



First Quarter 2009

Mastercam[®] Notes

Customer Newsletter

Dear Mastercam Community,

I would like to take this opportunity to give you an update on some of the major milestones and new endeavors CNC Software has accomplished this past year. As you know, we are all facing a rather tumultuous economy and uncertain financial times. Despite the ever present bear market, our 25th year of business came to a close as our strongest year yet, thanks to you. Our latest release, Mastercam X3, enjoyed a positive response and delivered features such as new 2D High Speed Toolpaths, Feature Based Machining, and more surface machining and five-axis improvements. Upcoming X4 improvements and enhancements include additional toolpath and CAD features and a new product called Mastercam[®] for SolidWorks[®], a collaboration that should satisfy a certain segment of our CAM market.

Our green business improvements include a new shop and training room, completed in 2008, that uses geothermal heating and cooling like the rest of the building. A total of 48 wells extract enough heat from the ground to keep the building warm on the coldest New England day. Since the only energy source the building uses is electricity, we took an additional step toward energy independence and installed a 73kW ground-mounted solar array last August. The panels generate enough energy to meet a third of our electrical, heating, and cooling demands. To date, we have produced over 36,000 kWh of energy. Put into terms we can relate to, this is about the amount of energy it would take to power 750 homes for a day.

Our focus has not changed since 1983 when we released our first CAM product on an Apple 2E ... "Understand the needs of our users and meet their needs." Because your needs have changed dramatically in the last 25 years, so has the Mastercam product. We plan to continue meeting your needs with a high level of awareness and support of green business practices.

Thank you for remaining Mastercam users and keeping us all employed. Your support and longevity as customers are much appreciated.

Sincere thanks,
Mark Summers
President, CNC Software, Inc.

Product Highlights



Mill Feature 2-3
Mastercam X4 Sneak Preview



Lathe Feature 4
Back to Basics: Mill/Turn
Operations



Wire Feature 5
Increasing Efficiency in Your
Wire Programming

MILL

Mastercam X4 Sneak Peek: Multi-Threading in Mastercam X4

The news:

Development work has been underway to take advantage of the newest computer hardware technology. Systems with multiple processors/cores are becoming common, and the trend toward using additional processors and cores will continue. CNC Software will roll out Mastercam support for multiple cores and processors starting with High Speed multi-surface Toolpaths (HST) and multiaxis toolpaths during the X4 product life cycle.

How it works:

Mastercam X4 includes an option that allows a user to turn the multi-threading capability on or off. With multi-threading on, a toolpath will be processed in a multi-threaded mode if the toolpath type is supported. Toolpath types not supported by multi-threading will be processed as they would normally. The multi-threading manager directs the supported toolpaths to their own processor/core for processing or regeneration in separate threads as processors become available. During this processing, Mastercam is free to continue working on other tasks. The user has control over the number of processors to use and the priority of specific operations being processed. The user can also pause and cancel processing of any particular operation.

Process-Name	State	Progress	Status
3 - Surface Hi...	Processing	<div style="width: 70%; background-color: green;"></div>	HMM: Calculating consta...
4 - Surface Hi...	Pre-translation	<div style="width: 100%; background-color: blue;"></div>	Waiting to translate...
5 - Surface Hi...	Processing	<div style="width: 60%; background-color: green;"></div>	HMM: Calculating pencil...

Benefits to the user:

The new multi-threading capability delivers benefits in two ways. First, users can continue working in Mastercam as the multi-threading manager is processing. Without multi-threading, users have had to run multiple Mastercam sessions so that other work could continue while waiting for Mastercam to process a toolpath. By targeting the surface high speed and multiaxis toolpaths first, we are able to provide the greatest possible benefit. A second benefit provided by this technology is a dramatic decrease in toolpath processing time. Users can see the benefits themselves by trying the multi-threading capability and viewing the time savings recorded in the Mastercam Event Log (running in the system tray during a Mastercam session).

X4 Mill Sneak Peek

Dynamic Pocket –

A new pocketing toolpath that adapts to the pocket shape for increased tool life and efficiency.

Updated Interface –

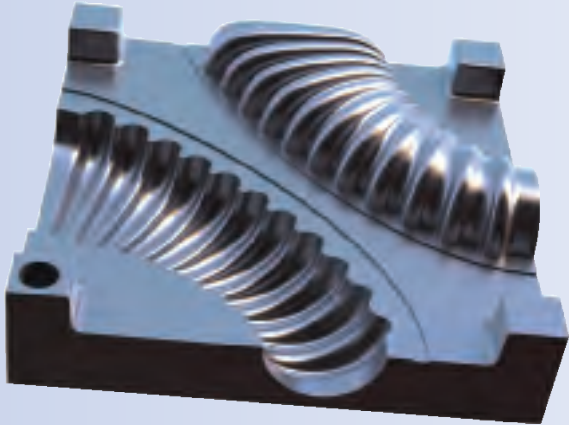
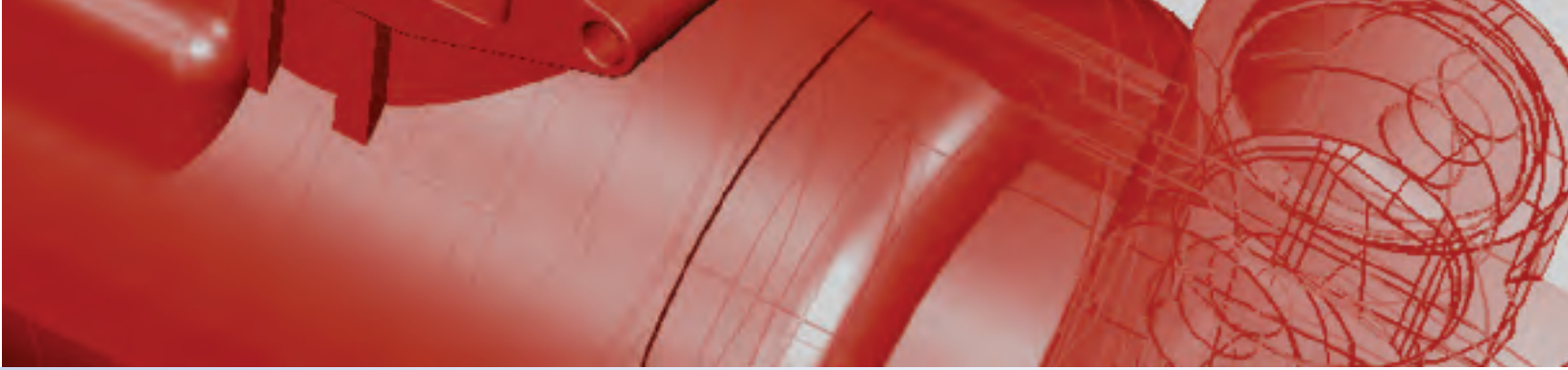
2D toolpaths get a face lift with a new tree style structure. Hidden options are more easily found, allowing users to take advantage of all features available.

Multi-threading –

Support for high speed toolpaths and multiaxis toolpaths.

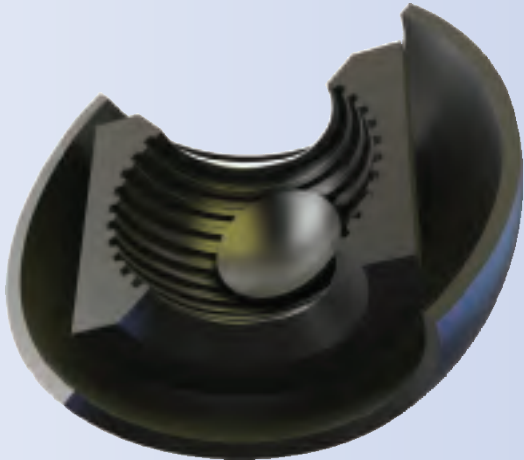
Refine Toolpath

Tools – New options in surface machining give greater control and big improvements in surface finish.



File
 Flow 5ax part (83 comp surfaces, 1 drive, 1 check, toolpath tolerance 0.001in.)

<i>Non Multi-Threaded Processing Time</i>	<i>Multi-Threaded Processing Time</i>	<i>Percent Change</i>
4 min. 57 sec.	2 min. 1 sec.	59% reduction



File
 Scallop mold (5 HST toolpaths, 21 MB NCI file, toolpath tolerance 0.03mm)

<i>Non Multi-Threaded Processing Time</i>	<i>Multi-Threaded Processing Time</i>	<i>Percent Change</i>
4 min. 30 sec.	2 min. 31 sec.	44% reduction

Test Machine: Dell® Precision™ T5400, Quad-Core, 3GB RAM, Windows® XP Professional. Throughout the X4 Beta cycle and into the final release, additional multi-axis toolpaths will support multi-threading.

If you would like to test these new capabilities now, you can join the beta testing program by sending a request to prototypes@mastercam.com. ☒

LATHE

Back to Basics: Mill/Turn Operations

Back to Basics is an occasional series exploring some of Mastercam's core functions. We hope you find this useful!

Mastercam X3 Lathe includes two main techniques for creating milling toolpaths on your lathe machine.

For the most common contour and drill operations, you can use the Lathe C-axis toolpaths that are already included with Mastercam Lathe. When you select one of these toolpaths, Mastercam will automatically set the Tplane and Cplane to the appropriate settings for the desired application.

These toolpaths also support Y-axis rotation and axis substitution. To do this, go to the **Toolpaths** menu, choose **C-axis**, and select your desired toolpath. Just keep in the mind that the active machine definition must be a lathe or mill/turn machine that supports the appropriate axes.

Another technique can be used if your workstation is licensed for both Mastercam Mill and Lathe. This will allow you to access any mill toolpath while a lathe or mill/turn machine is active.

Use mill toolpaths to machine in planes or orientations not available with the standard Lathe C-axis toolpaths. You can also use this technique to create toolpaths other than contour or drill toolpaths (for example, pocket toolpaths). You can set the Tplane to any view available in your part, or use the Cview utility as a guide. To do this, go to the **Toolpaths** menu, choose **Mill**, and select your desired toolpath.

To use either type of toolpath, your lathe must be equipped for live tooling, and your Mastercam machine definition must be properly configured.

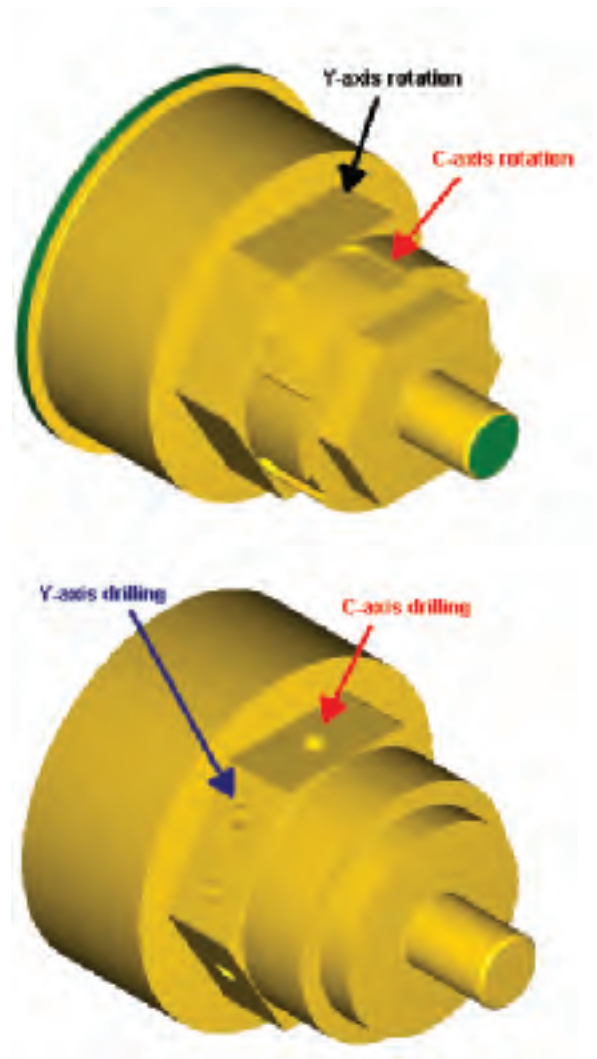
Note: Tool collision, stock, and other boundaries are not used for milling toolpaths on mill/turn parts. However, these parts often begin with turning toolpaths in order to remove the majority of the material. Since the turning toolpaths can use tool boundaries, you should set them even if you will be creating milling toolpaths later on. ☒

X4 Lathe Sneak Peek

New multi-groove selection.

Separate feeds and speeds for rough and finish grooving.

Post-based tool inspection.



WIRE

Increasing Efficiency in Your Wire Programming

Mastercam Wire provides a great way for your shop to increase its efficiency – subprograms. We will give you a brief description of subprograms, and more information can be found in Mastercam’s Help system.

Mastercam Wire provides the capability to include subprograms in an operation. A subprogram is an NC program that is called from the main NC program. It is used to repeat code (XY moves) in an operation, and creates a shorter NC program.

Both contour and 4-Axis wirepaths can use subprograms. In 4-Axis wirepaths, subprograms are used to repeat passes that are the same. For contour wirepaths, subprograms are used to repeat passes that are the same, and also to create copies of the same part. All contours use the same subprogram. You can use subprograms on a multi-contour operation where the contours are not identical, and Mastercam Wire gives each contour its own subprogram number.

The **All contours use same subprogram** option in the Subprograms page replicates the wirepath information from the first contour to create each consecutive contour in the file. Choosing the **All contours use same subprogram** check box causes Mastercam Wire to use the same subprogram on all chained contours in the operation.

The following restrictions apply:

- You cannot use **All contours use same subprogram** on non-identical contours.
- When using subprograms on identical contours, the contours must be oriented the same way and have the same relative distance from the thread position to the start of contour.
- You must individually chain each contour and start/thread/cut position that you want to include in the subprogram.

The NCI file contains all of the parameter information for each contour. However, the amount of information that is contained in the NC file is minimized by creating and using a subprogram to machine the same contour.

Note: When you backplot a file that contains subprograms, each contour is backplotted and displayed in the graphics window. ☒

X4 Wire Sneak Peek

Custom support for Agie EDM with Agievision controls. The Agie Work Flow (AWF) in Mastercam X4 consists of wirepaths specifically designed for the Agievision. These Agie-specific wirepaths (Contour, Collar, No Core (Pocketing and 4-Axis)) have a completely new user interface that anyone familiar with Agievision will instantly recognize.





ROUTER

A Router Hobby Turns Into a Real Business

What would you do with a 4 ft. x 8 ft. CNC router in your garage? Make furniture? Make signs? Make ANYTHING? Keith Butzgy does just that. A few years ago Keith purchased a CNC router to do some woodworking projects.

X4 Router Sneak Peek

Automatic nesting now offers the ability to edit nested sheets using drag and drop.

Vertical and/or horizontal cut-off operations can now be automatically generated in toolpath nesting.

“The hardest part was convincing my wife. At the time, we owned a house in New Jersey with a one-car garage, so getting a router meant losing our valuable garage space to a noisy and dust-creating machine. Thankfully, she agreed, but I certainly had to ‘sell’ the idea to her.”

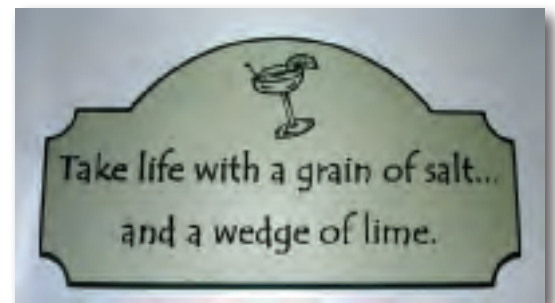
Keith uses the machine for various jobs. Sometimes the machine is running every evening for days straight. Other times, it is sitting idle for weeks at a time. Originally, he decided to purchase a machine to cut lithopanes using Mastercam Art. A lithopane is a carved picture (usually cut into an opaque white material like acrylic or Corian®) that when backlit becomes a beautiful three-dimensional picture.

“Since lithopanes are relatively small, I was originally looking for a small table-top router. During my research, I realized that I could be much more versatile with a larger machine. This slowly grew into the 4 ft. x 8 ft. router.”

In addition to lithopanes, which make great wedding gifts, recent projects include an oak poker table, house signs, commercial signs, a 4-foot dinosaur used at a tradeshow in the Mastercam booth, and lawn ornaments. Recently, Keith and his wife Karen developed a product line of signs and wooden letters used to decorate a home. They signed up for a couple of craft shows in their area to see how they’d sell.

“The signs are great for living rooms, kitchens, and bathrooms, and the letters are great for kids rooms. We set up a booth in my hometown during their yearly festival and just waited to see what came of it. I was anticipating the worst, but slowly people would

come into the booth and just stare at my signs, reading them all. They’d come across a funny saying and laugh, or just comment on the nice sayings.”



Not all of this is fun and games. Since starting his router hobby, Keith became a full time employee at CNC Software. Often his travel requirements limit what he can do while at home. Four years ago when he started this CNC hobby, he had a different job and no children. Today, he works at CNC Software and has a family at home.



"My priorities have certainly changed. I don't get out into the garage much, and when I do, I have to be more efficient with what I do with the time that I have. The biggest benefit I have realized is the knowledge that I have gained from using the machine with Mastercam. I now have real experiences to share with customers and prospects, and with our development team at CNC Software."

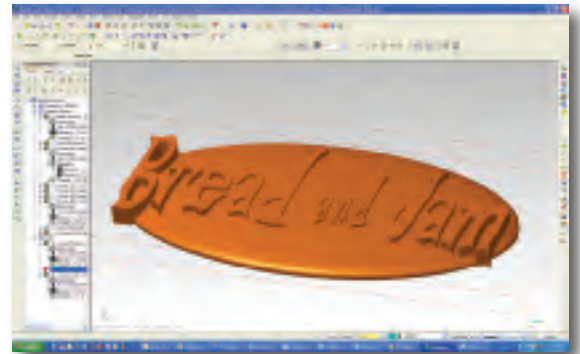
One commercial sign project had some unique criteria. Keith's high school friend, Sal Prizio, quit his job in Manhattan and moved to Cohoes, New York to open a coffee shop with his wife. Since the coffee shop also hosts live bands, Sal used a play on words and named the shop "Bread and Jam." Sal asked Keith to make the sign and had some design ideas that he wanted to implement.

After weeks of discussion, the two ironed out a design that was very creative, and also extremely challenging. It was a double-sided sign to be hung over the entrance to the coffee shop. The sign was mostly dome shaped with the text protruding out from it. To account for the weather in upstate New York, they decided to use Sign•Foam, which doesn't react greatly to extreme temperatures. Insects and woodpeckers don't like the material. Sal was very happy with the results. "It was so great to be able to envision the sign and see it come to life through Keith's work."

Keith tracked his progress on this job and filmed the machining process. If you would like to see the video, please visit <http://tinyurl.com/dfvchk>.

Keith's future plans are nothing short of aggressive. He wants to build a line of poker tables, dinosaur lawn ornaments, and toys for his daughter, Abby. With patience and support from his family, Keith hopes to continue to balance both work and play.

"I work with Mastercam at the office to help our dealers and our customers, and I work with Mastercam at home as a customer. It's the best of both worlds." ☒



Mastercam
Books Videos
eLearning
1-877-873-6867
www.camInstructor.com

"CamInstructor books and videos are the best resources we have had to date. Not only are our programmers learning more about Mastercam, they are taking less time to learn it."

Tavi Udrea, Training Manager
Haas Automation

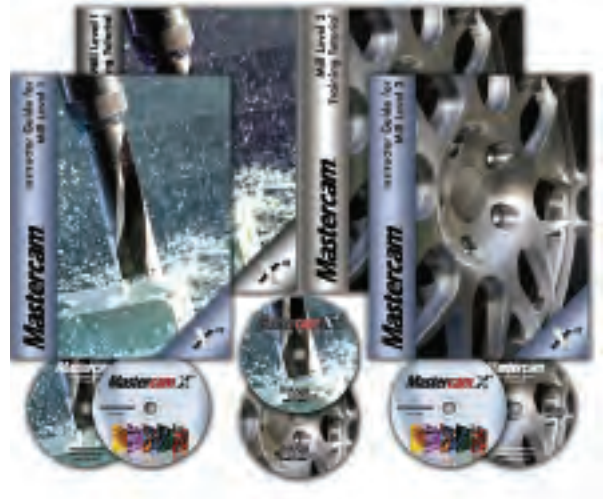
CamInstructor Inc. Mastercam Partner

ADD-ONS

In-House Solutions Releases Instructor Kit, Training Videos, and More

Instructor Kit

The Mastercam X3 Instructor Kit is designed to complement the Mill Level 1 and Mill Level 3 Training Tutorials, providing an enhanced set of teaching tools that are ideal for college instructors. The kit consists of 39 lesson plans outlining over 65 hours of content, 36 PowerPoint® presentations, numerous MCX files including over 50 training exercises, quizzes for every chapter, and four hands-on practical tests. For further practice, testing, or homework assignments, 25 dimensioned parts are included, along with Mastercam Demo software.



X3 Training Videos

Training video CDs are now included with Mill Level 1 and Mill Level 3 Training Tutorials. The Mill Level 1 and Level 3 CDs offer 19 and 26 training exercises, respectively. These exercises familiarize the user with basic Mastercam concepts such as creating geometry, working with levels, and creating/modifying surfaces. Each CD also walks the viewer through creating both geometry and toolpaths for many of the tutorials in the book.

Note: Multimedia content is also available for V9 to X2 milling applications.

For more information on these and other Mastercam Training Solutions, visit www.inhousesolutions.com, call 800-529-5517, or contact your local Mastercam Reseller. ☒



X4 Design Sneak Peek

Improved capabilities in the level manager.

New dynamic Xform and planes functionality.

Viewsheet improvements.

Xform 3D revisions for usability.

Wireframe Feature recognition in chaining.

Enhanced trimming interaction.

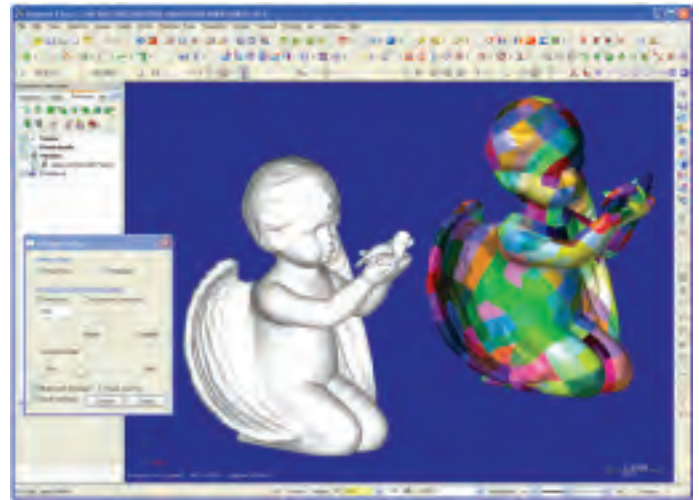
From Point Clouds to Solids, Verisurf AutoSurface is a Fast Reverse Engineering Solution

Many Mastercam customers have been using the reverse engineering tools from Verisurf Software for high-volume point cloud acquisition and management. These tools can reverse-design into a mesh, surface, or solid model right in Mastercam, and can be directly used for manufacturing or inspection purposes. Now, with the addition of AutoSurface, you have the ability to automatically create even complex surfaces from point clouds. AutoSurface takes a mesh and creates surfaces and boundary curves using a few, simple user inputs. The result is a collection of surfaces, the number of which automatically adjusts according to the complexity of the mesh and the desired level of surface detail. No other solution makes surfacing of complex geometries more simple or faster.

Verisurf AutoSurface is 100% automatic. No curve network is required. Import points, point clouds, or STL meshes, press one button, and get the model. All shapes, primitives, freeforms, and open/closed volumes are supported. "Using Verisurf, we can go from laser scans to accurate CAD models in a fraction of the time any other software could," says Andy Solis, Production Manager for Tri-Tech Precision, Inc. "It's fast, easy to use, and accurate."

Verisurf software tools, integrated with Mastercam, allow users to accomplish the entire product cycle of scanning, meshing, auto-surfacing, solids generation, machining, and inspection—all within a single application, using a single file.

Originally developed to inspect complex aerospace parts to the CAD model, Verisurf is now used for reverse engineering, model-based tool building, and inspection. For more information on AutoSurface or other tools developed by Verisurf Software, please contact your local Mastercam Reseller or visit www.verisurf.com/mastercam. ☒



**THE POWER OF MASTERCAM,
TURBOCHARGED BY VERISURF**

Mastercam
ADD-ONS
REVERSE

Verisurf Reverse is an advanced, powerful software tool used for processing 3D scan data to produce usable manufacturing models. It brings truly state-of-the-art technology to the engineering process of precisely measuring a physical part or artifact to produce a digital 3D model.



CONTACT YOUR LOCAL MASTERCAM RESELLER FOR PRICING AND DETAILS

VERISURF

**THE POWER OF MASTERCAM,
TURBOCHARGED BY VERISURF**

Mastercam
ADD-ONS
UG READ

With Verisurf UG Read, Mastercam users can open native Unigraphics/NX part/assembly files without a Unigraphics or NX license. You also gain the unique ability to import Product Manufacturing Information (PMI) data, such as properties, notes, dimensions, geometric tolerances, datums and datum targets.



CONTACT YOUR LOCAL MASTERCAM RESELLER FOR PRICING AND DETAILS

VERISURF



EDUCATIONAL NEWS

Cosmetologist Turns Machinist and Loves It



Back in the 80's, Debra Mann was running her own cosmetology business but was still having a difficult time making ends meet. Seeking a better way to support herself and her two sons, she took some career planning classes. It was then that Debra started considering a career as a machinist.

"When I found out what machinists actually did, I was amazed," she said. "Making things from metal is just as much of an art as working with hair and nails, and the benefits and job stability are so much better."

After earning an Associates Degree in machining at Front Range Community College, she spent the next seven years working in a mold shop. While Debra took a few bumps for having the audacity to venture into the then traditional male territory, she enjoyed her time there.

"Being a female in a man's world wasn't easy. Some older guys didn't believe that women should be in their territory. So they would make sarcastic

remarks. On the other hand, this industry allowed me to work in my own space and at my own pace, and I really enjoyed that."

Returning to school, Debra obtained an advanced degree in Manufacturing Technology Education from the University of Colorado. Soon after graduation she began teaching at Front Range Community College where she has been for thirteen years and is now the director of the Machining Technology Program.

Today, when Debra stands in front of her machining classes, she is rarely the only woman in the room. Usually, there will be at least one or two young women in the group. She believes that not only is machining a good career for them but that women, in turn, have something special to offer the industry.

"Women pay more attention to directions. They are more open-minded. We have some girls in our program who have done a great job. We have a girl who went through the program and then went to work for her dad. We had a girl last year that is taking over her grandfather's business. It is just a matter of dedication, of sticking with it." ☒

Streamingteacher
COURSEWARE and CERTIFICATION
UPDATED for **Mastercam X3**

Comprehensive videos, tutorials, samples and training.

- Secure On-Line Purchasing
- Subscribe today - Start training today
- Train anywhere you have internet access
- Certified Instructor Support
- Grading, Quizzes and Testing
- Distance Learning Program
- Earn 3 College Credits
- NO WAIT TO START!

Streamingteacher is eco-friendly
www.streamingteacher.com

Moldplus CATIA Translator

Import CATIA V5 data
CATPART and CATProduct files
MBO data (IGES Isovancing data)
List part features tree data

Import and Export CATIA V4 data.
MBO/import or add to be the CATIA V4 files
MBO/export or add to be the CATIA V4 files

Solutions that make a difference!
www.moldplus.com
info@moldplus.com



Cape Fear Community College Shows Off Their Mastercam Machine Technology Program with Video

In 2005, Cape Fear Community College in Wilmington, North Carolina integrated computer-controlled machining and Mastercam into their machine technology program, giving it new life. Their success was outlined in an article in Techniques Magazine's February 2006 issue.

More than three years later, Cape Fear's program is still going strong. Film and video students from the college created a promotional video for the Machine Technology program. It was shot entirely in their facilities and demonstrates how Mastercam plays a critical part in their curriculum. Watch the video at www.vimeo.com/2308573.

Read the full article at www.mastercam.com/edarticles. 



What is Mastercam Certification™?

The Mastercam Certification program is a rigorous set of practical tests that recognizes a programmer's knowledge and ability to work effectively with Mastercam software, overcome common issues facing today's shops, and produce high quality finished parts. It sets the bar to a high standard and ensures that only the very best programmers become Mastercam Certified.


Benefits of Mastercam Certification

Benefits to the end user – Programmers and machinists completing the Mastercam Certification program carry with them a useful means of quickly proving and validating their Mastercam skills in an increasingly demanding industry.

Benefits to the employer – By hiring Mastercam-certified employees, employers are guaranteed to get the level of experience and expertise that they demand.

Employers can hire with confidence in knowing that their new employee has the skills that it takes to get the job done right.

Find Out More about Mastercam Certification

If you have questions about Mastercam Certification and would like to know more about it, please feel free to contact us at training@mastercam.com. 



"Using Mastercam has helped me do very complex machining that I used to only dream of doing. I currently program, set up, and run a 5-axis milling machine to build complex prototype parts for the defense industry. I had my skills formally documented through the Mastercam Certification program to demonstrate to my company and to our customers the level of proficiency that I have achieved."

- Jackie D. Edwards
Toolmaker
Kansas City, Missouri



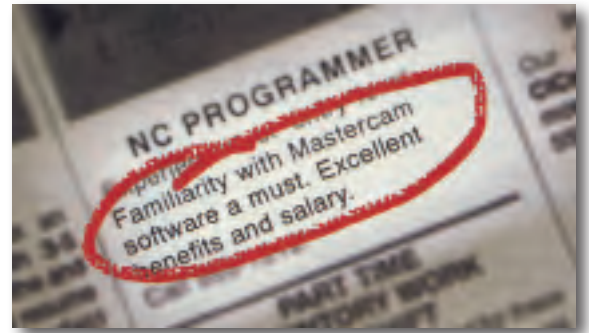
OTHER NEWS

Want to Help Your Fellow NC Programmers?

Do you know an NC programmer who is looking for work or a shop that is looking for an NC programmer? CNC Software can help. Head over to the **Employment** section of www.Mastercam.com. You'll find it on the right side of our home page, under the **CAM ZONE** sidebar.

The **Employment** page lists open NC programming jobs from shops across the country as well as jobs at CNC Software. CNC Software offers many opportunities, and we even have our own on-site machine shop. If you know any NC programmers looking for work, this is a great place to point them.

If your shop has an opening, the **Employment** section is also an ideal resource. Just click the link under **Free Listings for Your Shop** and fill out the form. We'll post the job opening to the Mastercam community. ☒



Mastercam
inhouseposts.com

5-Axis
Router
Mill-turn
Machine Simulation
Custom Machines & Controls

postdev@inhousesolutions.com

ProDrill
Advanced feature based drilling for Mastercam
Runs on surface and Solid models

ProDrill Version 3 with:

- 1. optimal tool management including easy firm/custom tool size
- 2. color controlled feature settings for tapping and multiple reaming sizes
- 3. more machining cycle operations 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 grinding speed!

Rapid Return on investment!

ProDrill is a product of Tecnomatix S.L. and Mikplus S.A.

Solutions that make a difference!

www.mikplus.com
info@mikplus.com



Wanted: Customers to Participate in Mastercam Focus Group Program

Starting in 2008, during the Mastercam X4 development cycle, CNC Software initiated a new Focus Group Program. The goal of this new program is to get customer feedback early in the development process, field-test potential new enhancements, and allow for interface improvements and changes before final release.

So far, this new Focus Group Program has been very successful. It has resulted in several new areas of improvement and has led to new enhancements for the Mastercam X4 release, such as multi-thread processing, toolpath refinement, dynamic pocketing, customized setup sheets, bump-mode nesting, Agjevision custom user interface, and more. The Focus Group members were able to see their input addressed throughout the development cycle and had a direct impact on the development of new features.

Not many people are aware of the program because we kept the initial program small and manageable in order to refine processes and procedures for future releases. Like the Beta program, the Focus Group Program requires a commitment of time, and members must be willing to meet certain requirements:

- Commit to spending a specified amount of time based on the needs for a given product enhancement.
- Provide regular feedback and reporting to CNC Software.
- Participate in regularly scheduled webinars and feedback sessions.
- Agree to strict non-disclosure.
- Meet Mastercam Beta tester qualifications.

Although participating in Mastercam Focus Groups requires a substantial commitment, members receive enormous benefits. They have the ability to give and receive early feedback on emerging enhancements, help direct feature development, ensure that their requirements are being met in the final release, and influence enhancements and improvements in future releases.

Initial Focus Groups for each project are kept relatively small (typically no more than 10 to 15 users). Most candidates are recruited by our Authorized Resellers and coordinated through the CNC Software Product Management department.

We are currently looking for additional Focus Group candidates in a number of application areas. Those interested in participating should send an e-mail with their contact info to their Authorized Reseller or Distributor with a Cc: to prototypes@mastercam.com. The e-mail should include details on their area of application expertise, challenges they are facing, and areas of special interest.

As we get closer to the Focus Group Program release stage, we'll contact Resellers and candidates to verify interest and commitment. ☒





OTHER NEWS

New Multimedia Section on Mastercam.com

We recently added a new multimedia section to our Mastercam website. In this area you will find Product Demos, Customer Testimonials, Tutorial Videos, and Machining Videos.

- The **Product Demo** section features highlight videos of Mastercam's latest and greatest enhancements. Not only can you see the latest "What's New" videos for the most current release of the software, but you will also find video archives that spotlight many of the most popular features since the release of Mastercam X.
- Watch some Mastercam customers tell their own personal stories about how Mastercam has helped their business in the **Customer Testimonials** section.
- A variety of technical videos and information is available in the **Tutorial Videos** section.
- **Machining Videos** offers a library of our most popular YouTube videos. This is a great option to watch all the Mastercam videos for those who may have restricted access to YouTube.



This new multimedia section will be updated as we make new Mastercam videos. Please bookmark the URL and check back often for new content. ☒

www.Mastercam.com/Multimedia

Robotmaster[®]
CAD/CAM for Robots



Programming of Robots from CAD/CAM software with the same control as CNC machining

In-House Solutions
Mastercam X
robotmaster@inhousesolutions.com

Mastercam[®]
Tips For Manufacturing Video Training

- Complete "Step By Step" training,
- Great for new and experienced users.
- Learn at your own pace, your own time.
- Use it on any PC / No installation.
- No subscription / No internet access.
- Review infrequently used processes.
- The perfect refresher training course.
- Includes DXF, DWG & Solid Models.

www.TipsForCadCam.com or...
www.TipsForManufacturing.com
Questions? Call Us At (262)-789-7124



Upcoming Mastercam Events and Trade Shows

2009 marks yet another busy year for CNC Software, Inc. From March 30 through April 2, come see both Mastercam X4 and our latest product, Mastercam for SolidWorks, at the WESTEC show located at the Los Angeles Convention Center in California, booth #3258. On April 2 and 3, we will spotlight our latest advances for the woodworking industry at the Mid-Atlantic Industrial Woodworking Expo, in the Toyota Arena at the York Expo Center in York, Pennsylvania, booth #310.

Some additional corporate shows include:



March 26-28
ITEA
Louisville, KY
Booth #211

October 6 - 8
SOUTH-TEC
Charlotte, NC
Booth #6050

May 19 - 21
EASTEC
West Springfield, MA
Booth # 5245

November 3 - 6
SEMA Show
Las Vegas, NV
Booth #TBA

July 15 - 18
AWFS (Woodworking)
Las Vegas, NV
Booth #3313

December 3 - 4
MidWest Industrial
Woodworking Expo
Indianapolis, IN
Booth #408

September 1-7
World Skills
Calgary, Canada

December 10 - 12
PRI
Orlando, FL
Booth #4619

November 19-21
ACTE
Nashville, TN
Booth #441

For a comprehensive list of Mastercam's corporate shows, as well as events, seminars, user groups, and shows sponsored by our Authorized Resellers, please visit www.mastercam.com/events. ☒

Mastercam
for SolidWorks

Mastercam for SolidWorks brings the world's leading modeling software together with the world's most powerful and widely-used CAM software.

www.MCforSW.com
info@mastercam.com • 1-800-228-2877

Mastercam is a registered trademark of CNC Software, Inc. © Copyright 2009 All rights reserved. SolidWorks is a registered trademark of SolidWorks Corporation.

Sign Up Today for *Mastercam Shop Talk*

Are you always hungry for the latest tip on how to make the best use of a specific Mastercam function? Are you looking for a discussion on migrating from one release to the next? Are you interested in learning more about the new Event Logger that came with Mastercam X3? *Mastercam Shop Talk* is a great new way to keep on top of many of the technical issues that Mastercam users encounter on a daily basis.

This information-packed quarterly e-mail newsletter is available to Mastercam Maintenance customers only.

If you would like to see the first issue published in December 2008, please go to www.mastercam.com/shoptalk. The entry form has a few required fields, one of which is your customer number. You can get your customer number from the mailing address area on this page above your name and address, following the word "Customer Number". A second option is entering your Mastercam serial (also known as SIM) number. To get your serial number, simply start Mastercam and press ALT+V. Your serial number will appear in the upper right under your company's information.

Even if you're not a Maintenance customer, you can still register to receive important messages on updates and product news via e-mail.



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 1984-2009. 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