

Mastercam® 2018



From the
PRESIDENT'S
Desk ►

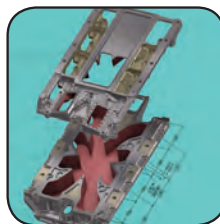
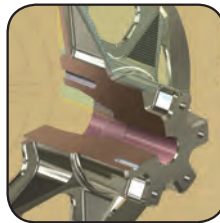
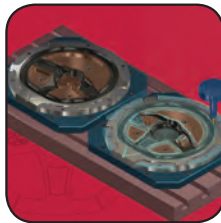
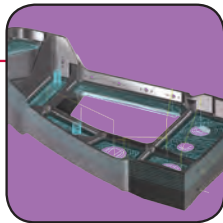
Mastercam 2018 is here...

...bringing a new suite of programming tools focused on delivering speed, automation, and efficiency to your shop. In this newsletter, we will introduce you to the next generation of the world's #1 CAD/CAM software, which includes Dynamic Motion improvements, Multiaxis and Mill-Turn enhancements, and many Design and System features to make your job easier.

There are many more features in the Mastercam 2018 release that we just couldn't fit in this newsletter, so please be sure to visit whatsnew.mastercam.com for more information.

And, with the release of Mastercam 2018, we are once again hosting LIVE Rollout events throughout the world so you, our customers, can see it in person. The worldwide Rollouts deliver information-packed sessions demonstrating what Mastercam 2018 can do for you and your shop. For more information, please visit www.mastercam.com/Communities/Events.

Thank you for being a loyal Mastercam customer and we hope you find Mastercam 2018 to be a powerful tool in the shop! **Meghan West, President**



If you are interested in learning about new features in Mastercam, visit whatsnew.mastercam.com.

this issue

P2
System
Features

P3
Design
Improvements

P4
Milling

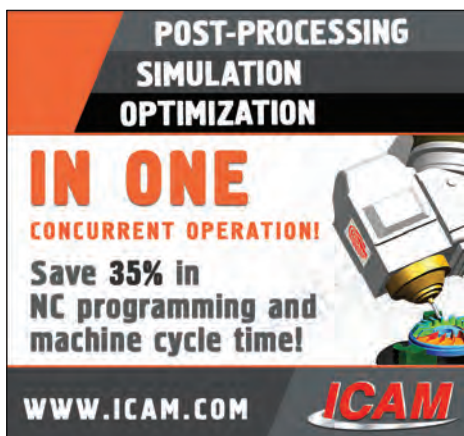
P5
MCfSW

P6
Mill-Turn

P7
Other
Product
News

Third-Party Advertisements

Throughout this newsletter, our partners offer complimentary solutions that can help increase your shop's productivity.

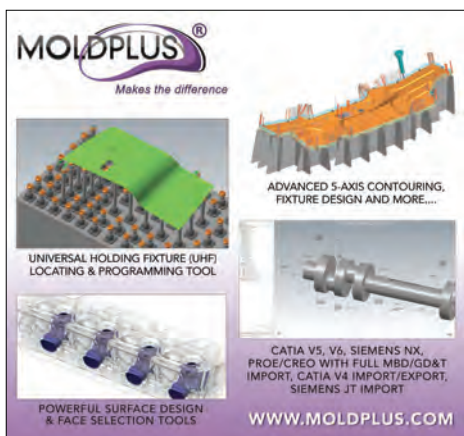


**POST-PROCESSING
SIMULATION
OPTIMIZATION**

IN ONE
CONCURRENT OPERATION!

Save 35% in
NC programming and
machine cycle time!

WWW.ICAM.COM **ICAM**



MOLDPLUS®
Makes the difference

UNIVERSAL HOLDING FIXTURE (UHF)
LOCATING & PROGRAMMING TOOL

ADVANCED 5-AXIS CONTOURING,
FIXTURE DESIGN AND MORE....

CATIA V5, V6, SIEMENS NX,
PROE/CREO WITH FULL MBD/GD&T
IMPORT, CATIA V4 IMPORT/EXPORT,
SIEMENS JT IMPORT

POWERFUL SURFACE DESIGN
& FACE SELECTION TOOLS

WWW.MOLDPLUS.COM



Streamingteacher
Your Place. Your Time. Your Pace.

**LEARN
Mastercam 2018**

System Enhancements

In the Mastercam 2018 release, there have been many system enhancements that can be seen throughout the entire Mastercam Family of Products. We will be discussing the big highlights in this article, but be sure to visit whatsnew.mastercam.com for a list of everything in this new release.

Analyze Toolpath – Now includes XYZ location data corresponding to the mouse location at the start or end of the toolpath segment. This is in addition to the feed and speed data that is shown between the start and end of the segment. And, Analyze Distance Along Curve allows you to find the distance along a line, arc, or spline edge without having to trim or break it first.

Command Finder – Searches for and opens Mastercam functions. It opens as a floating window that can be repositioned and remains on-screen until you close it. Enter a word in the Command Finder and Mastercam returns a list of possible matches that include a short description of the function.

Endpoint Display – Has been enhanced, now displaying as solid white circles with a black outline to provide contrast with different geometry and background colors.

Probe Operations – Are now associated to their selected geometry. As with other operations in Mastercam, when you move or modify the geometry used in the Probe operation and regenerate, the operation will update with the new geometry. You no longer need to recreate the Probe operation.

Mastercam Simulator – You can now create bookmarks that refer back to various points during Verify, Backplot, or Simulation. The bookmarks are then displayed on the playback bar. Simply select any bookmark icon to display the part as it is at the moment. You can now create a presentation from Mastercam Simulator. Saving a presentation lets you preserve your settings and layout of your Simulator session and can be used to show a simulation on any computer, even if it does not have Mastercam.

Planes Manager – You can now drag and drop a supported file onto the Planes Manager to import the file's planes. And, two new standard planes, Iso Reverse and Trimetric, have been added to the Planes Manager.

System Configuration Enhancements – You can set a default file type to be used when opening a file. You can also set a default color for the selection window on the Colors page. Mastercam 2018 has been updated to the Parasolids 29.1 kernel, the latest Parasolids kernel. This allows for compatibility with other CAD modelers, new features and capabilities, and more.

Work Offsets – When you update a plane's work offset, there is now an indication of how many operations are affected by the change. After selecting an update option, the right side of the dialog box shows which planes will be updated. You can select different update options and see which operations will be affected before committing the change.





What's New

What's New in Design?

Mastercam 2018 includes many new features and enhancements to the Design side of the software.

Here are some of the important additions in Mastercam Design. Since we didn't have room to go over all the great enhancements, please visit whatsnew.mastercam.com for a complete list, explanation, and videos showcasing Mastercam's functions.

Angle Sweep – To improve the creation of more complex wireframe functions and when creating or editing primitives, there are now on-screen sweep and rotate controls. These controls can snap to the AutoCursor positions of existing entities, and can recognize horizontal and vertical snap positions based on the gnomon orientation.

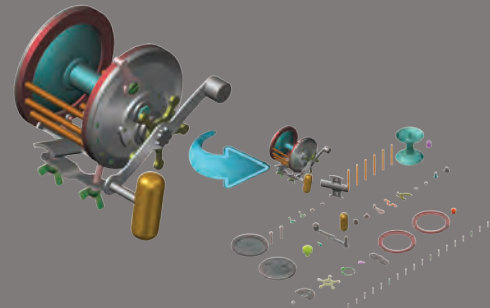
Align Note – Previously available as a C-Hook, Align Note is now available as a Drafting function and allows you to move existing notes and labels in such a way that they are more organized and easier to read.

Hole Axis – Hole Axis is now easier to use with a large number of holes, especially holes with matching diameters that are selected as a group. It has new Direction options that allow you to change the direction of the holes easily, and new Orientation options determine the direction of the top of the holes.

Solid Sweep – Solid Sweep has more options and controls available. It now supports 3D Along curves in addition to 2D Along curves. *End Profile Chains* allows you to transition from one profile to another while traveling down the *Along Chain*. Use a *Guide Chain* to influence a Solid Sweep operation by introducing a chain that alters the shape of the results. Solid Sweep used to require a fillet between entities on the Along curve when there was a change in direction of more than 5 degrees. In Mastercam 2018, the profile geometry easily runs along sharp corners, and if any are encountered, Mastercam miters them. This gives you greater flexibility and precision when modeling.

Undo/Redo – In previous versions, any changes made to solid models using the Model Prep functions could not be reverted. Now, you can undo and redo up to twenty-five Model Prep functions using the new *Undo* and *Redo*, located in the Solids Manager. They only affect changes made using the Model Prep functions.

Seam Control – To make Roll and Unroll results easier to visualize, a new gnomon based rotation control is now available. You can visually rotate the seam to see how geometry will respond, as well as snap to AutoCursor positions.



Other Design Features:

Wireframe Enhancements – The Curve functions, Curve One Edge and Curve All Edges, have been enhanced and will help in the creation of more accurate containment boundaries. Curve All Edges also allows you to create curves from only exterior edges of a solid and ignores the inner edges.

Previously only available in 3D mode, Curve One Edge and Curve All Edges can now create geometry in both 2D and 3D mode. By supporting 2D geometry creation, geometry created from the selected edge can be projected to the current Cplane and Z depth, instead of requiring you to use another function, such as Project.

Edit Spline can be used to refine and prepare splines. It works on NURBS splines and converts any selected lines, arcs, or parametric splines into a NURBS. It allows you to modify the tangencies and magnitudes of a curve at the spline's node points, or to edit tangencies at the vectors of control points. You can move, rotate, or lengthen the tangent vector or move the control point. For more control and a finer adjustment of the curve, you can add or remove node points, or increase the number of control points.

Modify at Intersection is a new function that allows you to trim, break, or create points with lines, arcs, and splines where they intersect with surfaces, solid bodies, solid faces, and solid sheet bodies. This is particularly useful when modeling or prepping for multiaxis toolpaths.

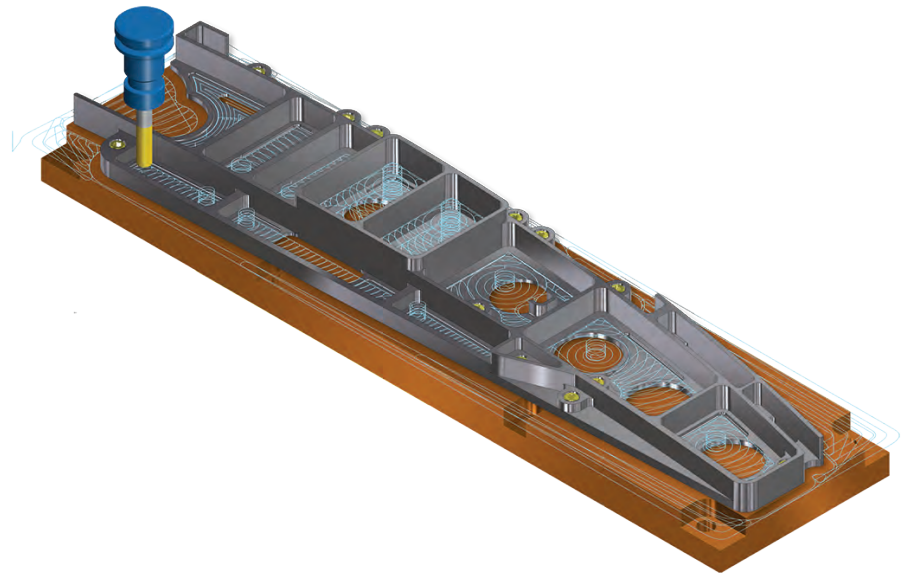
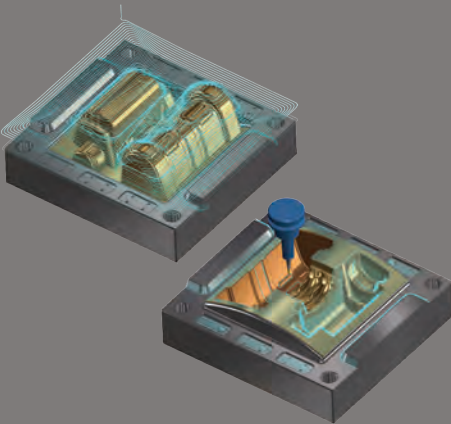


Multiaxis Features:

The **Collision Control** page has been redesigned for 2018 to make programming easier and more intuitive.

Common direction has been added and keeps the tool at a uniform tilt angle to reduce tool and machine motion. **Maintain tilt** keeps the tool tilted during retract and feed moves in an effort to reduce tool and machine motion.

Dropping has been added to the 3- and 5-axis toolpaths and projects the 3-axis toolpath onto the selected surfaces, adjusting the tool vectors to be normal to the surface.



Mastercam 2018 includes many new features and enhancements to the milling side of the software.

2D High Speed Toolpaths

Mastercam 2018 includes many enhancements, bringing our world-leading milling product to the next level.

Stock awareness has been added to select 2D toolpaths and allows tool motion on the top, bottom, or both values of the stock. This provides a safer and more optimized tool motion.

You can also optimize **finish passes** based on your rough stock, giving you more efficient and safer tool motion. *Add between* creates a finish pass at all depth cuts and adds a specified amount of passes between depth cuts. *Per number of* adds a finish pass only at every specified depth cut.

Plunge entry is now supported for Dynamic Mill toolpaths, which allows you to safely plunge directly into material.

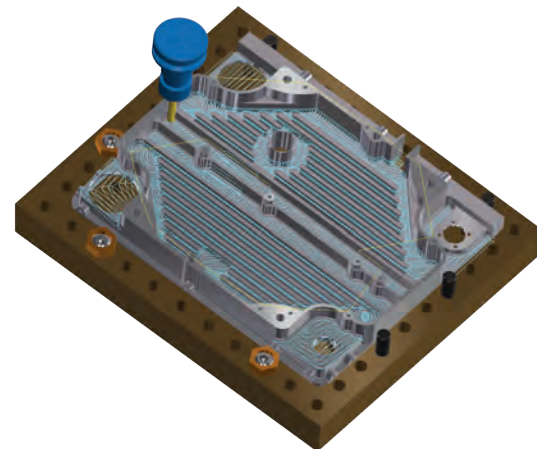
Spring Passes has been added and creates additional finishing cuts along the same path as the last finish pass. You can use this option to create additional finish passes with a spacing of zero. This can be used with thin material that may have flexed away from the tool during previous cuts.

3D High Speed Toolpaths

The workflow for all 3D High Speed toolpaths has been streamlined and gives you precise control over exactly where to cut.

Follow containment has been added to the Waterline toolpaths and allows you to follow the containment boundary back to the start of the cut, creating clean closed passes. This helps to avoid partial cuts when a contained boundary was directly on top of a vertical surface.

High Speed Hybrid toolpaths now offer **smoothing** controls and deliver a finer finish, requiring less handwork.



Mastercam® for SOLIDWORKS®

Mastercam 2018 for SOLIDWORKS delivers many new features to keep your shop running at peak performance. All of the Mastercam Mill and Lathe toolpath improvements that are referenced in this newsletter are also included in Mastercam 2018 for SOLIDWORKS, as well as the following enhancements.

In Mastercam 2018 for SOLIDWORKS, Face center points were introduced and were tied to Center points. Now, Face center points and Center points are independent of each other. You can use the Make Center Point option to create a single point in the middle of the Bounding Box and/or use the Make face center points option to create points at the center of each face.

New Router Support

Mastercam 2018 for SOLIDWORKS introduces support for Router, including Saw toolpaths, Block Drill toolpaths, and more. All specialized Router toolpaths are available through the menus once you activate a Router machine.

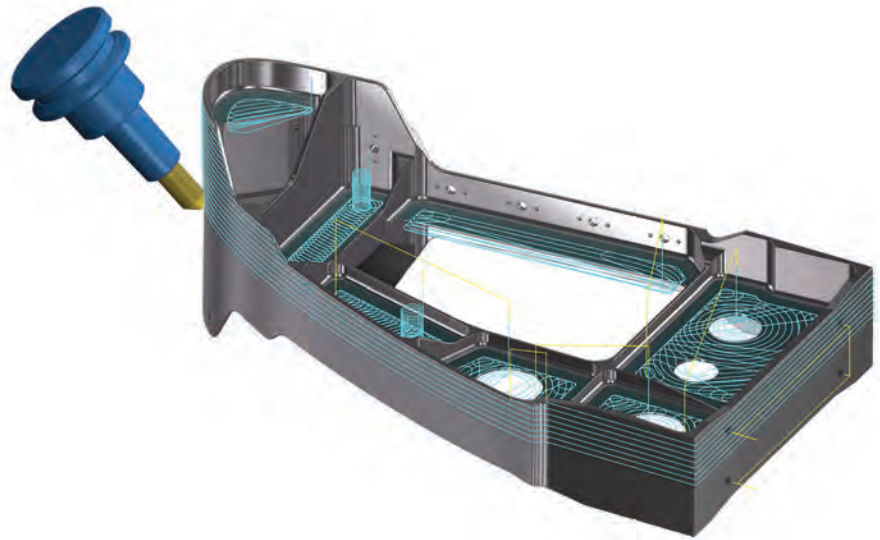
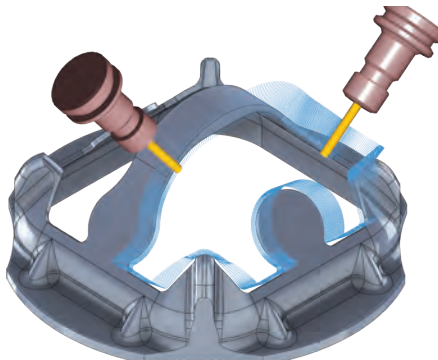
Automatic Toolpathing (ATP) automates the process of assigning toolpaths to geometry for recurrent parts using a machining strategy. A machining strategy contains one or more level names that are associated or mapped to Mastercam operations. Create your operations library, read your part files, create your strategy, pick your cutlist, and then begin cutting your toolpaths.

Block Drill toolpaths take advantage of the capabilities of a drill block or gang tooling. During a Block Drill operation, Mastercam for SOLIDWORKS selectively drops different drills depending on the holes that need to be drilled. Use the different drills on the drill block to automatically drill several holes at the same time, or mount different size drills in the block and have Mastercam for SOLIDWORKS drop the proper drill for each hole.

Saw toolpaths are used to drive a saw blade cutting along a straight line. You can program multiple saw cuts using a single tool plane in a single operation.

And, with Mastercam for SOLIDWORKS Router, you can also create door and stair geometry.

Those who purchase or update to Mastercam Mill 2018, Mastercam Lathe 2018, or Mastercam Router 2018 will also be able to use the corresponding version of Mastercam for SOLIDWORKS, giving you "The Best of Both Worlds."



Add Value to Mastercam

- Automated Inspection
- Reverse Engineering
- Guided Assembly



www.verisurf.com • 866-340-5551



POSTABILITY
DRIVING CNC

Post processors that are requested worldwide.



www.inhousesolutions.com/posts

camInstructor
BOOKS ONLINE COURSES VIDEOS

Specializing in **Mastercam**
Training Products

Also available:
Mastercam for SOLIDWORKS
SOLIDWORKS
Haas CNC Training Materials
Online Teacher Training

camInstructor.com 1.877.873.6867

PROSHOP 360.515.7576

COMPREHENSIVE WEB BASED ERP, MES AND QMS SOFTWARE
FOR THE MACHINING INDUSTRY

Designed and Built in a Shop by Machinists

- Connects every level of your shop
- The only ERP Partner with Mastercam
- Sync tool library, embed simulation videos & more
- Completely integrated Quality System
- Syncs with Quickbooks or your accounting system

It's all at Your Fingertips

- Job Travelers, FAls, Scheduling, Costing
- Have the data to make the right choices about the jobs in your shop! Increase profits, improve scheduling, go home on-time! Your wife misses you!

ADION AdionSystems.com

NEW! eMastercam.com
Sitewide Subscription

eMastercam.com
eBooks & eCourses

\$39.99/month
or less to access
all eBook and
eCourses!

**www.eMastercam.com/
sitewide-subscription**

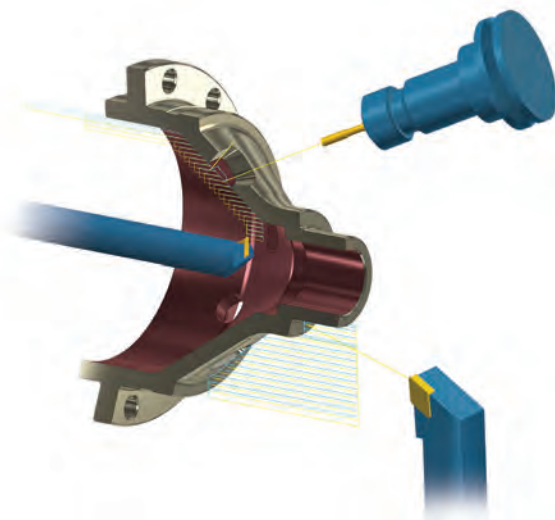
Robotmaster
CAD/CAM FOR ROBOTS

Mill-Turn Improvements

Mastercam Mill-Turn makes machining on today's high-powered turning centers and full-function machining centers simpler and easier than ever before. These machines eliminate multiple setups, manual part handling, redundant fixturing, and reduce idle time—boosting production and maximizing profitability.

Center Support and Tailstock Operations

With the support for center tools, you can now create a Center Point toolpath, which is used when you need to mount a center tool in a turret and engage the tool for live center support.



And, you can now create Tailstock Operations to quickly create a set of operations to load a center tool, move to stock clearance and, if applicable, advance the quill. The available tailstock options differ depending on the selected machine. You can also create a set of tailstock operations to retract the quill and tailstock, and unload the center tool.

Mastercam now supports center tool definitions. Now, when creating a Tailstock or Center Point toolpath, the center tool will display in the graphics window.

General Enhancements:

- You can now chain geometry on multiple planes, similar to Lathe chaining. All features can be toolpathed on multiple sides in one operation.
- Edit common parameters is now available in Mill-Turn.
- If you import an operation, Mastercam will check for the Work Coordinate System (WCS) of the operations.

Job Setup Enhancements:

- You can now select *Enable Right Spindle* or *Enable Left Spindle* when you want to use the second spindle for a stock pull.
- *Manually Define Right Spindle Stock* can be selected to choose the stock for the second spindle, which is the stock that shows in simulation when it is launched. This entity can be derived from a stock model that reflects the initial spindle's operation.
- You can now use STL models to define stock.
- Ability to machine different parts on the left and right spindles.

Tool Locators for B-axis Heads

Mastercam 2018 now includes support for tool locators for B-axis heads. Spindle tool locators enable proper positioning of tools in the B-axis head. This support greatly improves collision detection and gives a clear representation of how the tool is mounted in the machine. Additionally, multi-station tool locators are supported which allows for realistic simulation of machine tool change motion. Instead of moving all the way to the tool change position, the spindle can move to any point you specify, index the locator, and quickly get back to cutting. These new locators are fully utilized in collision detection, including all tools mounted in the locators, regardless of which tool is currently cutting. The Tool Setup Manager also supports spindle tool locators.

Tooling Enhancements

Improved Inch and Metric Support:

Mastercam now supports mixing inch and metric tools and holders when building tool assemblies in Mill. You can use metric tools in inch part files and inch tools in metric part files. This includes proper tool compensation and scaling in Backplot and Verify.

New Tool Importers:

Support has been added for importing tool assemblies from CoroPlus and MachiningCloud.

Other Products

What's New?

Lathe Enhancements

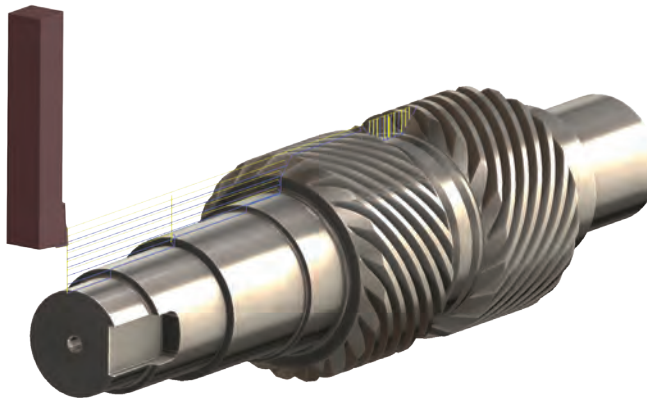
Chip Break is now available for Face and Finish toolpaths (already available for Rough and Contour Rough) and can be used to establish when chip breaks occur. This is a good option when working with stringy materials such as aluminum or plastic, and allows you to set length and time conditions, retract, and dwell options.

Groove Enhancements:

The *Bi-Directional* option has been changed to *Bi-Direction, Alternating*. After the toolpath's initial plunge, it alternates between negative and positive directions until the required material is removed.

When cutting a face, Mastercam now calculates the optimal plunge point so that each side of the tool removes the same volume of material. This balances wear between the two sides of the tool.

Prevent up cutting defines the orientation of the non-cutting portion of the insert. Mastercam now generates a toolpath that cuts only with the lower portion of the tool insert.



Lathe Rough now has **Section turning**, which lets you cut stock in multiple sections. You can break the toolpath into equal sections, into sections of exact distance with one section being the remainder, or equal sections based on the given distance.

The **Stock Model** operation now supports Lathe and Mill-Turn part handling operations. After the part handling operation is complete, the stock model will end up with the stock boundary.

Mastercam has partnered with Sandvik Coromant on the development and support of their innovative **PrimeTurning™** method and **CoroTurn®** Prime tooling. With these tools and methods, Mastercam allows you to turn in conventional turning and PrimeTurning directions. Find more at whatsnew.mastercam.com.



Our Offices

Corporate Headquarters

CNC Software, Inc
671 Old Post Road
Tolland, CT 06084
1-860-875-5006
1-800-228-2877
info@mastercam.com
www.mastercam.com

West Coast Office

5717 Wollochet Drive NW
Gig Harbor, WA 98335
1-800-ASK-MCAM

Our Community

facebook.com/mastercam

twitter.com/mastercam

youtube.com/mastercamcadcam

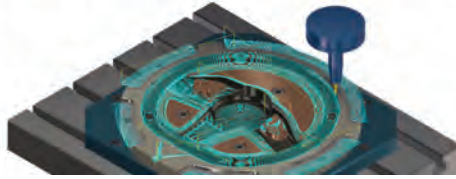
linkedin.com/company/cnc-software

plus.google.com/+mastercam

instagram.com/mastercamcadcam



Mastercam® 2018 ROLLOUT



Come See Mastercam 2018 LIVE!

Mastercam 2018 LIVE is hitting the road! These worldwide Rollouts deliver information-packed sessions demonstrating what Mastercam 2018 can do for you and your projects. Take a look at the dates on the Events page (www.mastercam.com/Communities/Events) to find an event near you.

Attendees get a chance to win some great prizes—free subscriptions to Mastercam University® online training, Mastercam swag, or even a seat of Mastercam Mill. This is truly an event you won't want to miss. More events will be added all the time. Please be sure to check the site often or contact your local Mastercam Reseller for events in your area.



cnc software, inc.

671 Old Post Road
Tolland, CT 06084
USA

Ses3000 CNC ^{25th}
Your Solution Partner...