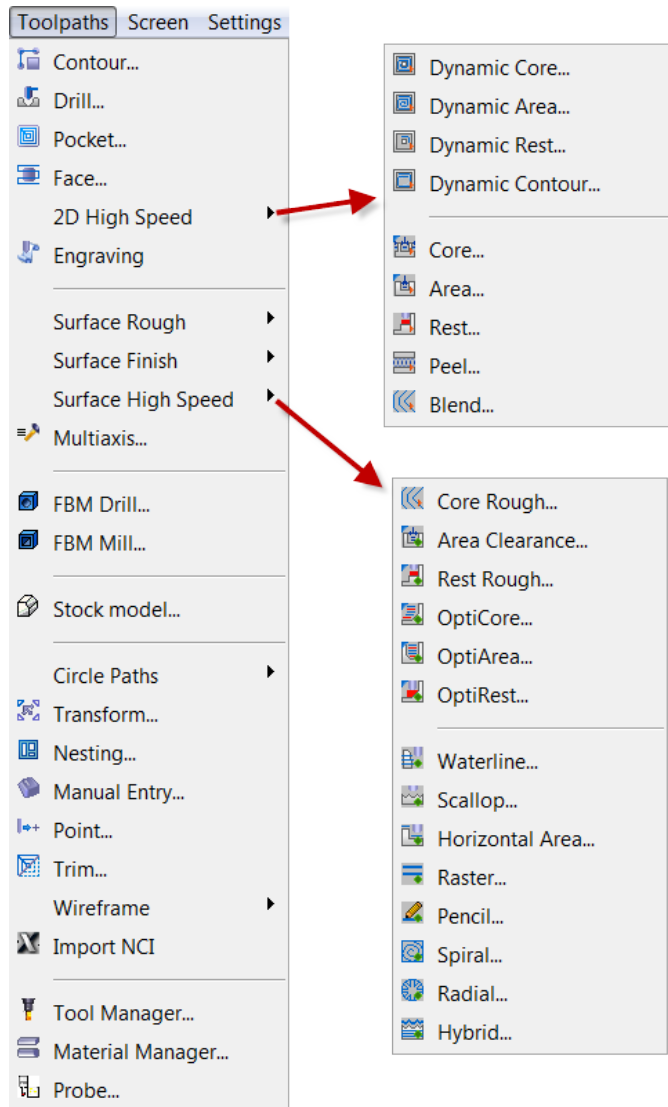
Mill/Router Toolpath Menu Reorganization

By restructuring the Mill and Router toolpath menus, you now have easier, faster access to all 2D and 3D high speed toolpaths.

Mastercam X5 Mill

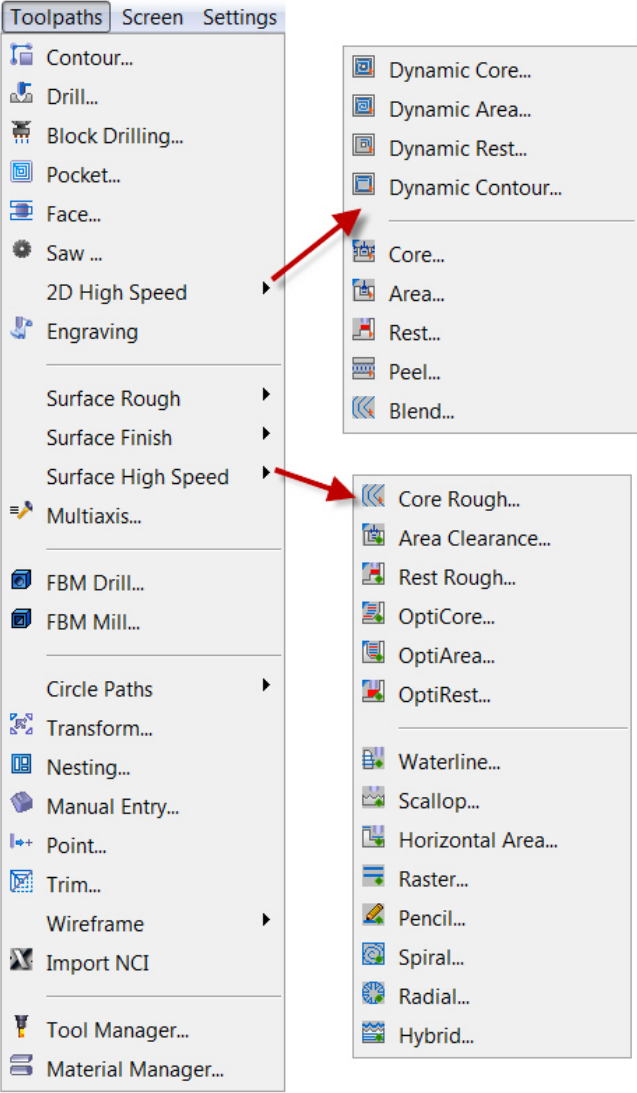
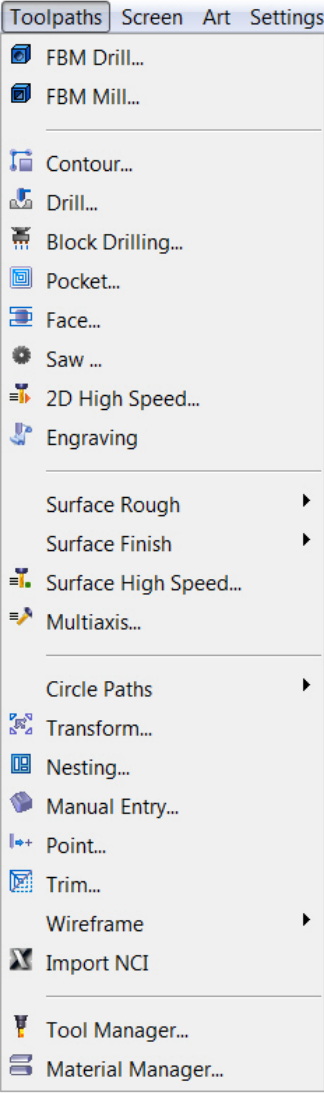


Mastercam X6 Mill



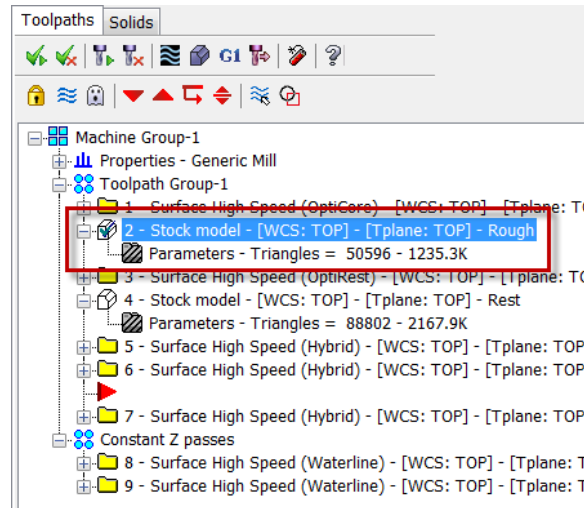
Mastercam X5 Router

Mastercam X6 Router



Stock Model

In Mastercam X6, the stock model for Mill 2-axis through 5-axis toolpaths is now a separate entity in the Toolpath Manager with more power and flexibility than ever before.



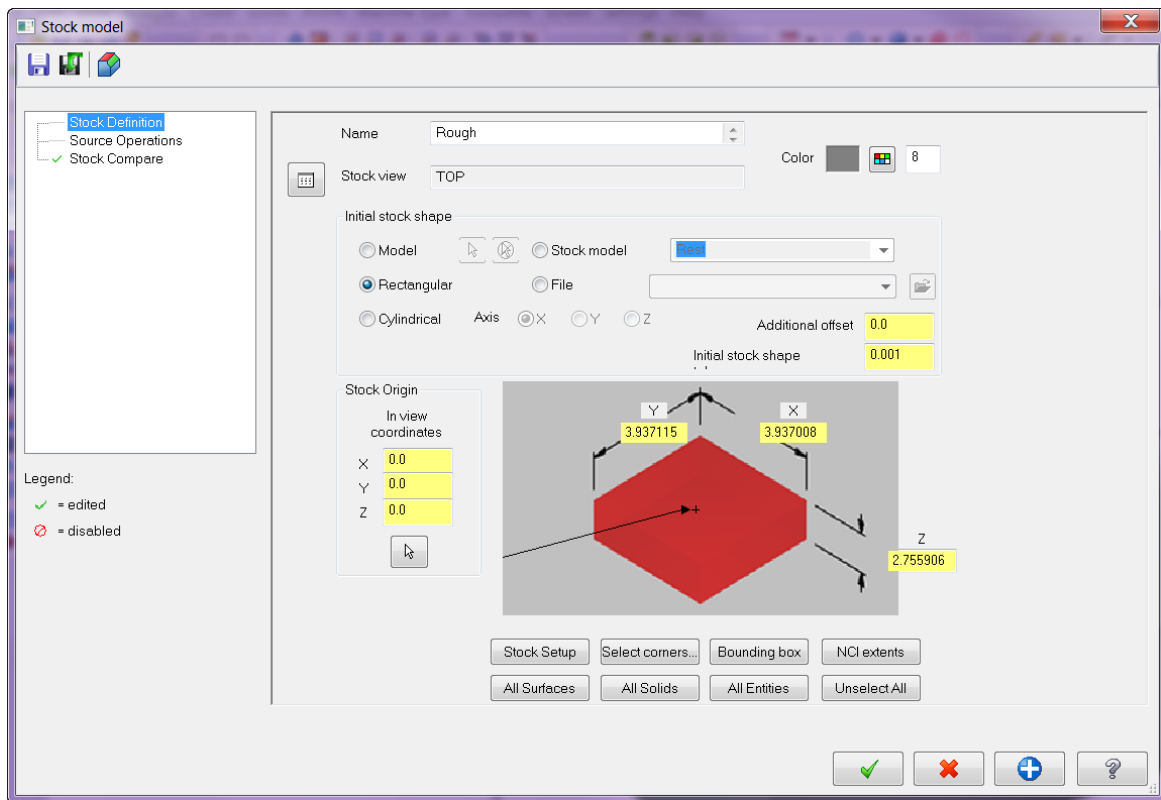
You can generate an initial stock model by choosing **Toolpaths, Stock model** from the Mastercam menu. The stock model is fully associative and has parameters that can be edited at any time similar to toolpath or solid operations. The stock model operation is marked dirty and requires a regen if you make changes to the stock model parameters or any toolpath the stock model is referencing.

Once created, you can use stock model operations to:

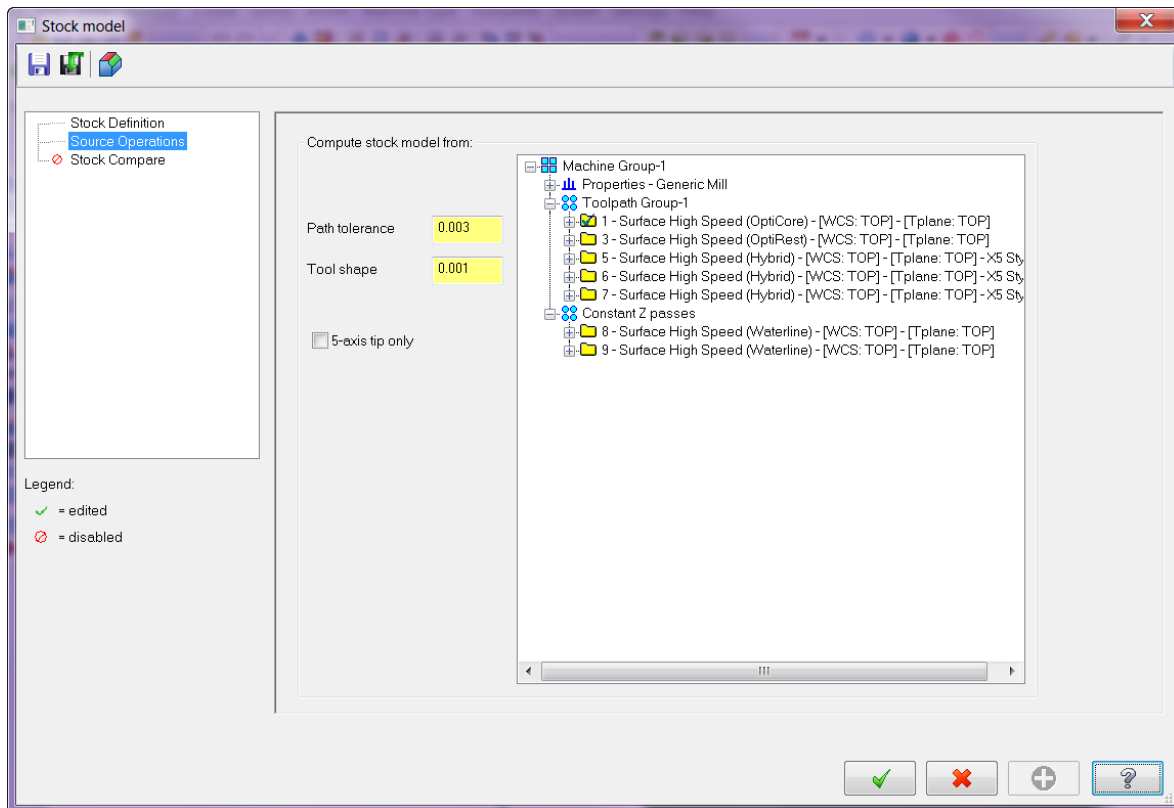
- Verify stock
- Define stock for rest machining
- Visualize stock
- Generate stock model compares

Note: If you have activated multi-threaded processing for your configuration, Mastercam generates stock model operations as multi-threaded processes.

In the Stock Model dialog box, the Stock Definition page lets you create a name for the model, select a color, and set the basic shape parameters. Solids, mesh entities, surfaces (even single surfaces), STL files, and other stock models can all provide the basis for the new stock model. You can also pull in any values you entered in the Stock Setup page of the Machine Group Properties dialog box.



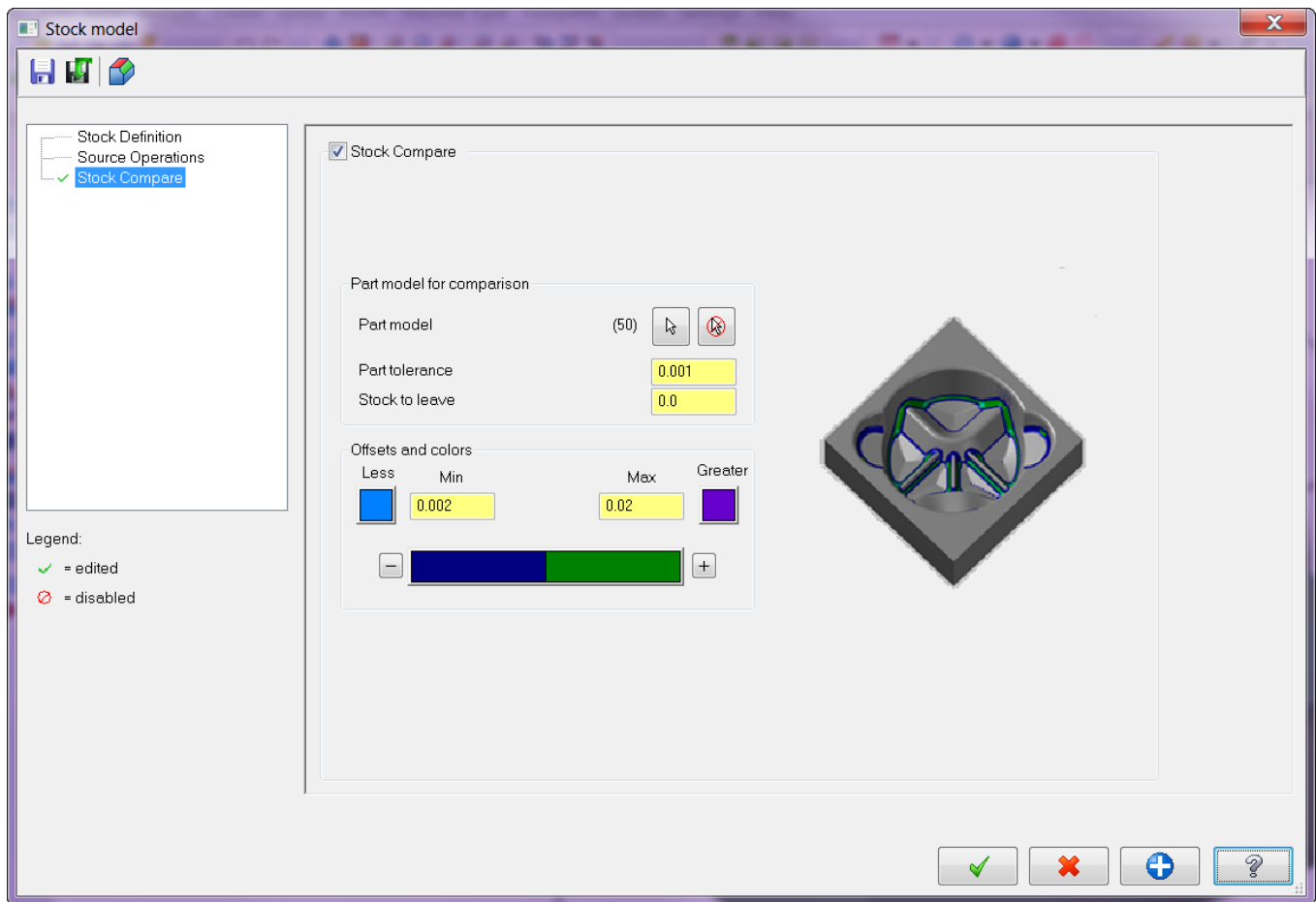
The Source Operations page is where you select the source operations to process against the defined stock in the stock model operation. Use **[Ctrl+click]** and **[Shift+click]** methods to select any combination of operations in the tree, including those from different machine groups.



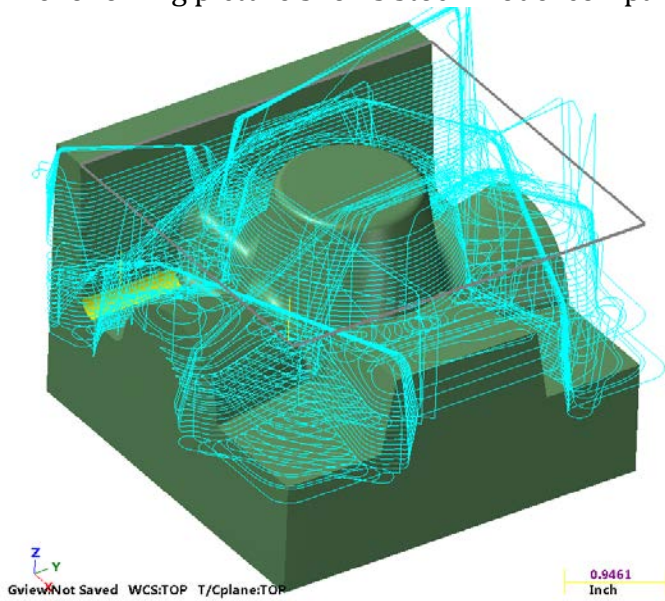
In addition, you can use the stock model to display leftover material. Using the Stock Compare page, follow these general guidelines:

- Select the **Stock Compare** checkbox.
- Use the **Select** button to return to the graphics window and select the part model entities to use in the comparison against the stock model.
- Define the tolerance for the stock model compare.
- Enter a value for stock to leave on the part model.
- Set the minimum/maximum offset values and colors for the compare results.

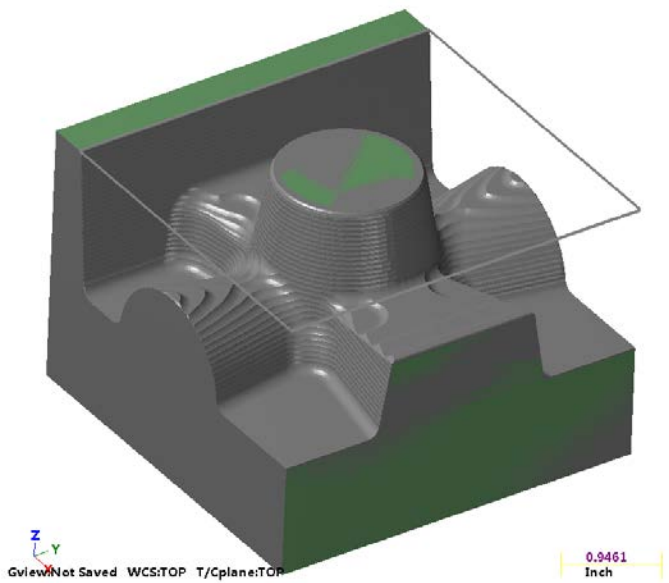
For more details on using this function, please click the Mastercam Help button at the bottom of the dialog box.



The following picture shows stock model comparison results:



Toolpath operation



Stock model result from operation

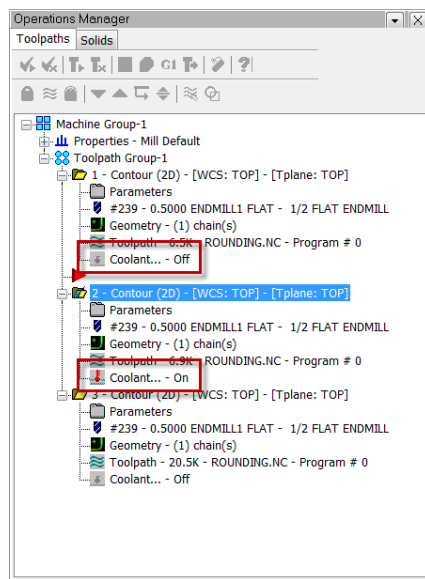
Bull-Nose Tool Support in Dynamic and Optimized Roughing Toolpaths

Toolpath motion in the following Mill and Router toolpaths is now optimized for bull-nose tools as well as flat endmills. If you select a bull-nose tool, the toolpath strategy alters the tool's trajectory to prevent any remaining material cusps.

- 2D HST: Dynamic Core, Dynamic Area, Dynamic Rest
- 2D HST: OptiCore, OptiArea

Coolant Icons in Toolpath Manager

These new icons let you quickly see whether coolant is on or off for each toolpath. These icons are only used by Mill and Lathe toolpaths. Clicking on the icons takes you to the toolpath parameters so you can make changes if necessary.



Mill/Router Tool Improvements

Improved Tool Preview Zoom Functionality

A new **Zoom** button on the Define Tool dialog box quickly focuses on the tool's cutting portion and lets you see details on very small shaded tools. Clicking the Zoom button again returns you to the Fit view. The new preview makes it much easier to see the cutting portion of the tool and reduces the chance of errors.

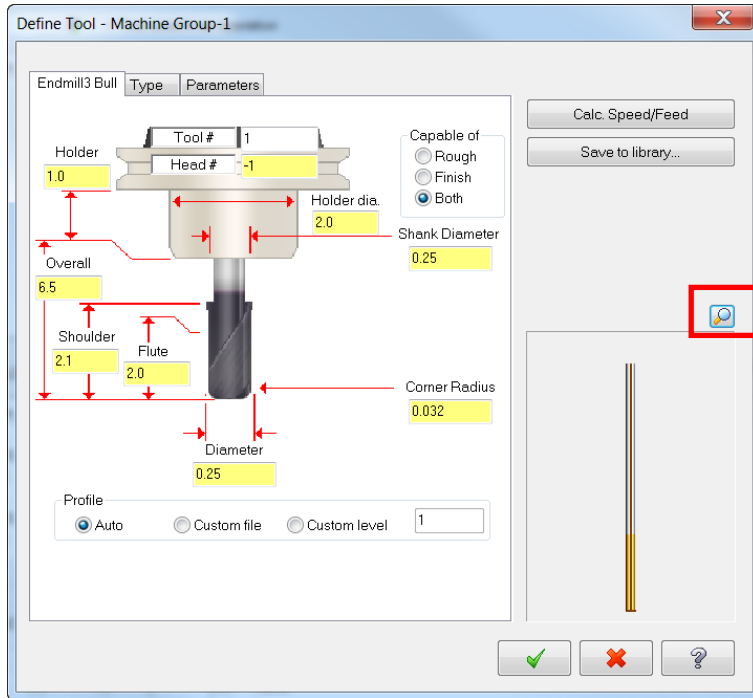


Figure 1 - The Define Tool dialog previewing a long tool with a small corner radius

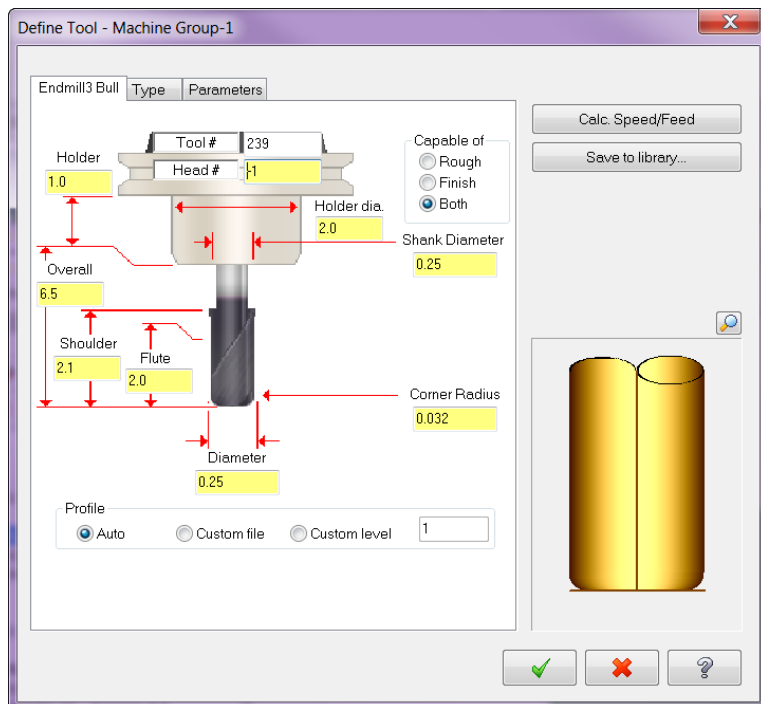
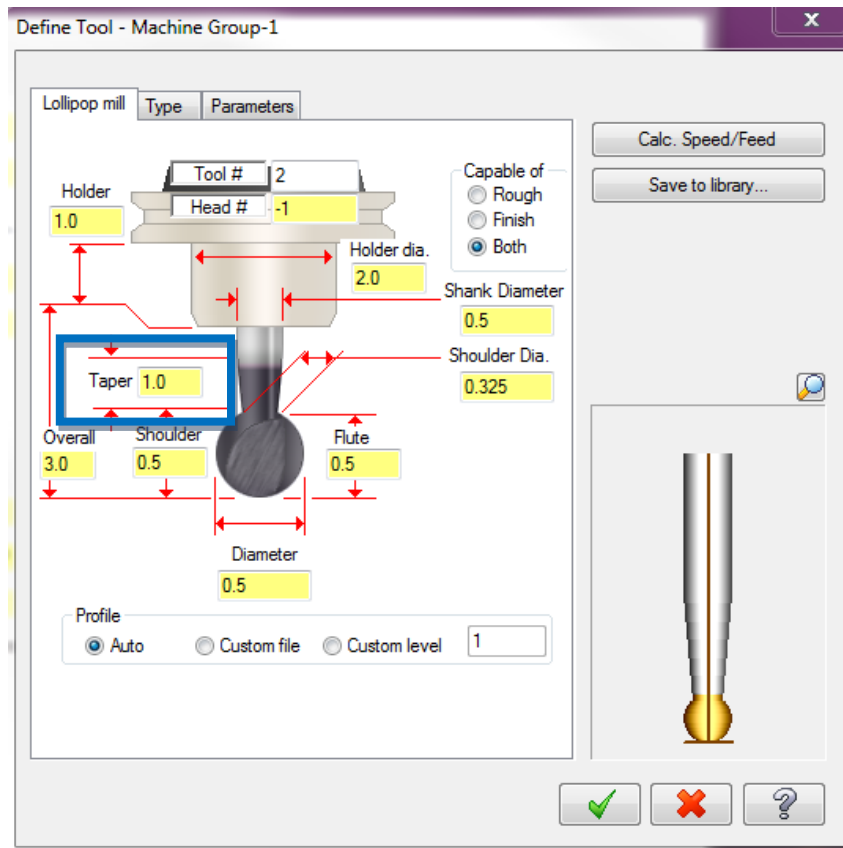


Figure 2- The Define Tool dialog previewing a long tool with a small corner radius in zoom mode

New Taper Section for Lollipop Tools

The lollipop tool shape is often used for undercutting operations. In order to make better use of the tool in these situations, it's common to have a tapered section on the shank of the tool. Rather than creating a custom tool or representing your tool as a short shanked tool with a tapered holder, you can now define the taper in the Define Tool dialog box.



Holder Support Added to TOOLDB

In Mastercam X6, you can now read tool holders from the TOOLDB (tool database) file. The HOLDERS file format is still supported for backwards compatibility. You can use the **Save to library** function on the Holder toolpath parameter page to convert your HOLDERS file into a TOOLDB file. You can also save individual holders to an existing TOOLDB file.

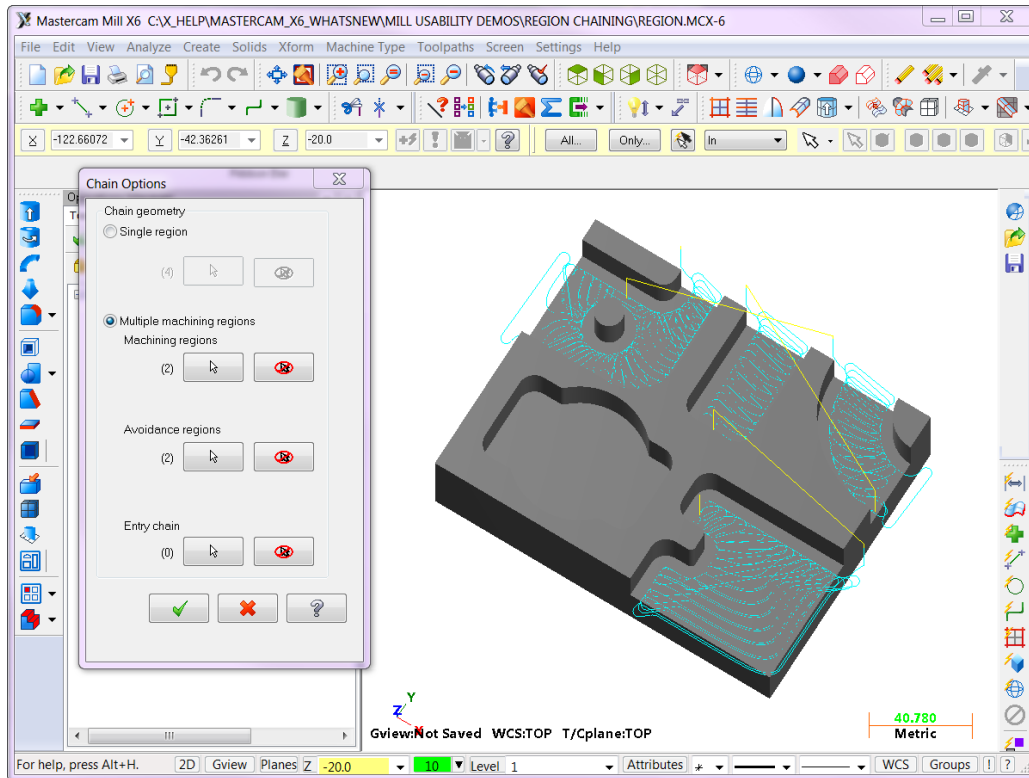
Besides saving individual holders to a library, one of the most important benefits is that as new TOOLDB files become populated with holder data, you can access to them through a new library without waiting for a new software release.

Note: Saving individual holder components to an existing library is only supported when using the new TOOLDB file.

Mill/Router 2D Enhancements

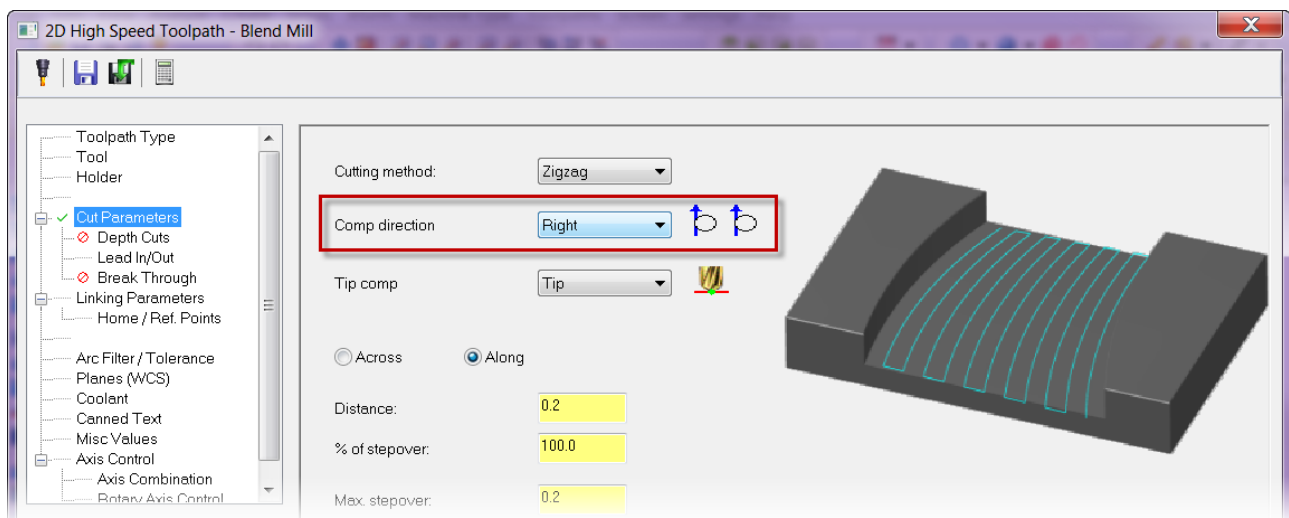
New 2D HST Region Chaining

The new region chaining options in Mastercam X6 give you more control over areas to machine and areas to avoid in your 2D HST toolpaths. After you select your initial chains, the Chain Options dialog box lets you adjust your machining areas.



2D HST Blend Mill Compensation

Cutter compensation options are now available in 2D HST Blend Mill toolpaths. The options include left, right, inside, and outside. Tip compensation and stock to leave on walls are also supported.

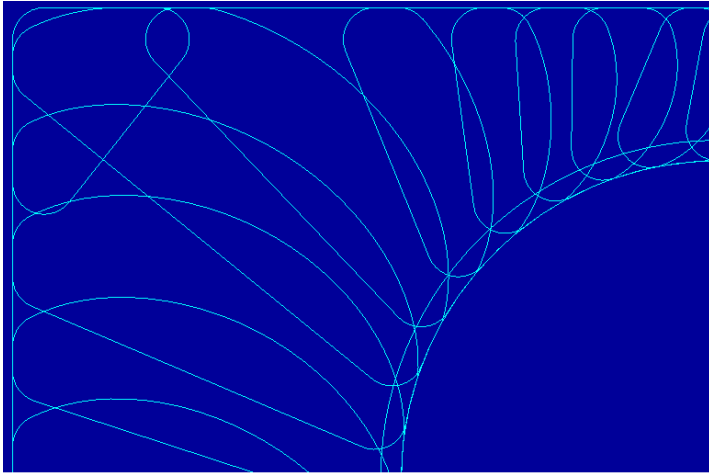


2D HST Dynamic Toolpath Improvements

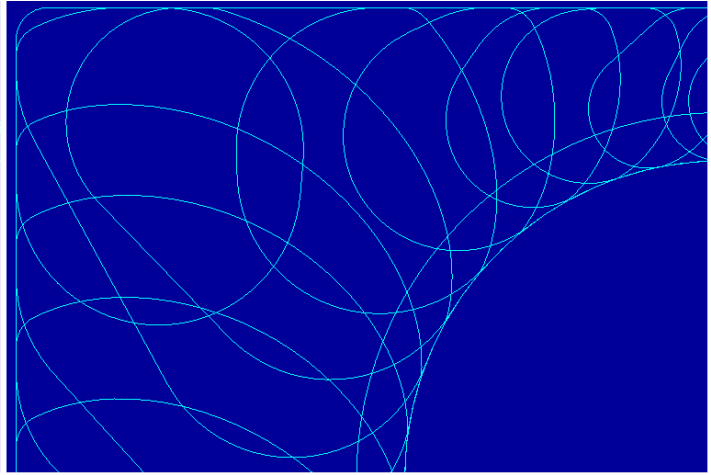
The 2D HST Dynamic toolpaths (Core, Area, Rest, and Contour) include several enhancements in Mastercam X6:

Variable Toolpath Radius

The 2D HST Dynamic toolpaths currently use the toolpath radius value as a hard value for both arc on/off motion and sharp corner smoothing. At times, the tool cannot fit into narrow channels or between islands due to the toolpath radius size. Instead of using a smaller toolpath radius to fit into these areas, which limits the efficiency of the arc on/off motion, these toolpaths now automatically adjust the toolpath radius in these areas. This allows the tool to fit into tight areas and mill deeper in sharp corners. You can enter a minimum toolpath radius on the Cut Parameters page.



Without variable radius

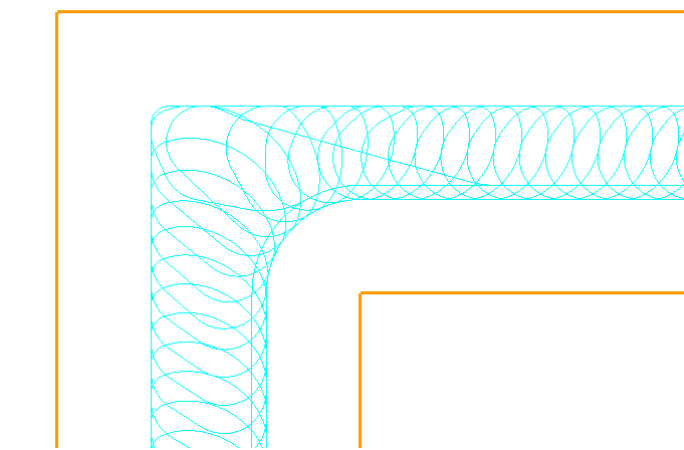


With variable radius

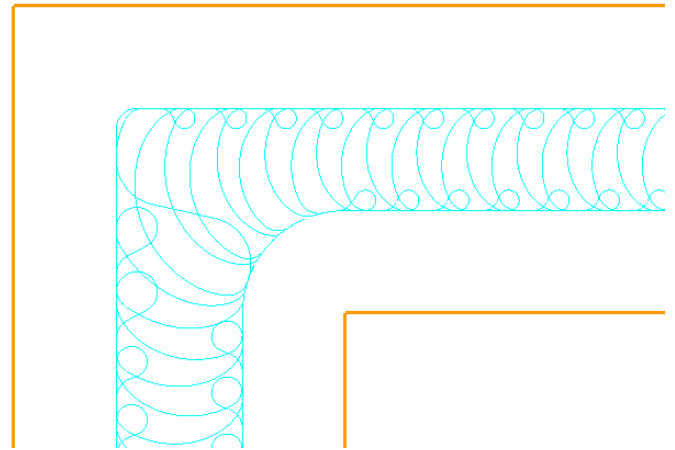
Zigzag Motion Support

With this new feature, you can now use zigzag toolpath motion for open passes to optimize your dynamic toolpaths. The toolpaths use the toolpath radius to smoothly exit and enter material during direction changes. When machining closed contours, the toolpath defaults to climb milling.

Note: This change does not apply to Dynamic Contour toolpaths.



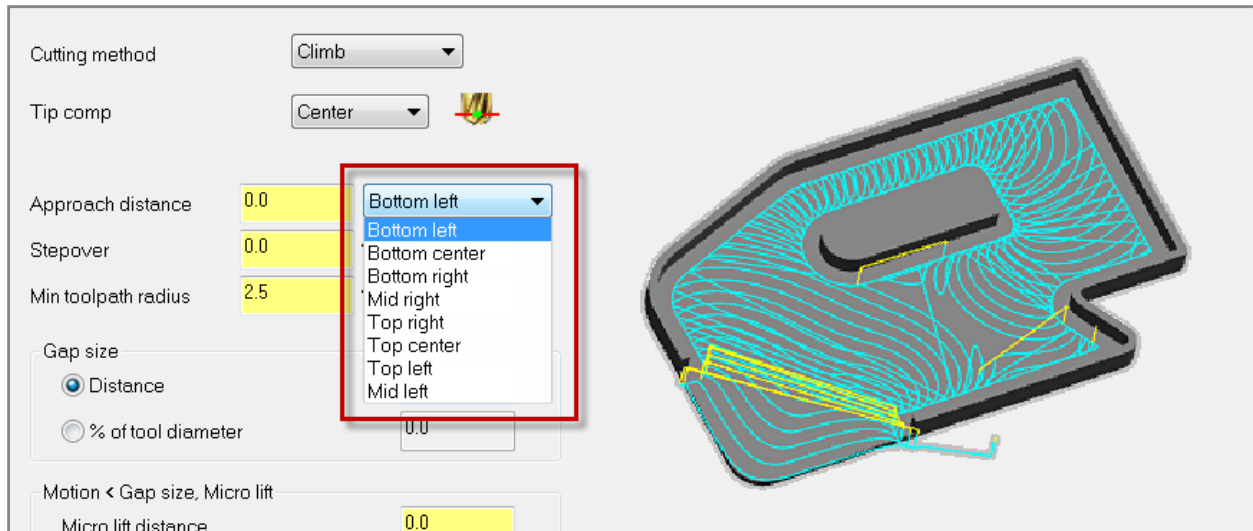
Climb milling



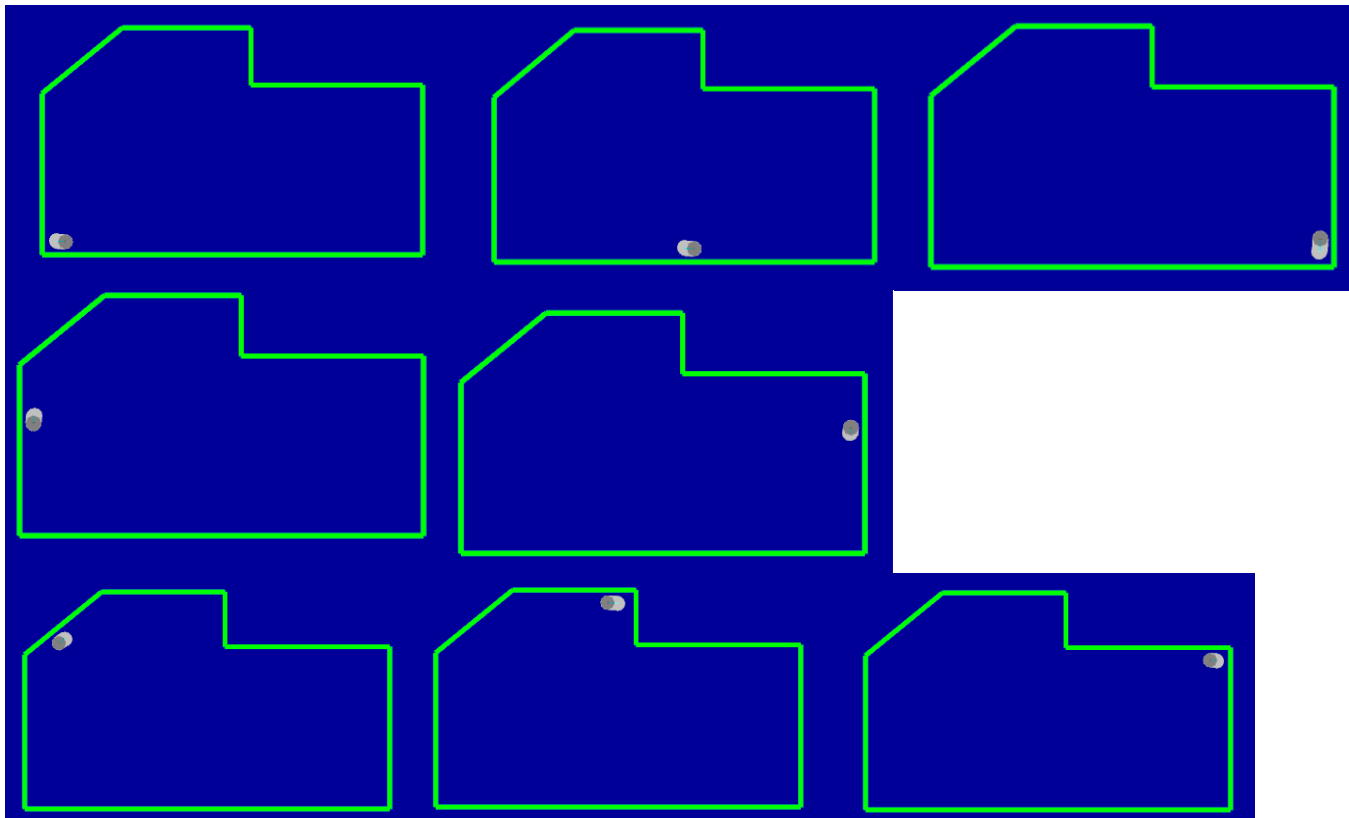
Zigzag milling

Outside Approach Control

Dynamic Facing, Core, and Contour toolpaths now include the ability to specify which side of the block the tool approaches from. You can use this functionality to avoid over-travel conditions when large parts might be taking up most of the travel in one direction.

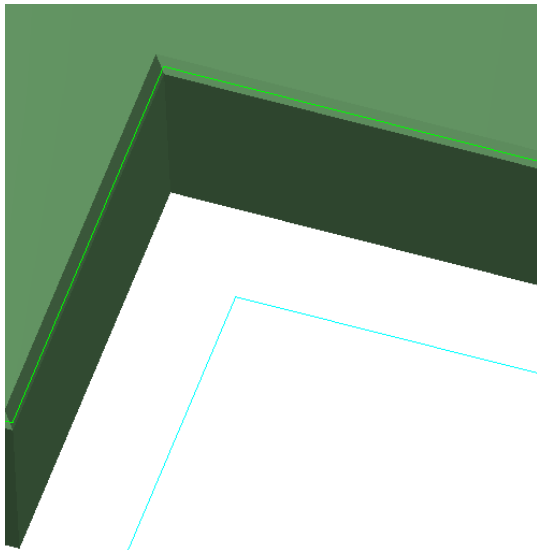
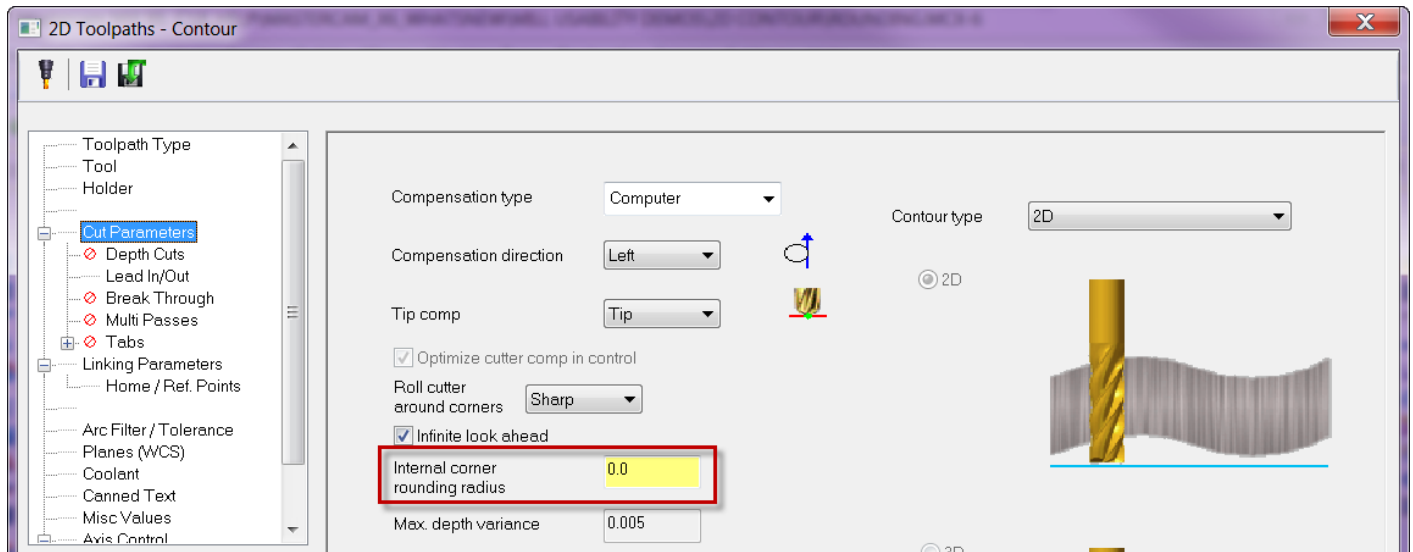


The following pictures illustrate the approach options:

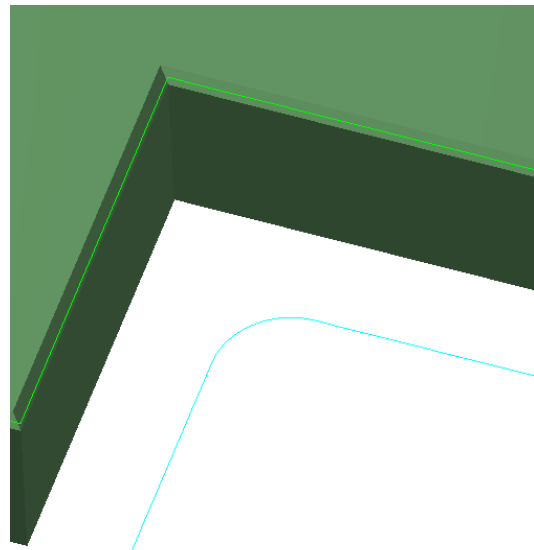


2D Contour Smoothing

In Mastercam X6, the 2D Contour toolpath now creates smooth motion in sharp corners. Removing sharp corners reduces tool wear and makes your tool motion more efficient. You can enter an Internal corner rounding radius on the Cut Parameters page. Corner smoothing also supports multi passes.



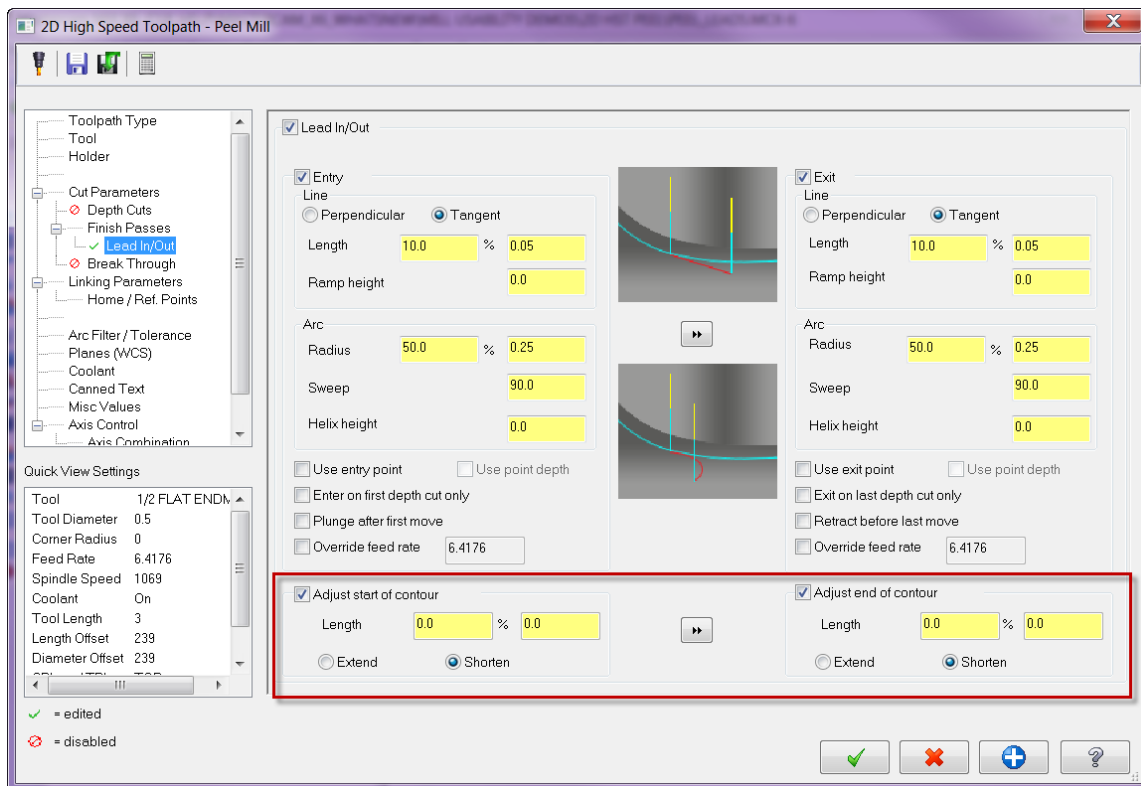
Without corner rounding



With corner rounding

Adjust Start/End for Peel Mill and Dynamic Contour

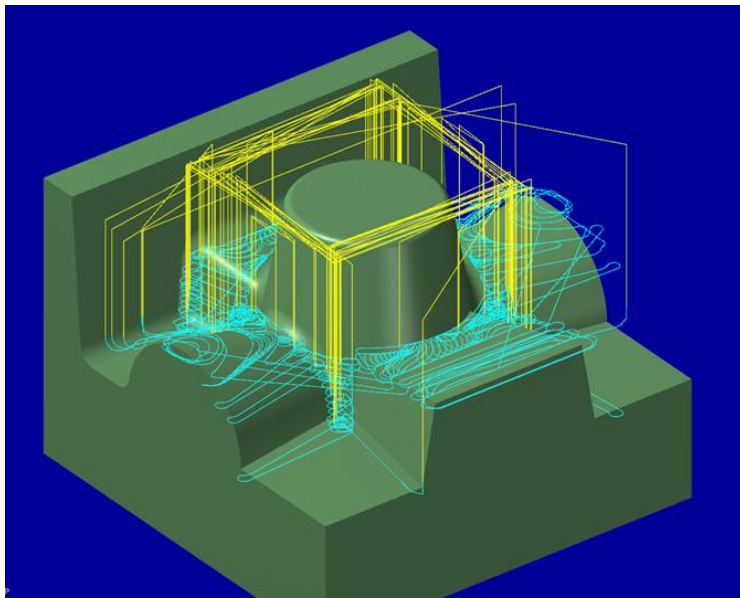
Peel Mill and Dynamic Contour toolpaths support lead in/out for their finish passes. This additional functionality allows the finish pass lead in/out to be extended or shortened without having to create a separate operation or having to modify geometry. This functionality is in addition to the entry/exit extensions already supported in the peel mill roughing motion defined on the Cut Parameters page.



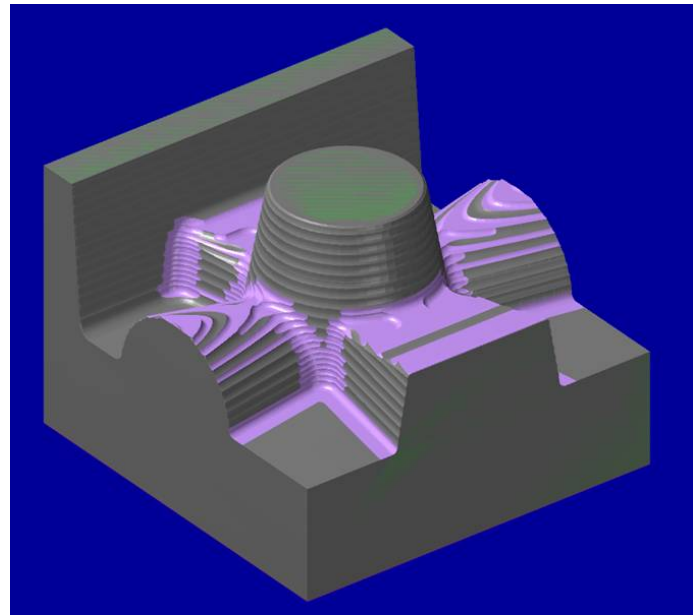
Mill/Router 3D Enhancements

New OptiRest Toolpath

Mastercam's latest 3D HST toolpath is OptiRest, which performs rest milling operations to remove remaining material with optimized roughing motion. It is the first Mastercam toolpath to take advantage of the new stock model functionality for defining stock removal areas. In addition, containment boundaries are required for this toolpath.



Toolpath motion



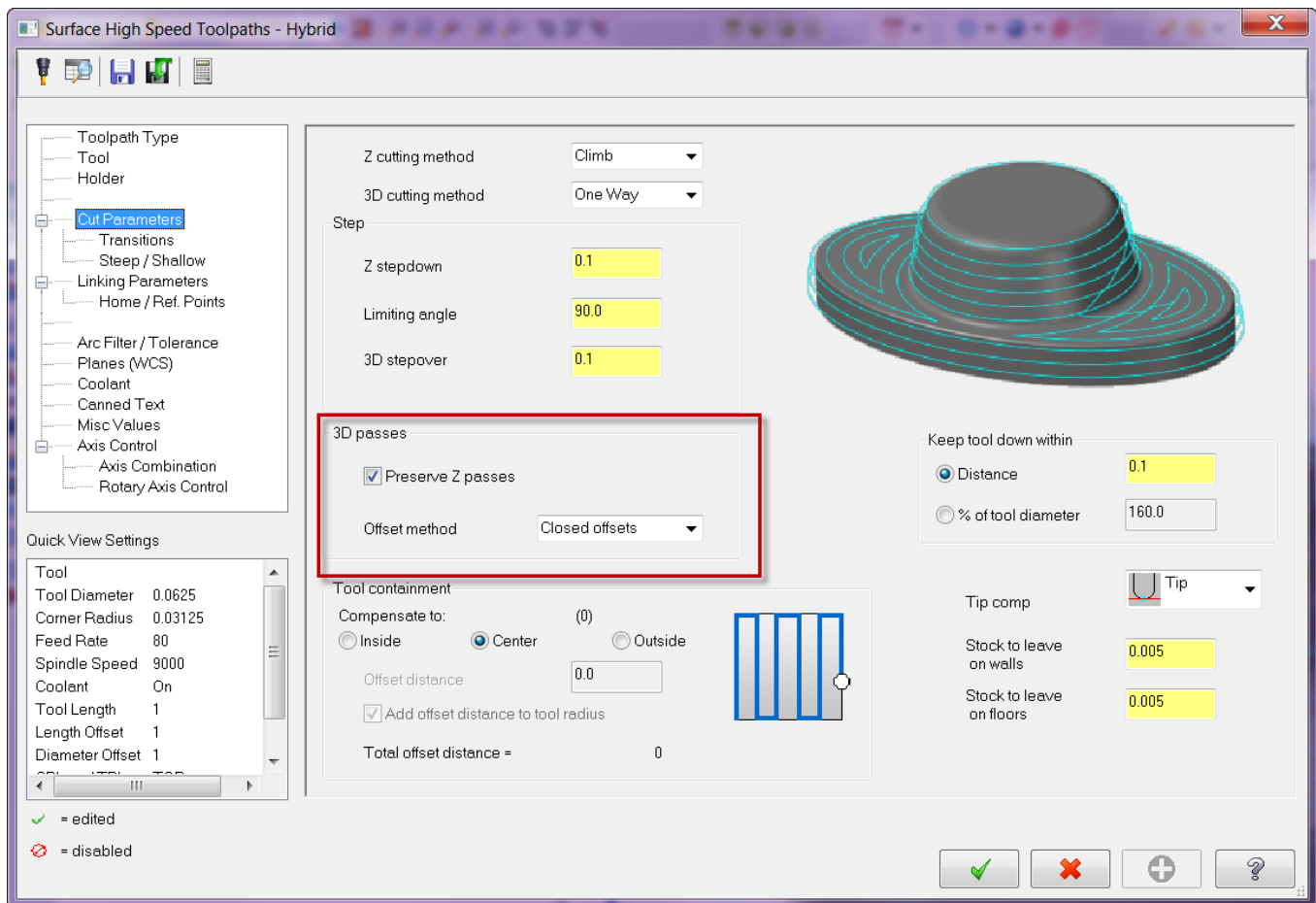
Gray = stock model, purple = OptiRest machining area

3D HST Hybrid Finish Filler Passes

3D HST Hybrid toolpaths now maintain all of the Z level cuts in the steep areas of your part and fills in the shallow areas with scallop motion. It combines the best of both worlds with waterline (constant Z) motion and scallop where needed. In X5, shallow areas would have resulted in 3D collapse (essentially scallop) motion for all Z cuts at that level. Now you can have both waterline and scallop motion at the same Z level so the entire part is cut.

You can control this motion on the Cut Parameters page for Hybrid toolpaths. Use the **3D passes** section to control how the toolpath handles shallow areas. Select **Preserve Z passes** to keep all waterline motion in the toolpath. Then select a method for handling the boundaries between steep and shallow areas from the drop-down.

- **Closed offsets:** The boundaries are treated as closed chains when calculating the 3D collapse motion.
- **Upper to lower:** The boundaries are treated as closed or open chains that collapse from the highest Z value to the lowest.
- **Lower to upper:** The boundaries are treated as closed or open chains that collapse from the lowest Z value to the highest.



OptiRough Toolpath Split into OptiCore and OptiArea

The current OptiRough toolpath uses a checkbox to control whether you use an area pocketing approach or a core from outside approach. In X6, OptiRough is split into two dedicated toolpaths for areas and cores – OptiArea and OptiCore. Both toolpaths can be found on the Surface High Speed Toolpaths menu.

Multiaxis Enhancements

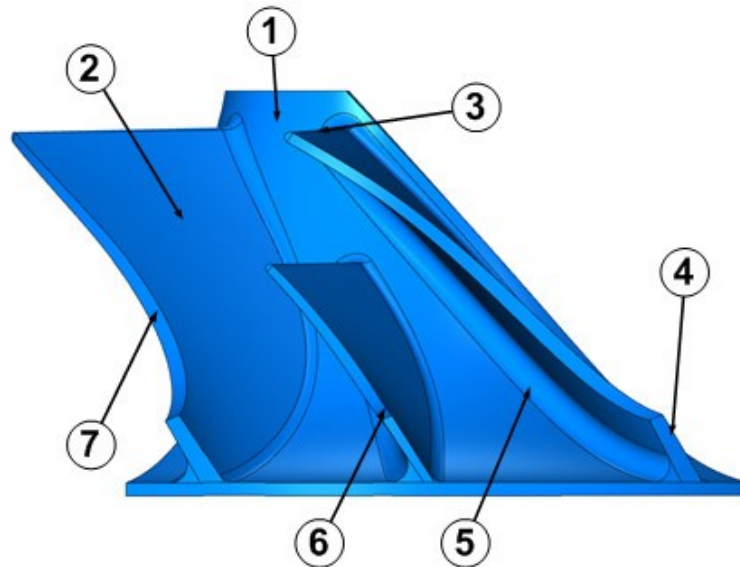
Blade Expert

Multi-bladed parts are used in many industries, including marine, aeronautical, energy, and automotive. Machining these complex shapes usually requires special multiaxis equipment, but Mastercam's new Blade Expert is specifically designed to generate the necessary toolpaths for many of the different multi-blade configurations.

Blade Expert is designed to generate the toolpaths for the following segments of the multi-blade workflow:

- Roughing between the blades
- Semi-finishing the blades
- Finishing the blades
- Finishing the hub

Multi-bladed parts typically include the following key areas:

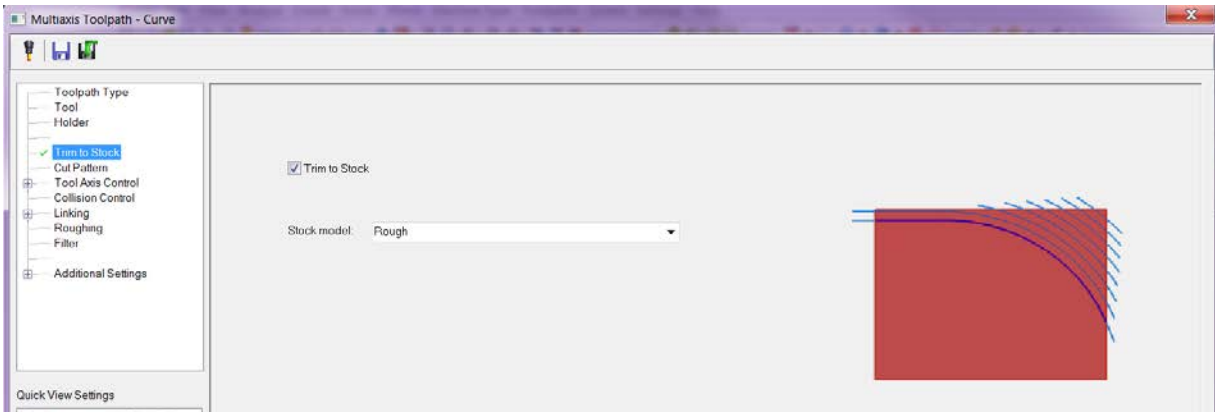


1. **Hub surface** - revolved floor surface that the blades sit on.
2. **Blade surface** - usually free-form surfaces with a double curved shape.
3. **Blade leading edge** - the suction side for the transported medium. For impellers and blisks, it is usually rounded. The leading edge is usually closest to the center of the hub.
4. **Blade trailing edge** - the exhaust side. For an impeller, this side is not rounded. The trailing edge is usually closest to the circumference of the hub.
5. **Fillets** - part of the blade. They guarantee a rounded connection between the blade and the hub. The fillets can have a constant radius as well as a dynamic radius.
6. **Splitter** - found on impeller geometries. It is a shorter blade, similar to the main blades, and located between the two main blades. The leading edge can be horizontal (as in the picture above) or tilted.
7. **Shroud surface** - the top surface of the blades and the splitter. It is usually an overturned surface from the stock, but can also be a free- form surface.

Note: Blade Expert is a Mastercam add-on that must be purchased separately. You must own Mill Level 3 or Router Pro to purchase the add-on, but you do not need to own the Multiaxis add-on.

5-axis Stock Model Toolpath Trimming

For any of the multiaxis toolpaths in the Classic toolpath family, you can trim your toolpath to the new stock model. Trimming to stock reduces the amount of time the tool is not in contact with the material.

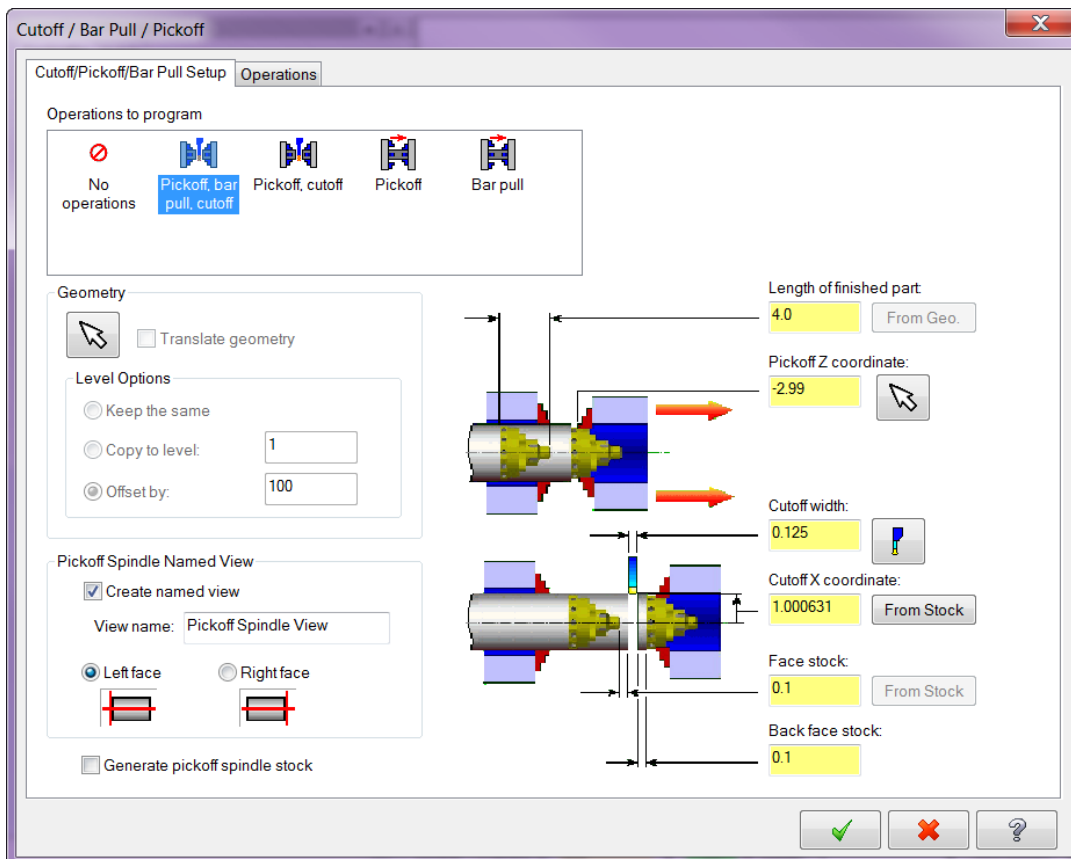


Lathe Enhancements

Improved Cutoff/Pickoff Operations

Mastercam X6 Lathe includes improvements to the cutoff and pickoff operations. Choose **Toolpaths, Misc Ops, Pickoff/Pull/Cutoff** to access this function.

Note: You must have an updated post processor to use this function. Please contact your Mastercam Reseller for more information.



Use this dialog box to translate geometry and create a part transfer operation from the main spindle to the sub spindle and/or a bar pull operation prior to a cutoff toolpath. You can choose to just create the geometry without creating any operation, or you can create the geometry and the following types of operations:

- Pickoff, Bar pull, Cutoff
- Pickoff, Cutoff
- Pickoff
- Bar pull

The following conditions must be met to use this function:

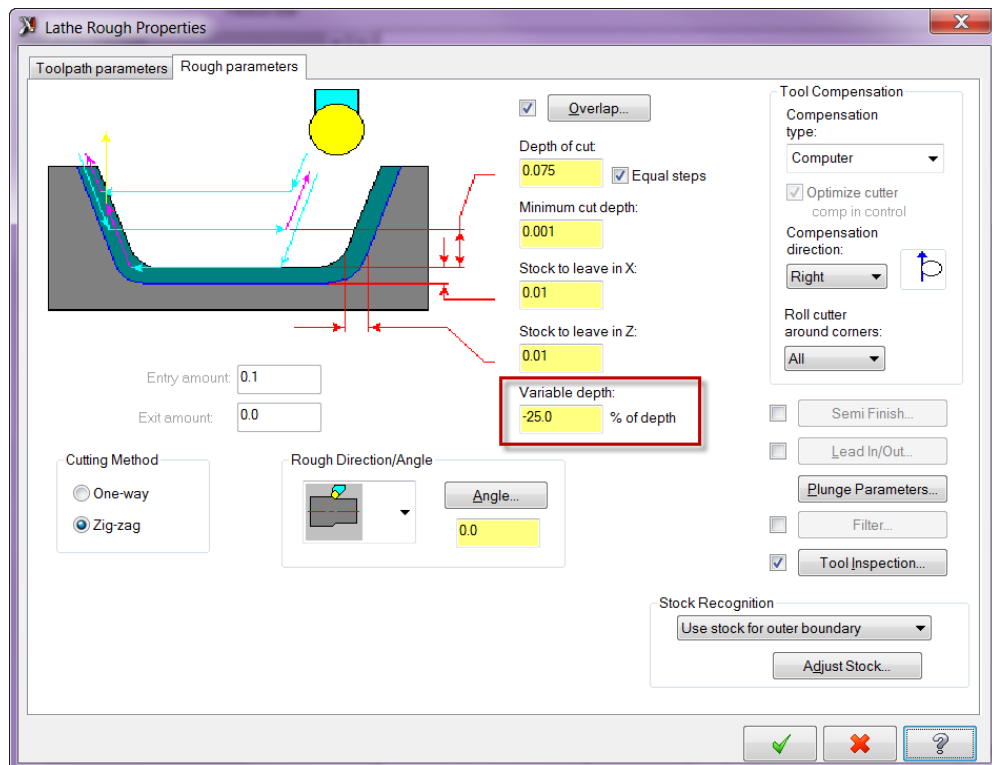
- The red Insert Arrow must be in a Lathe machine group.
- The machine group must have stock defined in a fixed spindle, or both spindles, and the chuck jaws defined for both spindles. If both spindles can move in Z, the spindle not containing stock will be assumed to be the pickoff spindle. If both spindles contain stock in this case, the right spindle is assumed to be the pickoff spindle.
- A cutoff operation must exist in the currently active operation library.

For more details on using this function, please click the Mastercam Help button at the bottom of the dialog box.

Variable Depth Roughing

This new roughing parameter allows you to vary the point that the surface contacts the tool insert to prevent notching and improve tool life. The depth can vary up to 25% of the cut depth. A positive value results in an upward cut and a negative value results in a downward cut.

- The passes alternate between angled and straight.
- If the cut length is less than three times the cut depth, a straight cut is made instead of an angled cut.
- In flat areas, a straight cut is made instead of an angled cut.
- One way and zigzag cuts are both supported, as well as ID, OD, Face, Back and Angled.



Finish Toolpaths - Up/Down Cutting Strategy

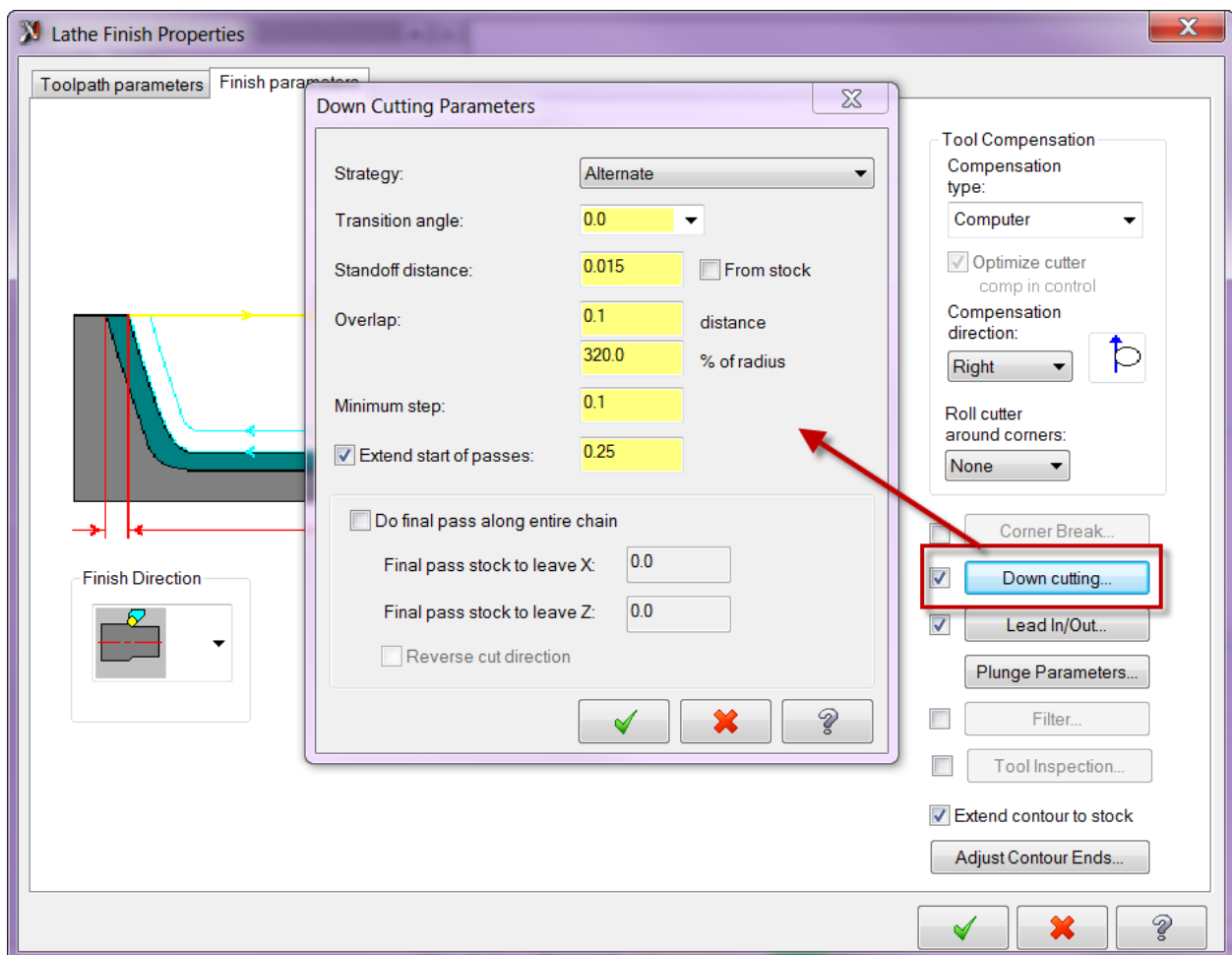
Lathe finish toolpaths are now optimized by detecting and finishing up (flats) and down (walls) cut regions of the profile using a single chain.

Note: This motion uses the following terms:

- **Walls:** other terms include steep, down cut and faces. Walls are cut in a vertical or downward direction.
- **Flats:** other terms include shallow, non-down cut and diameters. Flats are cut in a horizontal or upward direction.
- **Transition angle:** the deviation from the wall angle before the geometry is considered to switch from a Wall to a Face. A 90 degree transition angle implies that all regions of the geometry are considered Wall geometry. A 10 degree transition when cutting OD implies that geometry with an angle between 80 and 100 or 260 and 280 degrees is considered to be Wall geometry.

What is considered a wall or flat depends on the specified transition angle and finish direction. On the Down Cutting Parameters dialog box, you can choose from the following cutting strategies:

- **Walls:** Down cut Walls only using a down cut strategy.
- **Flats:** Machine Flat areas only in the chain direction.
- **Walls then Flats:** Machine all walls (wall strategy), followed by all flats (flat strategy).
- **Flats then Walls:** Machine all flats, followed by all walls.



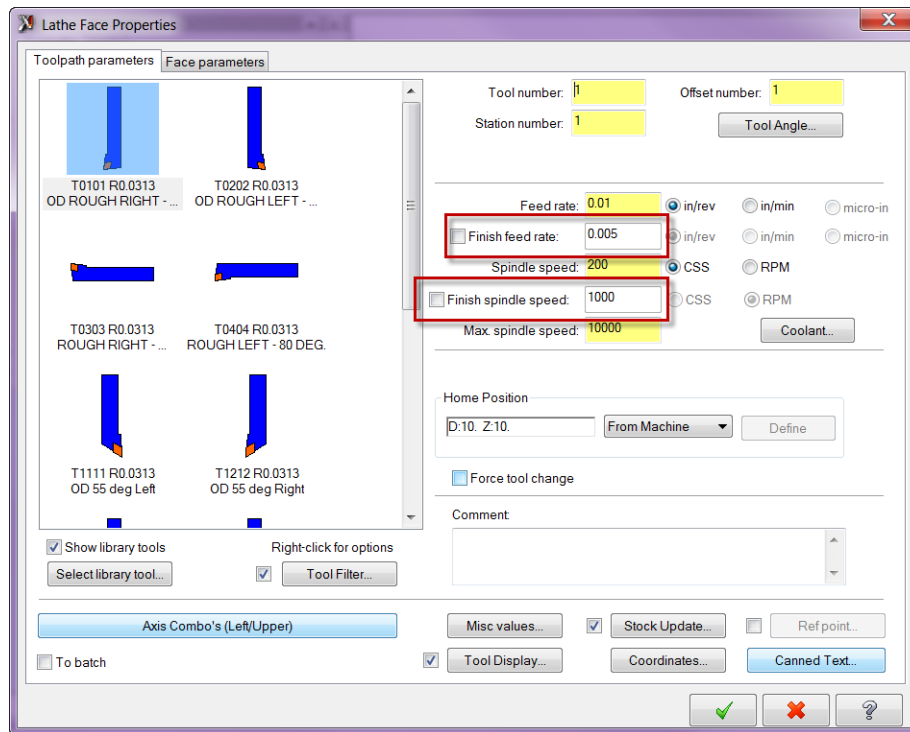
Canned Rough and Finish Improvements

Mastercam X6 Lathe includes several improvements to canned rough and finish toolpaths:

- Added clearance options to the canned rough operation parameters to extend the chain for tool clearance.
- Added parameters to shorten the start and end of the chain in the Lead in/out dialog. This is especially useful when cutting on the OD towards the chuck. The chain can be shortened to prevent the tool from hitting the chuck.
- Canned finish operations now use the cutter comp options from the canned rough operation. The cutter comp options from the canned rough operation will be displayed but disabled.
- Support for Type I and Type II canned cycles
- Added a clearance value for canned finish. The start point used in canned finish is shifted above the part by the clearance. Then at the end of canned finish, the tool moves up and away from the part on the move to the start point.

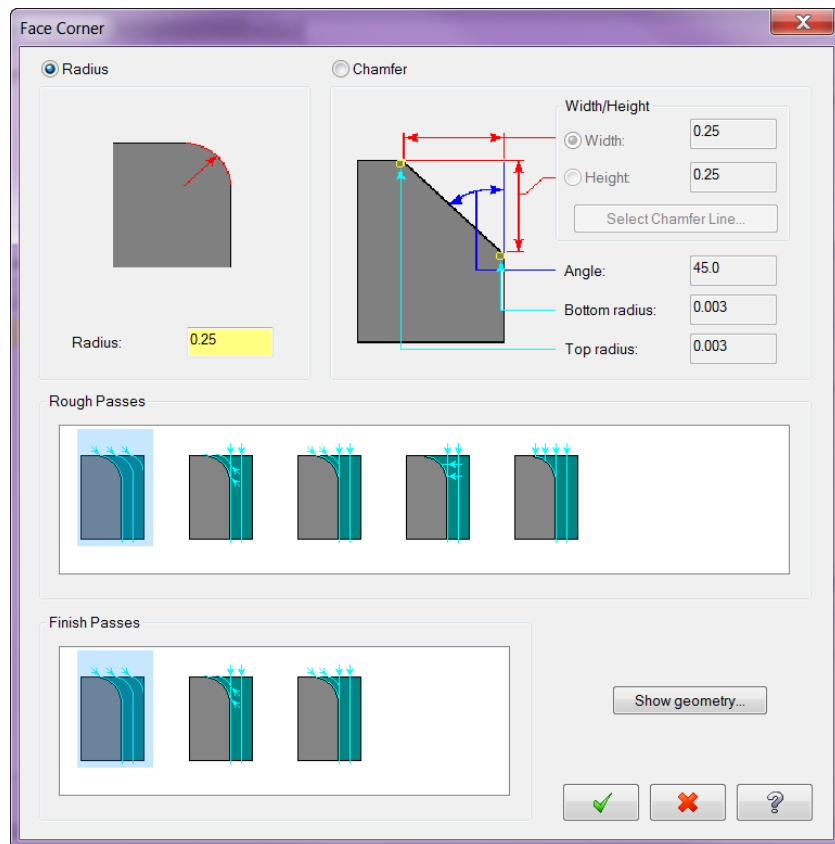
Facing and Canned Groove Toolpaths - Optional Finish Feed/Speed

Facing toolpaths in Mastercam X6 Lathe offer additional feeds and speeds specifically for finish passes. This functionality has also been added to Canned Groove toolpaths.



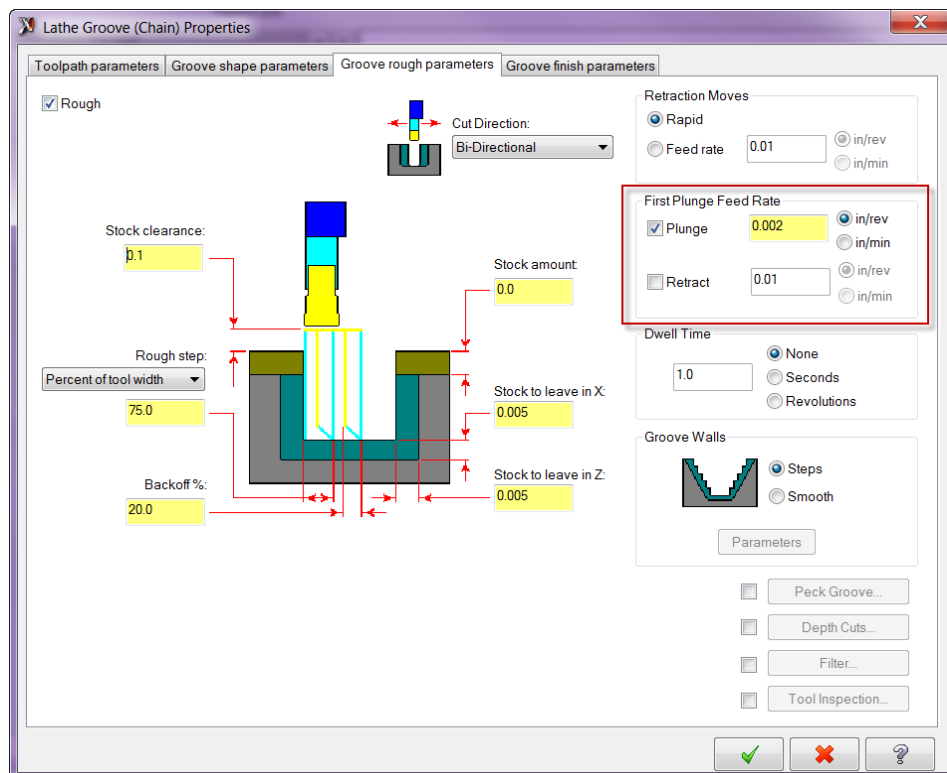
Facing Corner Break

The new **Corner** button on the Face parameters tab provides options for adding a chamfer or radius to the edge of a part. The break is only allowed on the “leading” corner of the face (the first corner encountered moving in the direction of the toolpath).

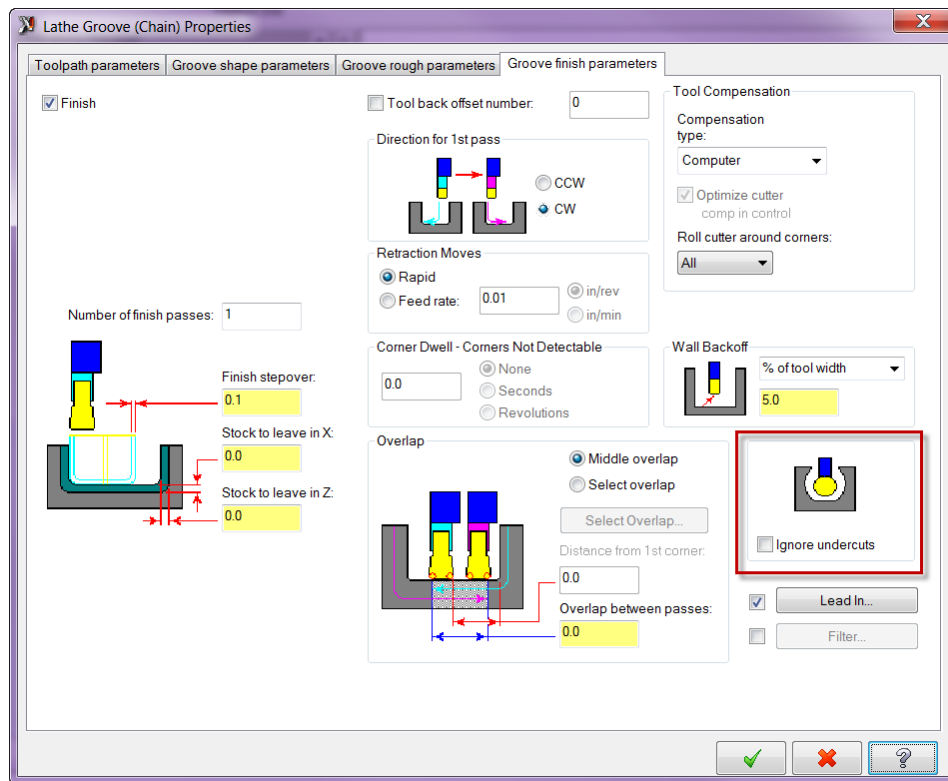


Grooving Toolpath Improvements

For groove roughing toolpaths, you now have the option to adjust the plunge and retract feed rates on the first plunge cut.



Groove finishing toolpaths support undercut removal and negative stock to leave.

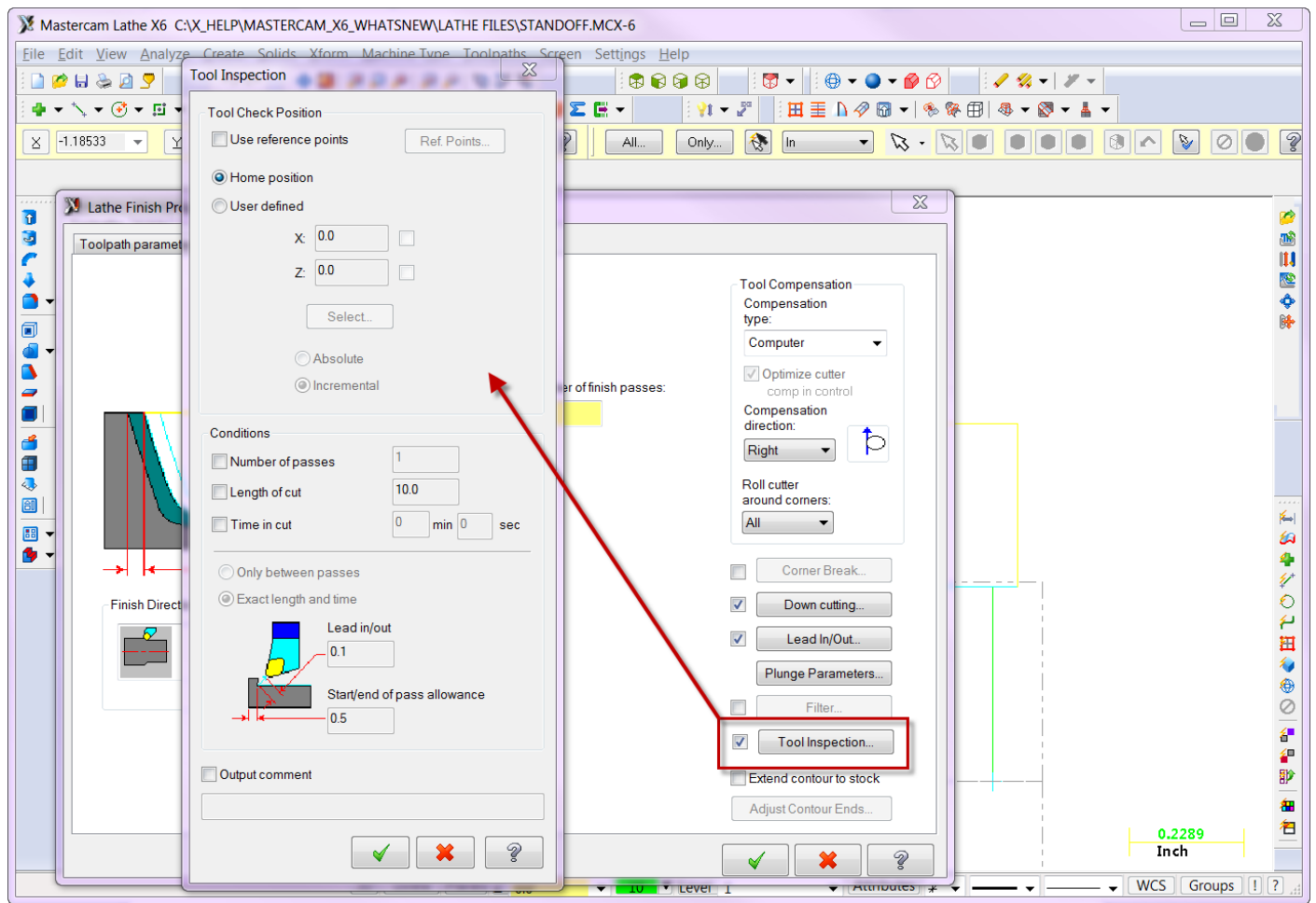


Finish Tool Inspection

Tool inspection is available in Finish toolpaths. This function had previously only been available in Rough and Groove toolpaths. You can establish points during operations where the machine can be programmed to a specific position and stop to allow you to perform tool inspections. You can set the conditions on which the tool inspections are performed, such as:

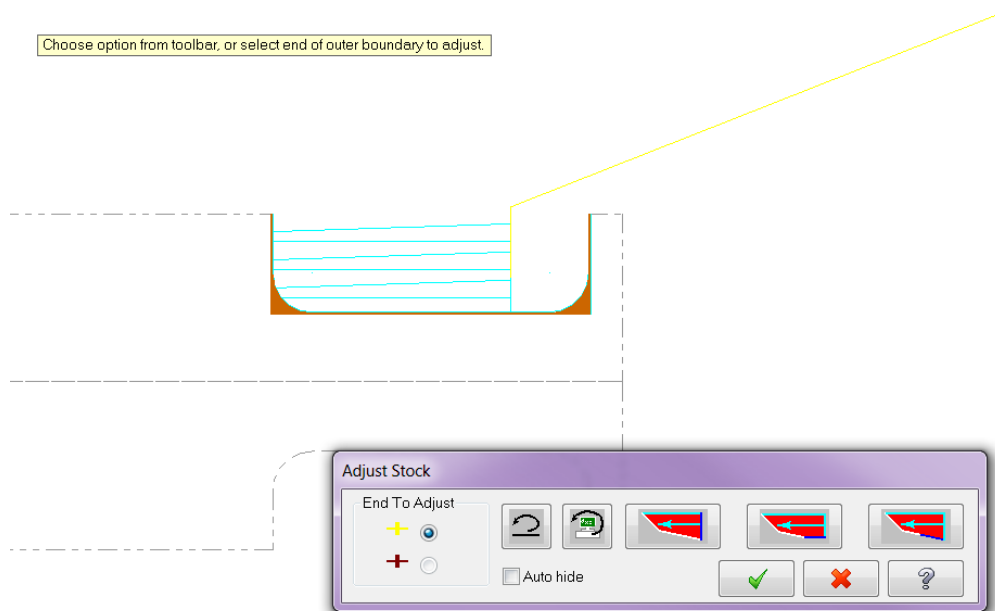
- After a number of passes
- After a specified cut length
- After a specified cut duration

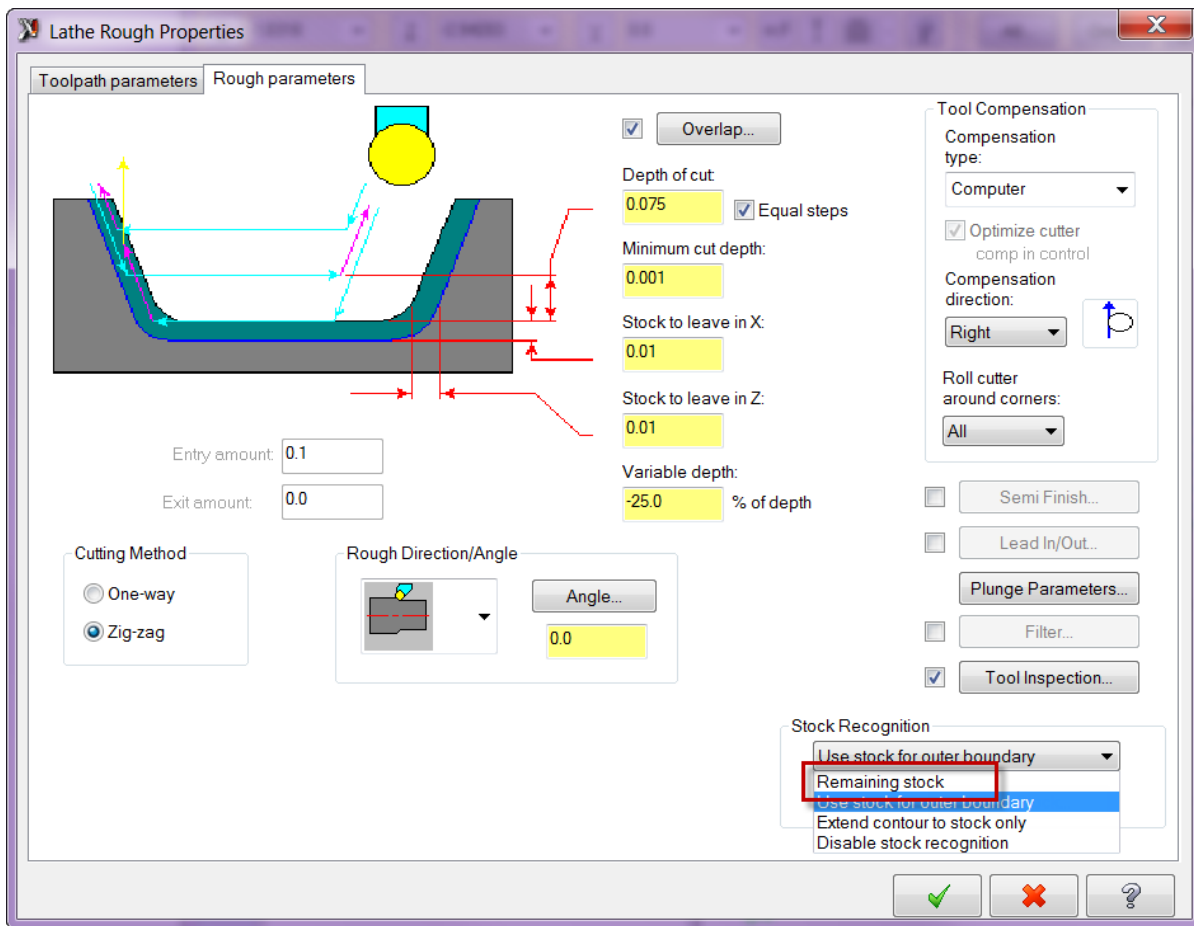
Additional options let you control whether a tool inspection can occur during a pass or near the start or end of a pass. You can also use reference points to get the tool in and out of the cut, and send the tool to a selected point or to the home position for the tool inspection position.



Rough Remaining Stock

This new option adjusts your roughing toolpaths to minimize air cuts by recognizing remaining stock. You can then click Adjust Stock to choose your machining area.





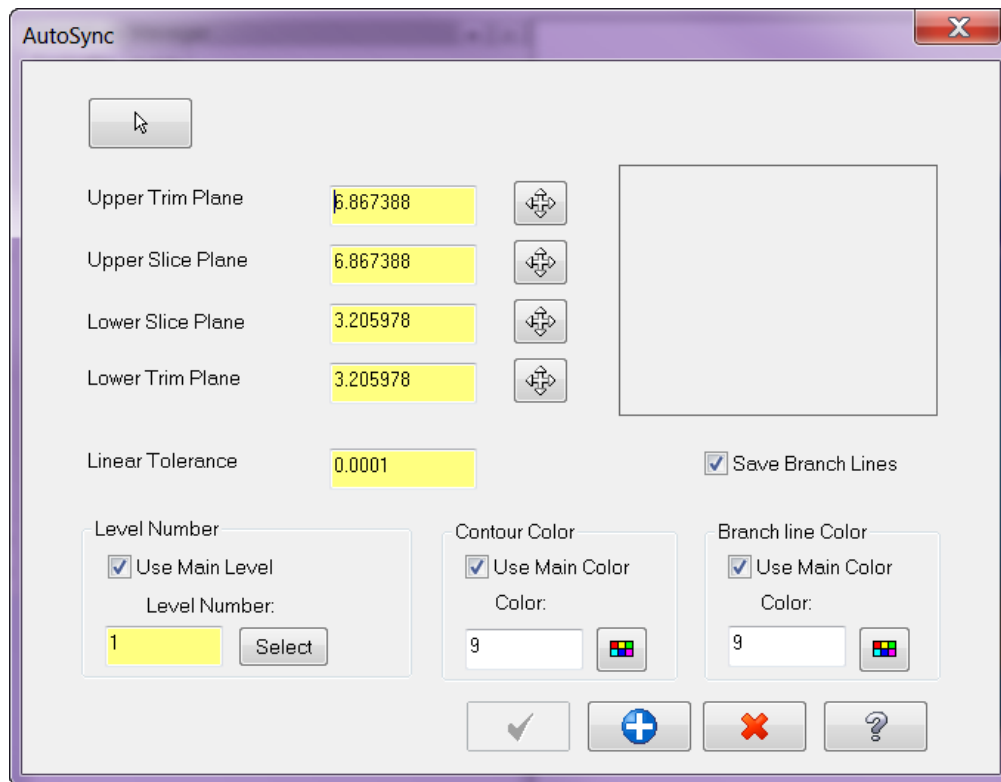
C-Axis Face Contour and Face Drill – Support for Plane Rotation

The C-Axis Face Contour and Face Drill toolpaths now support plane rotation when using Y-axis motion.

Wire Enhancements

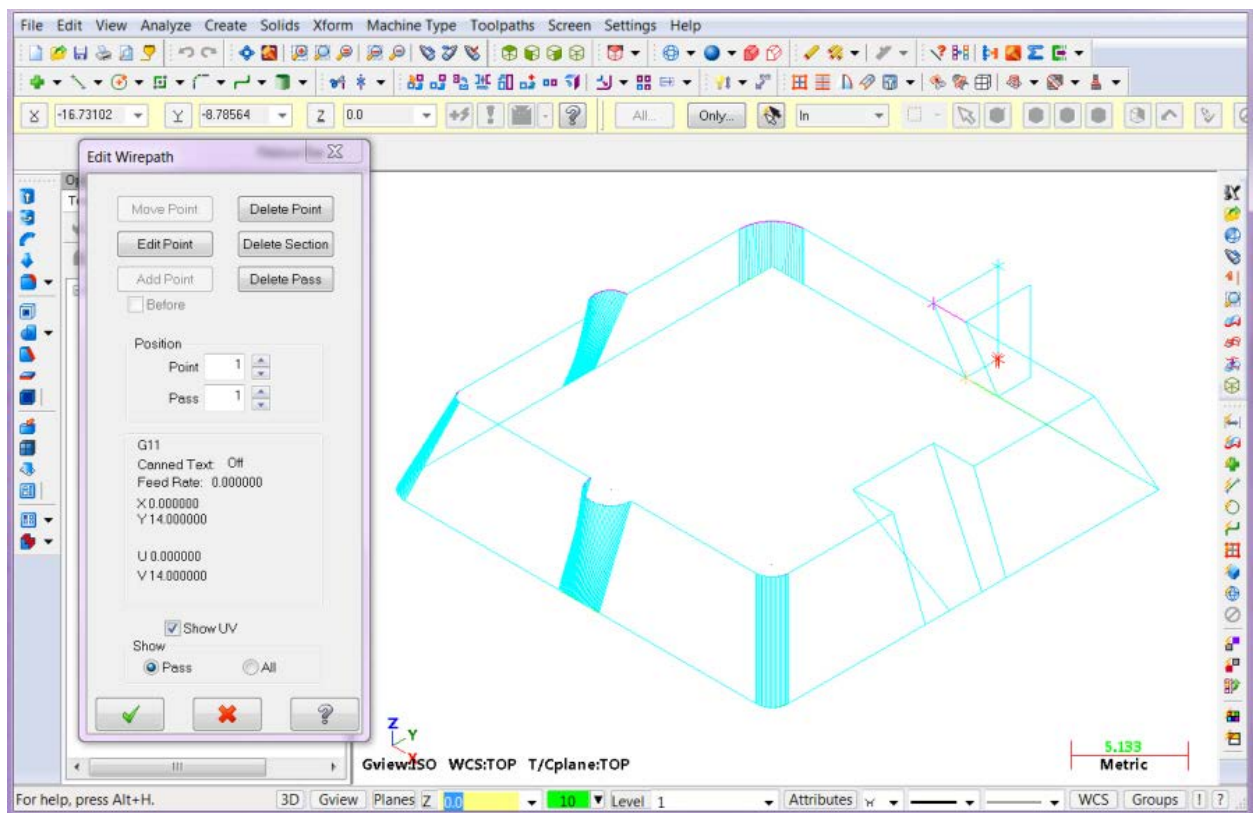
New AutoSync Rails

This function has been improved in Mastercam X6 Wire to create better synchronized wirepath chains from selected solids and surfaces. Choose **Create, Autosync Rails** to access the new functionality.



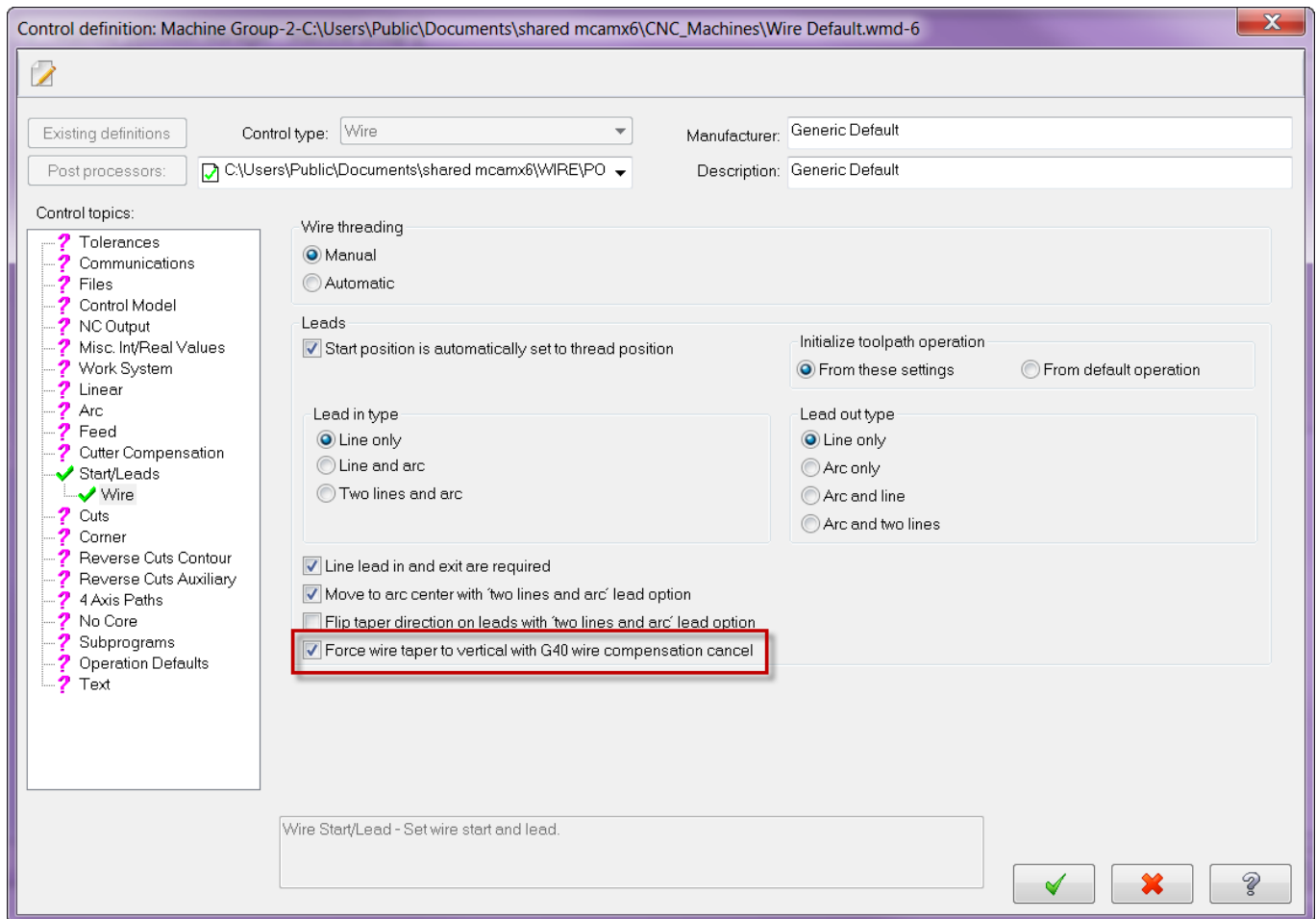
Wirepath Editor Improvements

The Mastercam Wirepath Editor now supports 4-axis Direct wirepaths.



Control Definition Option to Force Wire Taper to Vertical

Mastercam X4 introduced a **Reset Taper** option on the Leads page that allowed you to force the wire back to a vertical position when the **Max lead out** distance is used. In X6, a new option in the Control Definition allows for this Reset Taper mode to be always active. Some machines require the wire to move to a vertical position on a change-over from G41 to G42 compensation.



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



mc software, inc.

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