

Fourth Quarter 2010

# Mastercam® Notes

Customer Newsletter



In this issue of Mastercam Notes, we will focus on a few specific features from the Mastercam X5 release and how you would use these functions in your every day work. We will also look at a few stock issues in-depth, such as how to eliminate ring burrs in Lathe and how stock to leave is applied to Wire.

There are also a couple customer profiles, which we think you will enjoy, featuring Jet Propulsion Laboratory and Taylor Guitars.

And lastly, we are showcasing a few third-party products that can get you up to speed with Mastercam X5. These products are training tools that can assist you with learning the new Mastercam X5 release.

We hope you find this issue exciting and that you learn a few things about the X5 release!

## Product Highlights



**Mill Feature** .....2  
Mastercam's Hybrid Toolpaths



**MCfsw Feature** .....3  
Mastercam X5 for SolidWorks is Released



**Wire Feature** .....4-5  
How Stock to Leave is Applied in Mastercam Wire



**Router Feature** .....6-7  
Toasting the Toolmaster - Taylor Guitar and Mastercam



**Lathe Feature** .....8  
Eliminating Ring Burrs

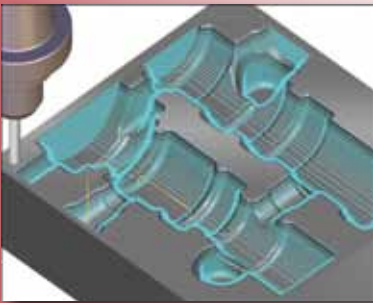


**Add-In Feature** ..... 9-11  
Third Party Products for X5

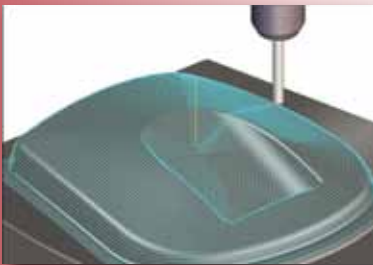
# MILL

## Hybrid Finishing 101

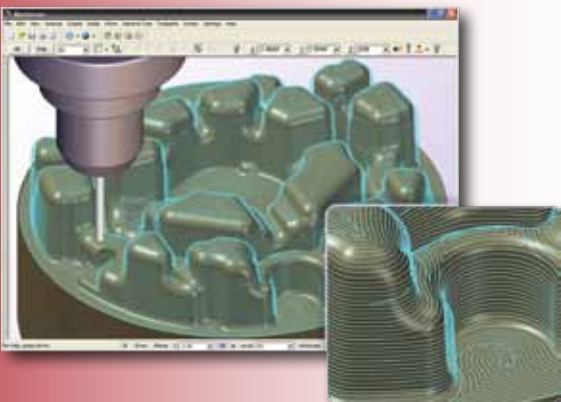
Mastercam X5 introduces a new cutting technique called Hybrid Finishing. The word “hybrid” has become more and more common: hybrid cars, hybrid bikes, even hybrid plants. But hybrid finishing? What is it? How does it work? And what can it do for you?



Constant Z cutting is excellent for steep-walled areas.



Scallop machining maintains a consistent 3D stepover.



Hybrid Finishing smoothly combines different cutting techniques for steep and shallow areas.

### What it is

Mastercam X5's new Hybrid Finishing toolpath combines two separate, proven cutting techniques to provide faster programming and a better single-pass finish. The two techniques it combines are:

**Constant Z cutting.** Also called “waterline” cutting, this type of toolpath carries out its cutting pattern at a given depth with no up or down movement in Z, and then moves down to do the next level. Essentially a series of contour cuts, this technique is especially useful on high vertical walls or very steep surfaces where other techniques might leave a “stair step” look.

**Scallop machining.** This high speed machining technique measures its stepover in 3D as it moves along the surface, rather than parallel to the toolplane as other techniques do. The result is cut spacing that is maintained as the surface angle changes, creating a consistent scallop height. This technique is good for complex surface shapes that need a consistent finish.

### How it works

Just like a hybrid car switches between gas and electric as the situation demands, Hybrid Finishing switches between methods based on the shape of the part. Steep areas will be machined using Constant Z cutting while shallower areas will be handled with scallop machining. The toolpath switches seamlessly between techniques and cuts in a logical optimized order.

Since shops have their own preferences and best practices, the setup for this technique is flexible. You can set the “steep” and “shallow” values in a variety of ways, letting you dial in exactly how you want the software to perform. Or you can just go with the defaults that we used to test cuts in our own shop.

### What it can do for you

Hybrid Finishing can help streamline your programming. Instead of creating two separate toolpaths to achieve the same end, you can now do it with just one. Since Mastercam automatically creates smooth transitions between the techniques, you get a better finish off the machine. Give it a try! ☒

# Mastercam for SolidWorks

## Mastercam® X5 for SolidWorks® is Released

Mastercam X5 for SolidWorks includes many new features that improve the speed and efficiency of your machining jobs.

### New OptiRough Toolpath

The new 3D surface high speed OptiRough toolpath supports cutters capable of machining very large depths of cut. It uses an aggressive, fast, intelligent roughing algorithm based on Mastercam's 2D high speed dynamic milling motion. A single OptiRough toolpath can cut material in two directions: on step downs (-Z) and step ups (+Z). This highly efficient bi-directional cutting strategy removes the maximum amount of material with the minimum of step downs, significantly reducing cycle times.

### Hybrid Finish

Hybrid machining is the latest addition to the list of finishing toolpaths, and is designed to address both steep and shallow regions of your models. Hybrid finishing intelligently blends two efficient cutting techniques in a single toolpath. This new toolpath evaluates the model shape and smoothly switches between Constant Z cutting and Constant Scallop machining. The result is a dramatically finer finish with less work (see more information on previous page).

### Start Hole Toolpaths

Use the Start Holes function to automatically create drill toolpaths at the plunge points for one or more toolpaths. Mastercam intelligently scans individual depth cuts and passes for each toolpath and, if necessary, creates toolpaths for multiple plunge points.

### Toolpath Trim

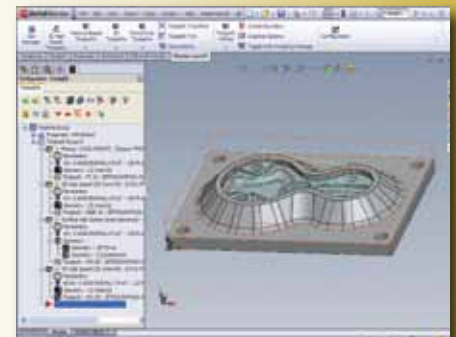
A toolpath trim operation is a special type of operation that lets you select one or more toolpaths and trim them to one or more closed chains. When you create a trim operation, Mastercam stores it in the Toolpath Manager, and also keeps the original source operation, marking it with a special Trim icon.

### New 2D HST Toolpaths

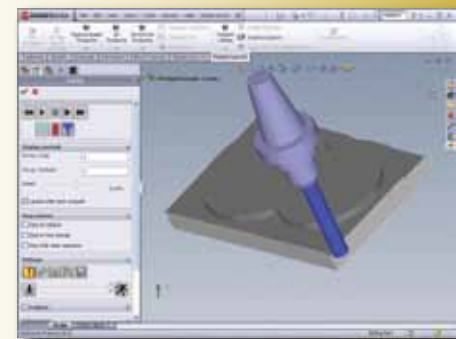
The original Dynamic Mill toolpath is now split into two application-specific toolpaths: Dynamic Area Mill and Dynamic Core Mill. Mastercam X5 for SolidWorks also adds two new dynamic milling toolpaths to the 2D high speed toolpath group: Dynamic Rest Mill and Dynamic Contour.

Additional enhancements to Mastercam X5 for SolidWorks include:

- Point Manager enhancements
- Chain Manager enhancements
- Plane rotation enhancements
- Island Facing in 2D HST toolpaths
- And much more! ☒



2D High Speed Area Mill toolpath.



Mastercam for SolidWorks verifying a set of toolpaths.

## ProDrill

Advanced feature based drilling for Mastercam  
Functions on surface and Solid models!


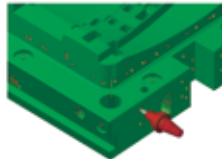
ProDrill Version 3 MR2 with:

1. IMPORT COLORED TAP AND REAM FEATURES from external CAD systems.
2. Easy CUSTOM/FORM TOOL USE, OPTIMIZED TOOL SELECTION CRITERIA and several new enhancements to cater to complex drilling jobs!!!

Automatically create and insert complex drilling geometry on Solid and 2D models. CAD system independent!!!

Complex 4 and 5 axis drilling at blinding speed!

Rapid Return on Investment!



MOLDPLUS Making the difference

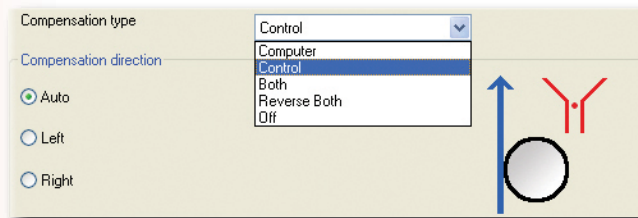
ProDrill is a product of Tecnicim S.L. and Moldplus S.A.  
Solutions that make a difference! [www.moldplus.com](http://www.moldplus.com)  
[info@moldplus.com](mailto:info@moldplus.com)

# WIRE

## How 'Stock to Leave' is Applied in Mastercam Wire

The **Stock to Leave** values, established on the Wire/Power page in the wirepaths dialog boxes, set the amount of stock to leave for all passes. **Stock to Leave** offsets the wire in the direction specified by the **Compensation direction** on the Compensation page.

**Stock to Leave** options are available only for Contour and 4-Axis wirepaths. The **Apply additional offset to** section is not available for No Core wirepaths, and none of the **Stock to Leave** options are available for Canned or Point wirepaths.



### Stock to leave application

- Program coordinates
- Machine offset register

The **Program coordinates** and **Machine offset register** options determine how the **Stock to Leave** value is applied to a wirepath. The defaults for these options are set in the **Stock to leave application** section of the Cutter compensation page in the Control Definition.

When **Program coordinates** is selected and:

- **Compensation type** is set to **Control**, all the wirepath coordinates in the NCI are offset by the **Stock to Leave** value.
- **Compensation type** is set to **Computer**, **Both**, or **Reverse Both**, all the wirepath coordinates in the NCI are offset by the **Total Offset value** (Wire Radius + Overburn + Stock to Leave).

Note: For operations based on TECH power libraries, the Offset values are equivalent to the calculated Radius + Overburn in the power libraries which are not TECH power libraries. These offset values are displayed in the **Offset** column on the TECH library Wire/Power page.

- **Compensation type** is set to **Off**, all the wirepath coordinates in the NCI are offset by the **Stock to Leave** value. When **Compensation direction** is set to **Auto**, Mastercam knows to which side of the wirepath to apply the offset, or you can choose **Left** or **Right** to manually set the compensation direction.

Note: When **Compensation type** is set to **Computer** or **Off**, Mastercam does not create an offset command line (G41/G42) output to your machine control. In this case, Mastercam Wire defaults the **Stock to leave application** to **Program Coordinates**, and the **Machine offset register** option is disabled.

When **Machine offset register** is selected and:

- **Compensation type** is set to **Control**, Mastercam does not offset the wirepath coordinates in the NCI. The post processor takes the combined offsets (Wire Radius + Overburn + Stock to Leave) and applies that value to the offset register in your machine control.
- **Compensation type** is set to **Both** or **Reverse Both**, Mastercam offsets the wirepath coordinates in the NCI according to the total value of **Wire Radius + Overburn**. The post processor takes the additional offset (**Stock to Leave**) and applies the value to the offset register in your machine control.

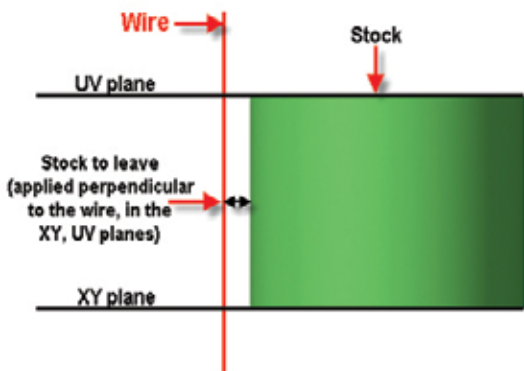
In any case where **Stock to Leave** values are applied, if the **Compensation direction** is set to **Auto**, Mastercam recognizes either a defined or implied start point, allowing it to determine to which side to apply the **Stock to Leave**.

**Notes:**

- The **Stock to Leave** values are not associated with the active POWER library. Therefore, the **Stock to Leave** field and related **Apply additional offset to** choices on the Wire/Power page remain active, regardless of whether or not the wirepath is associated with a POWER library.
- When you load an MCX file that was created in any Mastercam Wire version prior to X2 MR2, Mastercam Wire checks the **Stock to Leave** values programmed in the active POWER library. If the **Stock to Leave** values in the assigned POWER library contain a constant value for all the operation's passes, Mastercam loads this value into the **Stock to Leave** field.
- If varying **Stock to Leave** values have been set for different passes in the active POWER Library (allowed in Mastercam Wire versions previous to X2 MR2), a dialog lets you know that these varying **Stock to Leave** values will be added to the Wire **Overburn** values for each pass. In this case, **Stock to Leave** is set to 0.0, reproducing the previously established wirepath values.
- All Contour, No Core, and 4-Axis wirepath operations created in any Mastercam Wire version prior to X2 MR2 are marked dirty, and have the **Associate to library** option unchecked.
- When you apply a **Stock to Leave** value and **Compensation** is set to **Off** (on the **Compensation** page), the **Stock to Leave** amount will be reduced by the wire radius plus the **Overburn** value.
- The **Machine offset register** parameters do not apply to No Core wirepaths. Therefore, the **Apply additional offset to** options will not appear in the **Wire/Power** page for No Core wirepaths, and the **Stock to Leave** value is calculated as if the **Machine offset register** choice was selected.
- **Stock to Leave** settings are not available for Canned and Point wirepaths. Therefore, the **Stock to Leave** area (as shown below), including the **Apply additional offset to** choices, will not appear in the **Wire/Power** page for those types of wirepaths.



Find updated Operations/Defaults files for Wire X5 at [www.mastercam.com/Support/Downloads/MastercamX5](http://www.mastercam.com/Support/Downloads/MastercamX5)



- On 4-axis wirepaths, the **Stock to Leave** value is applied perpendicular to the wire, not in the XY and UV planes. Therefore, the actual offset of the XY and UV planes will vary, depending on the angle of the taper.

**IMPORTANT:** Applying **Stock to Leave** settings may require post customization. ☒



# ROUTER

## Toasting the Toolmaster - How Mastercam Helped Bob Taylor Carve Out His Future



During Bob Taylor's factory tours, people often ask which big-name artists were the "early adopters" who helped lift the company from relative obscurity to the radar of public recognition. A handful of names are usually trotted out — Neil Young, John Denver, and Prince, to name a few — but if you ask Bob Taylor, he'll cite a different kind of key player, one whose name is far from any venue marquee, yet whose impact on Taylor's success is greater than all of the above names put together. His name is Mark Summers, and he's the co-founder and president of CNC Software, Inc., whose flagship product, Mastercam, enabled Bob to revolutionize the way that Taylor guitars are made.

It was 1989 when Bob took the plunge, purchasing his first CNC mill and Mastercam software. He was making about seven guitars a day. Bob vividly remembers learning to use the Mastercam software on his IBM AT computer. "It wasn't easy, but it wasn't hard," Bob recalls. "What was great was that it made sense. It had the best user interface; the menu structure meant something, as opposed to other CAD programs where you'd have to learn what every word meant. The next thing you know, just through figuring it out, I'd drawn a guitar bridge like I'd drawn on plans, except, when I said six inches, it was six inches, not six inches and the width of a pencil lead. When you're making a guitar and you're trying to be accurate, lines have weight because they have dimension."

Once Bob had a drawing of his bridge, Mastercam allowed him to use the dimensions to generate the toolpaths for cutting and shaping it from wood. "It would memorize the shape of my bridge by zooming around the line I drew," he explains. "So, now it has a toolpath, and then another screen comes up, and it asks you all these things that it needs to know: What's the diameter of your cutter? How fast do you want to spin the spindle? Which direction, clockwise or counter-clockwise? Which side of the line do you want to cut on? And then you click it and it spits out G-code that the machine can read. I'd go back and forth between my office and the machine at night and figure it out. And then I'd push the button on the machine, and it would start feeding and cutting, and when it was done I'd pick the thing up and I'm like, whoa, I just cut this piece of wood into the shape of a bridge on this machine! I can do this. It's like learning the first three chords on your guitar; it doesn't ever get more fun than that."



Bob Taylor with Meghan Summers during their tour of the Taylor Guitar factory.

Bob says that Mastercam marked the first time that he as a guitar craftsman was able to get the precise shapes and slots he really wanted. "The cool thing was, when I was done with that, I could say to one of our employees, 'Hey, push this button for the rest of the afternoon, and put this wood on, and take this wood off, and make bridges.' Pretty soon they're learning to run the machine and they're setting it up and changing the tools, figuring out how the programs work. Meanwhile, I'm using Mastercam to draw a fingerboard and a guitar body. So, our guitars got more accurate, and more people in the shop could make more accurate parts."

Many Taylor innovations, such as the NT Neck®, would never have been possible without the precision and consistency of Mastercam software. The ever-evolving marriage of quality and efficiency that Mastercam has made possible is now deeply entrenched in much of Taylor's everyday production process. Taylor



currently operates 26 CNC mills in its main factory complex in El Cajon, California, and will soon be adding its 16th computer mill to its plant in Tecate, Mexico.

“Without Mastercam, Taylor wouldn’t look like we do at all,” Bob reflects. “You have to stop and appreciate someone like Mark, who had his dream, which nested with mine, and allowed me to grow Taylor.”

And yes, Mark plays guitar. He owns three Taylors: an 814ce, a Jewel signature model, and a slot-head 30th Anniversary Grand Concert XXX-RS. “I’ve been playing for about 35 years, but I always tell people three months because that’s about how good I am,” he quips.

Despite the fact that Mark and Bob have known each other professionally for years, Mark hadn’t actually visited the Taylor factory until this past March. He says he was surprised to discover how big Taylor’s operation truly was. “I had no idea it was so involved. It seemed very well organized and well run.”

Mark made the trip with Meghan, who came with a special agenda: She wanted to spec out a Build to Order guitar as a wedding gift for her fiancé Brendan, who has played guitar most of his life but had never owned his own. “I wanted to do something really special, and a guitar was the best thing that came into my mind,” she says.

Bob gave Mark and Meghan a personal tour, showing them the many aspects of production at Taylor that involve Mastercam programming, after which Bob and Taylor’s Joe Bina helped Meghan design the guitar. Meghan selected a spectacular master grade set, with a koa top as well. She opted for a GS body with a Florentine cutaway, mother-of-pearl tropical vine fretboard inlay, rosewood binding, black and white purfling, an abalone rosette, and Gotoh tuners.

“When I first got the guitar and opened it up, the wood was absolutely spectacular,” Meghan says. “It took my breath away.” She surprised Brendan with the guitar the night before the wedding, prior to the rehearsal dinner. “Brendan was shaking as he opened up the case, and then he just stared in amazement, going, ‘I don’t even know what to say.’ It was a really, really special moment for us.” ☒



The finished guitar that Meghan ‘designed’ for her fiancé.



# LATHE

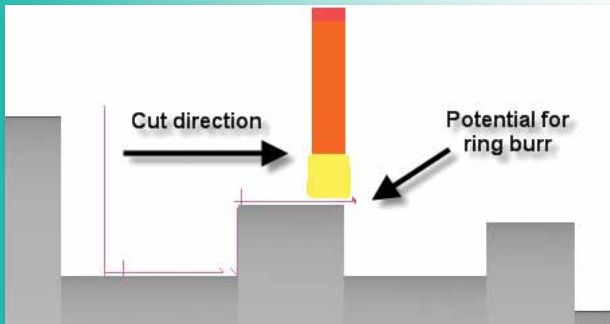
## Eliminating Ring Burrs in Mastercam Lathe

Mastercam X5 Lathe introduces several new enhancements, including a new toolpath strategy called Plunge Turn. This toolpath creates cutting strategies specific to a tool that can both plunge like a grooving tool, and turn like a finish tool, roughing and finishing all in one toolpath.

There are many benefits to this:

- Fewer tools (usually one) can be used to rough and finish the part.
- Because fewer tools are used, fewer tool setups and tool changes are needed.
- Because the toolpath can remove material in X (plunging) and Z (turning), more of the insert is being used which results in longer insert life.
- The combination of these factors results in much improved material removal rates and cycle times.

One of the features of this new toolpath is its ability to eliminate ring burrs while turning. So, one might ask, "what is a ring burr"?

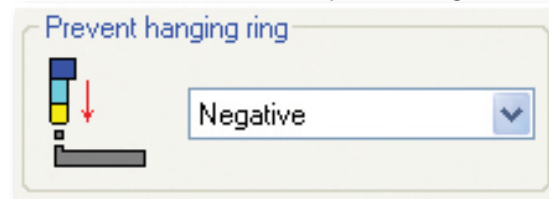


A ring burr can occur during a roughing or finish turning operation when the tool pushes material off of a shoulder and into open space, resulting in a ring of material that cannot be cleared from the part (see image to left).

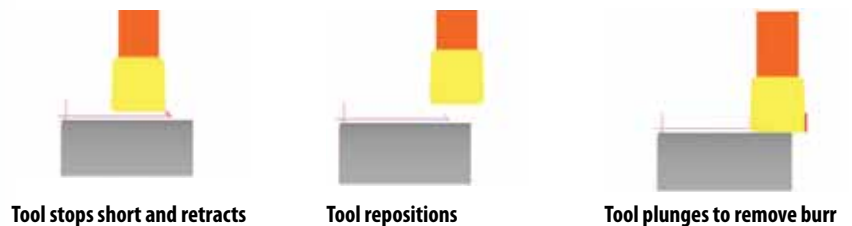
This can cause many issues from simple inconvenience to larger problems with automated part handling. In any case, an operator needs to manually remove the ring burr with wire cutters.

The new Plunge Turn toolpath in Mastercam X5 has a feature to prevent ring burrs from forming.

With this feature enabled, the finish toolpath will stop short of the shoulder and change to a plunging motion to avoid pushing a ring burr off of the shoulder. All the user needs to do is enable this feature with a simple drop-down selection



(above, right). Mastercam does the rest, by automatically adjusting all toolpath motion to avoid causing a situation where a ring burr can form. Here is what the adjusted finish toolpath will look like:



This motion will automatically occur anywhere a potential ring burr could occur. Because these tools can side-cut and plunge-cut, you are offered a greater range of programming flexibility and efficiency that is unmatched by traditional tools and cutting methods. ☒

# ADD-ONS

## camInstructor

camInstructor, a recognized Mastercam Training Partner, has provided schools and industry with books, eBooks, and online courses to help students and employees learn Mastercam quickly and efficiently and with great success.

Here is why the camInstructor method works...

### Certified Authors

The developers of camInstructor training materials are certified teachers, with over 10 years' experience, teaching CNC programming and Mastercam software. This ensures that their customers, whether students or employees, learn Mastercam quickly and efficiently without being overwhelmed.

### Content

The variety of content available includes Mastercam X through X5, and covers Mill 2D, Mill 3D, Lathe, Solids, 4-axis, 5-axis, Wire, and Art. All of the step-by-step instructions start at the beginning and proceed in simple-to-understand lessons that you can digest without becoming overwhelmed. Every lesson includes instructor-led videos that guide you step-by-step throughout the lesson. You can watch the videos for reinforcement; it's like having an instructor available on-demand.

### Variety of Options

Whatever your learning style, camInstructor offers a delivery method that suits you and includes books, eBooks, or online courses.

Books and eBooks are an excellent option for students taking a Mastercam course at a high school, college, or training center. All of the books include a DVD that provides the instructor-led training videos and Mastercam's Home Learning Edition software. The PC-friendly eBooks are identical to the printed version except that they are online and include videos and PDF files that can be printed and/or saved to your hard drive.

camInstructor online courses are perfect for employees and independent learners because they offer a number of features that the books/eBooks do not. The online courses guide you through the course content and provide a unique feature where you can e-mail a question at any time about any aspect of the course. You are never alone and have the peace of mind that comes with having access to a Mastercam Instructor.

For more information on what camInstructor has to offer, please visit [www.caminstructor.com](http://www.caminstructor.com). ☒



**Moldplus for Mastercam**  
Solutions for today's modeling challenges

- New 5-axis Design Toolkit:**  
Create complex 5-axis curves and vectors with a few clicks!
- Manipulate tilt vectors and set angles with superior ease!
- Easily add and remove new tilt vectors on created curves!
- 1-click graphical surface normal setting to same direction irrespective of plane direction!
- Minimum/maximum radius analysis tool.
- Easily change UV direction for surfaces.
- 1-click tangent and draft extensions
- Split complex models for exterior and interior faces with 1-click!

**MOLDPLUS** Solutions that make a difference! [www.moldplus.com](http://www.moldplus.com)  
[info@moldplus.com](mailto:info@moldplus.com)

Powerful and easy to use ELECTRODE maker features

**LEARN Mastercam TODAY!**  
Books with Videos  
Online Courses

Learning Mastercam with camInstructor is quick, easy, and affordable!

"The camInstructor online course was great. It helped me land a job quickly when I was unemployed. I'm now using Mastercam on a daily basis at my job."  
- Ken Johns, CNC Programmer

**camInstructor**  
877.873.6867 [www.caminstructor.com](http://www.caminstructor.com)

# ADD-ONS



## Tips For Manufacturing

Tips For Manufacturing has a long history of developing training material for CAD/CAM systems, Machine Tools, and CNC controls.

Going back 30 years, the company's owner, Mike Mattera, had developed classroom training material and a series of VHS training tapes. These days, the Tips For Manufacturing product line is focused on video training material for Mastercam. Having been involved with CNC Software since V2 (1989), Mike has seen Mastercam mature. His training CD's bring a keen insight, reflecting the strong history of the Mastercam product line.



The Tips For Manufacturing training system is simple. You learn by watching a series of project-based videos, each approximately 4 to 7 minutes in length, and then practice the lessons in Mastercam. The focus is always on the "process", not "feature." This is what makes the Tips For Manufacturing CDs more practical for "real world" training. Having the training on a simple-to-use, self-contained CD offers the user a number of advantages. A few are below:

- The CD is easy to move from PC to PC (work to home / job to job).
- No high speed internet access is required, making it suitable for remote locations.
- There is no recurring subscription to buy, and the CD never expires.
- All of this make it easy to review a process quickly, when you need it.

Some of the latest training CDs available from Tips For Manufacturing are focused on Mastercam's High Speed Toolpaths (HST). The 3D HST Surface Machining CD has been updated for X5. And the new 2D High Speed Toolpath CD is receiving great reviews from customers because of its comprehensive training on Mastercam's Dynamic toolpaths. Both first time users and long time users will find this CD very informative. Each CD contains approximately 55 lessons, using eight different parts, in almost five hours of video.

**Tips For Manufacturing** **Mastercam**  
Video Training

"Why Read The Book When You Can See The Movie"

- Complete "Step By Step" training
- No Subscription / No Expiration
- Use it on any PC / No Installation
- Great for new, or, experienced users
- Learn at your own pace, on your time
- Great for reviewing a single process
- Includes DXF, DWG & Solid Models

Postprocessors & On-Site Training Available.  
Now Located In Mesa, Arizona

[www.TipsForCadCam.com](http://www.TipsForCadCam.com) or...  
[www.TipsForManufacturing.com](http://www.TipsForManufacturing.com)  
Questions? Call Us at... (480)-986-9528

Tips For Manufacturing's philosophy is simple... "Why Read The Book, When You Can See The Movie". The latest enhancements in X5 are intended to reduce programming time. But this only happens after you've learned how the new features work. Reading a book can be time-consuming and always leaves some things subject to interpretation. This is why most people prefer to be shown how it works. The Tips For Manufacturing videos offer the most expedient solution to your training needs.

For more information on Tips For Manufacturing, or to see some of their sample videos, visit the web site at [www.TipsForManufacturing.com](http://www.TipsForManufacturing.com). ☒

## Streamingteacher Release 6 for Mastercam

Streamingteacher has released Mastercam training for Version X5. Being first in the online training market for Mastercam has provided time to make many discoveries about how people learn. Most individuals who use Mastercam have considerable pressure on them to be productive while migrating between versions. So they need a way to learn as they go, in a user-friendly, on-demand format. Providing this is Streamingteacher's #1 goal.

### Database-driven

Searching is a key feature of Streamingteacher. Our system is database-driven, and it allows you to search for specific lessons or topic segments. When the pressure is on, finding the solution quickly becomes paramount. You are able to keep track of what training segments have already been completed. Those segments can be re-opened and studied again any time during your subscription.

### Sequencing

At each update, Streamingteacher evaluates the sequence for training because as the Mastercam software becomes more advanced, the workflow changes. For example, if a user learns V9 methodologies and attempts to apply those in X5, they would not be using the software to its fullest potential, so the order and content of training is structured based on the Mastercam version.

### Video QC (Quality Control)

While Streamingteacher prepares courseware for each subsequent release, we QC existing videos, sample files, and downloads to make sure that they are accurate relative to the new release, modify those that are necessary, and add new content.

### Hybrid Training (On-Line and Classroom)

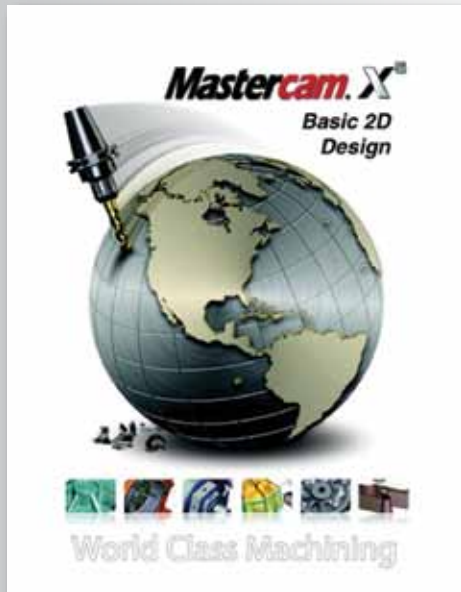
Streamingteacher is designed with hybrid training in mind. Our training, in combination with your Mastercam Reseller's training programs, has proven to be highly effective. Access to Streamingteacher before, during, and after will fortify what is learned in the class. If your instructor is familiar with Streamingteacher, they can provide you with specific segments to focus on after the class.

For more information, visit [www.streamingteacher.com](http://www.streamingteacher.com), e-mail [sales@streamingteacher.com](mailto:sales@streamingteacher.com), or call 503-653-5260. ☒





# OTHER NEWS



## Getting Started Tutorials Updated for X5

The *Getting Started* Tutorial Series is designed to get a new user started with Mastercam X5. The series currently includes seven books that introduce general Mastercam functions and teach basic skills. Each tutorial is packaged in a ZIP file that contains the tutorial book in PDF format, together with Mastercam parts and other accessory files, such as sample posts.

- *Welcome to Mastercam*
- *Files and Mastercam*
- *Working in the Graphics Window*
- *Basic 2D Design*
- *Basic 2D Machining*
- *Basic 3D Design*
- *Basic 3D Machining*

The tutorials are best used in sequence, but they can also be used independently. Please contact your Mastercam Reseller for more information. ☒

## New Customer Newsletter Format for 2011

With the New Year, the Mastercam Notes newsletter will be changing its format and delivery.

We are planning to deliver a more in-depth and informative printed newsletter twice a year, typically right around a major release and then again six months later. We will also release an e-mail newsletter at least once a year, focusing on Mastercam's third party products and partners, and other Mastercam materials.

In an effort to make sure you get all publications, we are in the process of setting up a system to collect your e-mail addresses, to ensure you don't miss any Mastercam news. More information on this will come out in the next issue of Mastercam Notes.

What's more, we will constantly post Mastercam information on our blog (<http://blog.mastercam.com/>), on Twitter (<http://twitter.com/mastercam>), and on Facebook (<http://www.facebook.com/Mastercam>), so be sure to bookmark all these links for future reference. ☒



# EDUCATIONAL NEWS



## Breathitt High School Success Article

When computer-aided drafting instructor Ken Combs wanted to ramp up his program to include a complete manufacturing lab, he turned to Techno and Mastercam for help. Due to his limited experience using CAM software and routers, he relied on support from his suppliers.

Techno provided Combs with two days of training on Mastercam and the controller. CNC Software provided numerous hours of support via phone and email. With the comforting knowledge that support from his new suppliers was only a phone call away, Combs was ready to launch his program.

Soon, Combs and his students were getting into high gear with class work and starting to make projects that could be sold to offset the cost of future projects. Combs and his students are all learning together, creating a true team atmosphere in their "Bobcat Workshop."



Teachers—Get the Training You Need!

Visit [www.mastercam.com/STTSurvey](http://www.mastercam.com/STTSurvey) and tell us what you're looking for in training. Respond by February 7 to be entered in a drawing to win a Mastercam jacket!

To read the full article on Combs and his program, please visit [www.Mastercam.com/TeachersStudents/InstructorsCorner/Articles](http://www.Mastercam.com/TeachersStudents/InstructorsCorner/Articles). ☒

## Students Experience Instant Success with Mastercam QuickPart

Updated for the X platform, Mastercam QuickPart delivers a motivational step-by-step, screen-by-screen approach that guides students through the CAD/CAM/CNC process. With students designing and cutting parts the first day, QuickPart is an excellent way to introduce students to machining and capture their interest.

Instructors are able to configure the software to match the abilities of their students and the available tools and materials. Then, students can combine graphics available in the vast art library with text in numerous fonts to create appealing and useful projects such as jewelry, signs, key chains, and more.

Once the students are finished designing their parts and adding the appropriate toolpaths, they can print a detailed setup sheet for each project. The setup sheets show toolpath, stock size, location, touch off position, selected tool, and more. Instructors can easily review this information before running the CNC machine.

Think QuickPart may be just what you need to get students started or to draw students into your program? Contact your local Educational Mastercam Reseller today! ☒

**Get Integrated PSE!**

Post-processing  
Simulation  
\*Emulation

\*using G-Code

Supports Multi-Axis  
Mill & Mill / Turns!

**ICAM**

icam.com/EASY-QUOTE JIMTOF 2010-E5031

**Mastercam**

**X TRAINING SOLUTIONS**

**In-House Solutions**  
800-529-5517

Available through your Authorized Mastercam Reseller  
[www.emastercam.com](http://www.emastercam.com) • [books@emastercam.com](mailto:books@emastercam.com)



# OTHER NEWS

## Jet Propulsion Laboratory Customer Profile



Richard Cournoyer, manager at Jet Propulsion Laboratory (JPL), puts today's head count at 30 machinists and two apprentices (with openings for three more). The next major flagship mission will involve sending a landing vehicle to Europa, a moon that orbits Jupiter. That launch is scheduled for 2020. Between 2014 and 2015, he anticipates the project will be in Phase C—

the build phase—and the machinist staff is likely to double.

When JPL rebuilds its machinist staff, Cournoyer will meticulously sift through the resumes of applicants supplied by three contract employment firms. In the past, finding enough qualified applicants was a problem. Most applicants were immediately rejected because they had no experience with Unigraphics, JPL's primary CAD/CAM software, or the other two CAM software programs for which the lab has licenses. Something had to be done.

### Challenge:

The cyclical nature of JPL's contracts sometimes makes it necessary to add 20-30 qualified programmer/machinists on very short notice.

### Solution:

Mastercam

### Benefits:

- Mastercam is the industry-standard software and has a large base of experienced users.
- Mastercam is easy to learn and use.
- Apprentice programmer/machinists study Mastercam at a nearby community college.
- A sophisticated 5-axis post processor is delivered as part of the Maintenance program.

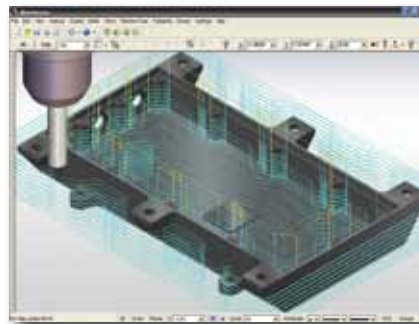
### The Story:

A challenge for JPL's machinists is working with exotic materials chosen for low coefficient of thermal expansion. Space is the ultimate vacuum and does not provide an atmosphere that will promote radiation of heat. Components tend to bake inside the spacecraft while cruising to a destination millions of miles away. These instruments are required to perform their tasks in space at temperatures down to -455°F. So they have to deal with unique machining requirements of materials like Kovar® and Zerodur®.

What does this have to do with staffing? Cournoyer found out the hard way when he was trying to quickly build up staff levels during the latter build stage of the MSL (Mars Space Laboratory), now called Curiosity mission, which will launch in 2011. He said, "It takes somebody with finesse for precision making delicate instruments to be able to work here. Our machinists are programmer/machinists. When they sign on for a given part, they own it from the sketch to launch pad. They decide which manufacturing processes to use

and write their own CNC manufacturing programs.

"All the resumes I saw from people with Unigraphics skills were looking for positions as design engineers. They only wanted to work behind a desk. I was getting pressured by upper



management to grow my staff faster but I just couldn't find machinists proficient in the CAM software we were using. We had a good variety of software but it was not industry-standard software."

What he saw on just about every machinist's resume coming across his desk was Mastercam



experience. Cournoyer was experienced with this CAD/CAM software, and knew that it could easily provide the tools for precision CNC manufacturing that JPL required. Mastercam CAD/CAM programming was also being taught in the Manufacturing Technology program at nearby Glendale Community College (GCC), which supplies JPL with some of its apprentices. So apprentices would have the benefit of using the same software that they were using in their courses.

Cournoyer made a formal budgetary request to acquire seats of the industry-standard Mastercam software, and soon his staffing problems were behind him. Instead of automatically rejecting three-quarters of the machinists' resumes, he now had three times as many candidates to evaluate for each position.

Shortly after JPL obtained several Mastercam licenses, Cournoyer went back to school to learn the latest version of Mastercam, which was X2. Based on the training, he is now an SME (System Material Expert) for apprentices, who receive 4,500 hours of training at JPL, and for contract employees.

Shortly thereafter, the group hired another SME, a "true Mastercam expert." Cliff Lengtat is an experienced machinist who worked as a certified Mastercam instructor before accepting a coveted contractor's position at JPL.

After acquiring its Mastercam seats, JPL was able to build its contract programming staff to a high of 23 and handle the final manufacturing push for the upcoming MSL Mission.

Much of the work is done using a paperless workflow. A major benefit of Mastercam is a proprietary kernel that allows the programmer/machinists to import Parasolid models directly without the loss of data. Then Mastercam's ease of

use and ultra-fine control of toolpaths make a significant contribution to the machinist/programmer's productivity in manufacturing technically complex parts.

Perhaps the most impressive immediate contribution Mastercam made at JPL was in the realm of post-processing. Cournoyer said that each of the CAM systems used at JPL had posts that worked at least adequately with 30 different CNC machines. However, none had a post that could be used in conjunction with a huge, multi-million dollar Mitsubishi 8-axis boring mill. As a result, that machine was woefully underutilized, and often called the "big drill press."

JPL asked each of the CAM vendors for its thoughts on getting a post to work with that machine. Two provided quotes, but the Mastercam Reseller went to work immediately writing a postprocessor. "It took a while for them to do it," Cournoyer said.

"It was not a trivial assignment, because it's truly a one-of-a-kind machine. However, within a year we had a working post, and had turned an idle machine into a valuable multiaxis CNC asset. In an emergency, we can and have performed a 22-hour machining assignment on that machine in 24 hours."

☒



## Season's Greetings from CNC Software



Everyone at CNC Software would like to wish you the best of luck in 2011!

**CNC Software, Inc.**  
**Corporate Headquarters**  
671 Old Post Road  
Tolland, CT 06084 USA  
(860) 875-5006  
mcinfo@mastercam.com  
www.mastercam.com

**Educational Division**  
5717 Wollochet Dr. NW  
Gig Harbor, WA 98335  
(253) 858-6677  
info@mastercamedu.com

*Mastercam is a registered trademark of CNC Software, Inc. Mastercam Certification is a trademark of CNC Software © Copyright 1984-2011. All rights reserved. SolidWorks is a registered trademark of DS SolidWorks Corporation. Other trademarks are property of their respective owners.*

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