

Fall 2010

Mastercam® Notes

Customer Newsletter

Mastercam X5 Debuts!

Mastercam X5 is here with a new suite of programming tools focused on delivering speed and efficiency for your machining jobs.

In this newsletter, we'll introduce you to some of the main features and improvements in Mastercam X5, including new "smart" roughing and finishing toolpaths, multiaxis enhancements, new toolpaths, and much more. We are very excited about this release, and will be hosting LIVE rollout events again this year throughout the world so you can see it in person (see the back cover of this issue for more information on Mastercam LIVE).

In future newsletter issues, we will explore individual X5 functions in depth, showing you valuable tips and tricks, and demonstrating how they will help you on a day-to-day basis. We are sure you will benefit from what Mastercam X5 has to offer you and your shop.



The world's #1 CAM software machines...the world! Watch Mastercam cut a globe with complex mill/turn toolpaths. This is the same globe that is featured in Mastercam X5 marketing deliverables.
<http://tinyurl.com/Mastercamworld>

Product Highlights



Mill Feature 2-4
What's New in Mastercam X5 Mill?



Lathe Feature 5
New Toolpath and More in X5 Lathe



Wire Feature 6
New Power Library Format in Mastercam X5 Wire



Router Feature 7
Block Drilling, Automatic Toolpathing, and More



Design Feature 8
Many New Enhancements for Mastercam Design



Art Feature 9
New Features in Mastercam X5 Art

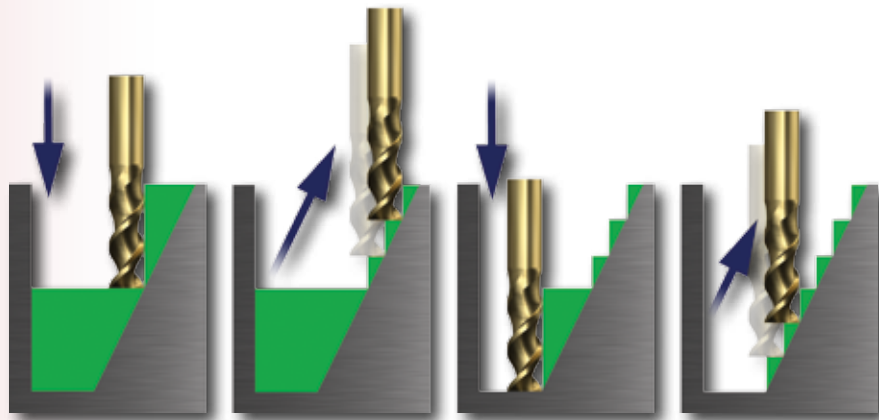
MILL

What's New in Mastercam X5 Mill?

Mastercam Mill users will get a huge boost in productivity with Mastercam X5. From new toolpaths and multiaxis improvements, to machine simulation and more, Mastercam X5 is focused on making your job easier and more efficient.

New OptiRough Toolpath:

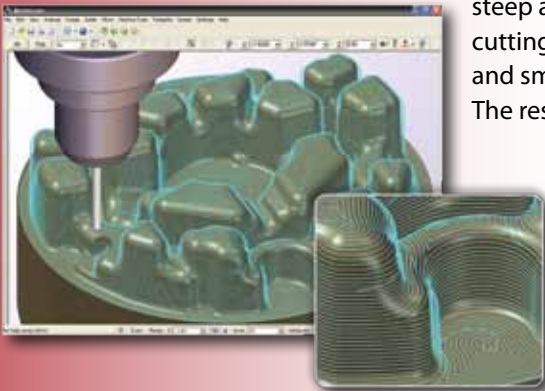
The new 3D surface high speed OptiRough toolpath supports cutters capable of machining very large depths of cut. Large, aggressive cuts are followed by fast, smaller up-cuts, safely delivering a fully roughed part faster than ever. A single OptiRough toolpath can cut material in two directions: on step-downs (-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.



Remove bulk material faster and with more even tool wear.

New Hybrid Finish Toolpath:

Hybrid machining is the latest addition to the list of finishing toolpaths, and is designed to address both steep and shallow regions of a model. 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.



Hybrid finishing intelligently blends two efficient cutting techniques in a single toolpath.

Easier Multiaxis Machining:

Mastercam's multiaxis suite has new machining techniques, and a new, intuitive, and workflow-oriented interface. Mastercam lets you choose the basic type of work you're doing using realistic illustrations, and then gives you a clear, step-by-step process for defining how you'll cut the part.





New 2D High Speed Toolpaths (HST):

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

Dynamic Area Mill and Dynamic Core Mill – Select Dynamic Area Mill to contain the tool within pockets or pockets with islands. Use Dynamic Core Mill when you need to approach material from the outside while still avoiding desired areas.

Dynamic Rest Mill – This new toolpath behaves in a manner similar to the current 2D high speed rest mill toolpath, only it uses dynamic milling motion instead of core mill or area mill motion for the rest operations. In addition, the toolpath includes a new Rest Material page, expanding your ability to specify the targeted rest material.

Dynamic Contour – This new toolpath uses an intelligent, efficient high speed contouring strategy to remove material along walls. It supports multiple passes and can optionally include finishing passes.

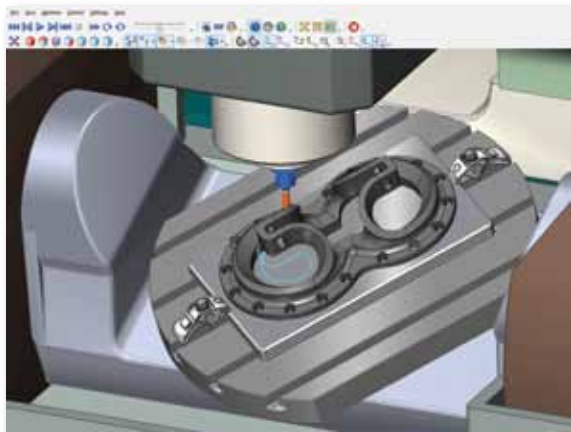
ISCAR High Efficiency Machining (HEM) Support:

Selected 2D and 3D toolpaths in Mastercam now support ISCAR HEM and RCTF (Radial Chip Thinning Factor) speed and feed calculation functionality.

- 2D HST (Dynamic Area Mill, Dynamic Core Mill, Dynamic Rest Mill, Dynamic Contour)
- 3D HST (OptiRough)

Mastercam Machine Simulation:

Machine simulation is a safe and cost effective way of proving out multi-axis toolpaths. Machine simulation lets you build your machine's virtual replica on your computer screen, where you can safely simulate the cutting process to make sure that you created the most effective cut, that the part is located in the machine's 'sweet spot', and that no fixtures, tools, or any machine components are meeting unexpectedly.



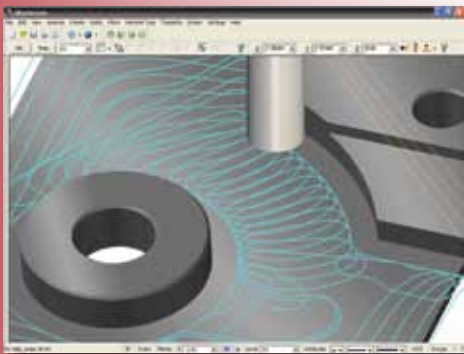
Machine simulation isn't only used to prove out, where the sole aim is finding errors in your code. It is also a tool to help you visualize the movements of the entire machine. It promotes innovation by enabling you to find the best work-piece location on the machine and then aids you in selecting or designing the right fixture for the job. Machine simulation can help you make clean, efficient, and accurate programs every time because it lets you test different approaches and different cutting strategies on different machines, while never leaving your desk. ☒



For more information on Mastercam Mill, and to see videos of the new functions, please visit www.mastercammill.com.

MILL

Mastercam and ISCAR Collaborate for Better Machining



We've found that some of the best techniques come from collaborating with experts in the manufacturing industry. We have our expertise—creating efficient tool motion—but that is enhanced when we develop techniques in conjunction with people who have highly specialized knowledge in other fields. An excellent example of

this is our new collaboration with tooling innovator ISCAR Metals, Inc.

After extensive testing and development, ISCAR unveiled a new line of tools called Chatter-Free. They have a variable pitch and Helix which do not allow harmonics to establish, which could result in poor finish and reduced tool life. An important part of these tools' creation was development of specific formulas to calculate the best feeds and speeds for these specific tools.

Mastercam is working with ISCAR to select toolpaths and produce toolpath settings which make the most of the new tools. This combination of technologies is called High Efficiency Machining (HEM).

What is HEM?

ISCAR developed formulas to calculate feed rate and spindle speed in steel and stainless steel. HEM can only be invoked if you are using an ISCAR tool from the Mastercam Tool Library. When that happens, certain tool usage rules come into play such as stepping a tool over less than 10%. The usage rules guarantee that the tool is used correctly to maximize its potential and fail safe usage. It's using small stepovers, and using full flute length, resulting in a high metal removal rate.

The concept is to marry the physical qualities of ISCAR's tooling and their special feed / speed formula with Mastercam's dynamic milling techniques to get the most out of both the tooling

and toolpaths. After selecting the tool you can access the ISCAR HEM checkbox. Simply enter an appropriate SFM and FPT based on ISCAR's recommendations.

How does it work?

HEM runs ISCAR's Chatter-Free tools at high RPMs and feed rates. The high RPM speed allows you to make a cleaner cut.

You are benefitting from the speed of the cut in a number of ways. In addition to the faster machining, you actually make the heat generated work for you rather than against you—it softens the metal slightly for easier removal. The combination of stepover, RPM, and feed make sure the chips have the appropriate thickness/mass to carry the heat thereby not transferring heat to the tool and part. Since heat is the enemy of carbide, this allows for better tool life as well as better finish.

What are the benefits?

There are substantial benefits to using the new Mastercam/ISCAR HEM settings. First, cycle time drops as you take a full depth cut and need fewer step-downs and less tool travel. Second, your tool life is extended as the tool experiences more even wear and takes advantage of Mastercam's inherently tool-friendly dynamic milling technique.

One example is Classic City Mold in Indiana. They were using Mold plates from the PCS or DME catalog. They had to cut 3 parts for A side and 3 parts for B side. With a feedmill tool it took 1 hour and 5 minutes to complete. With an ISCAR feedmill it took 45 minutes.

With an ISCAR CF tool and Dynamic Mill using HEM feeds and speeds it took only 15 minutes.

This is just one example of how some of the best techniques come from collaborating with other companies! ☑

LATHE

New Cutting Strategies and More in X5 Lathe

As with Mastercam Mill, Mastercam X5 Lathe sees a suite of new features aimed at increasing your productivity.

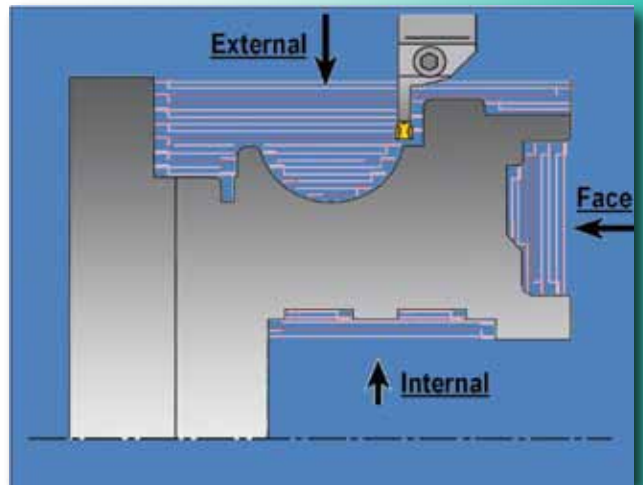
New Plunge Turn Toolpath:

Mastercam's new Plunge Turn toolpath was designed in partnership with ISCAR Metals, Inc. The toolpath consists of custom roughing and finishing strategies designed specifically for ISCAR Cut Grip inserts. This efficient turn/groove toolpath strategy achieves a high degree of accuracy with tight tolerances.

Cutoff Toolpath Enhancements:

Mastercam X5 Lathe includes the following improvements for cutoff toolpaths:

- Option to define the entry amount (for both main and clearance cuts) and incremental retract radius from the stock model, rather than the selected cutoff point.
- Addition of a back face stock parameter so you can select the cutoff point on the back of the finished part and still leave stock for back facing.
- Ability to apply a secondary feed rate and spindle speed at a defined radius.



Strategies designed specifically for ISCAR Cut Grip inserts.

Turn Profile True Arc Fitting:

Often using the Turn Profile feature, many small arcs are generated where there should only be one large arc in the geometry. This occurs because the profile is initially computed as a polyline to which the arcs are 'fit' in a subsequent step. A new module has been integrated into Mastercam X5 Lathe that fits 'true' arcs and lines to the polyline. 'True' arcs and lines are entities that exist in the wireframe construction geometry used to create the solid or that are part of the edge curves of the solid. By attempting to fit these arcs and lines to the profile, a much more accurate representation of the geometry can be generated.

Roughing Toolpath Improvements:

Tool Inspection – After successfully adding tool inspection to groove toolpaths, we have added similar functionality to the rough turning toolpath. This feature allows tool inspection between depth passes, after a specified cut length, or after a specified time.

Exit Amount – Rough turn strategies can now accommodate open space at the end of a toolpath. We have added an "exit amount" operation parameter that extends the cut beyond the stock by the amount specified.

Boring Bar Holder Improvements:

Mastercam's verification and collision checking has been enhanced to create a more accurate visual representation of boring bars, while still maintaining accurate collision detection.

Visit www.mastercamlathe.com for more information on Mastercam X5 Lathe. ☒

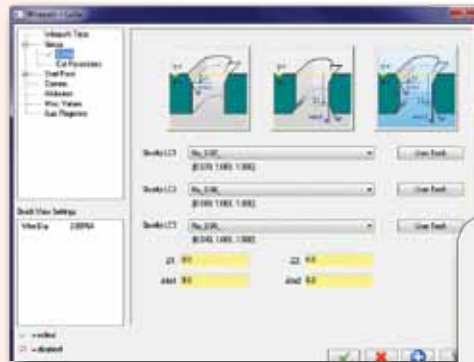
WIRE

New Power Library Format in Mastercam X5 Wire

In addition to Mastercam's current POWER libraries, Mastercam X5 adds new TECH power libraries. These new database files make it easier than ever to select the correct power settings based on your machining needs. A TECH library is machine control-specific. For example, when you load an EDM machine definition such as the Mitsubishi or Makino supplied with Mastercam X5, it also loads the control-specific TECH-based library.

When you are creating a wirepath using a TECH-based machine/control definition, the power library settings can be accessed through the **Wire/Power** page of the wirepath dialog boxes.

For more information on Mastercam X5 Wire, please visit www.mastercamwire.com.



Agie-specific controls offer a familiar working environment.

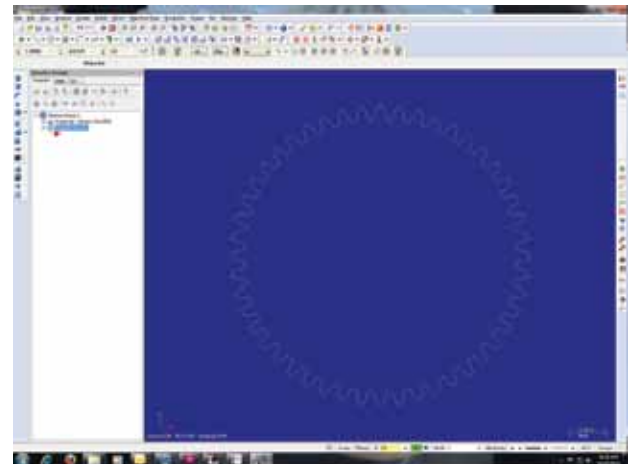


Streamlined settings help speed programming.

Did You Know About Mastercam's Gear C-Hook?

You can use this Gear C-Hook to create a single tooth or all teeth of an involute spur or internal gear. It also computes the roll measurement for checking the gear.

The values are saved in the parameter file referenced in the dialog box so that they will persist between Mastercam sessions. Change the name to recall parameters from a different gear. ☒



ROUTER

Block Drilling, Automatic Toolpathing, and More

Mastercam X5 Router continues the trend of faster and more efficient programming with enhancements that will keep your shop ahead of the competition.

Block Drill Support in FBM Drill:

To minimize tool change time, Feature Based Machining (FBM) Drill now recognizes block drill units that are configured in the active Router machine definition. FBM Drill can assign block drill tools and generate block drilling toolpaths.

Importing Solids into Toolpath Nesting:

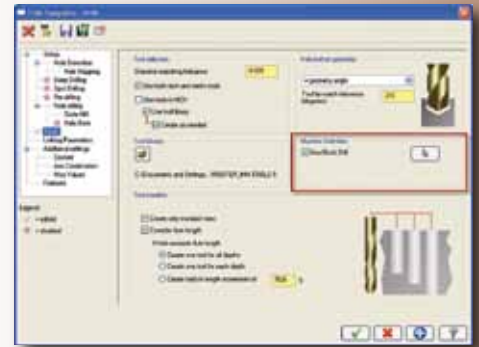
In Mastercam X5, both Rectangular and TrueShape Nesting support solid operations, including operations created by FBM toolpaths. FBM parent operations, however, are not yet supported.

Automatic Toolpathing Enhancements:

Mastercam X5 includes several improvements to Automatic Toolpathing (ATP).

- You no longer have to be in Mill or Router to run ATP. When running ATP, the ATP machine definition overrides Mastercam's machine definition.
- You can overwrite or append new levels to an existing scan.
- You can create new operations from the Strategy Explorer.
- Two new options have been added in the Options dialog box for cutlist configurations.
- WCS support has been implemented for ATP Single Part Processing.

For more Router information, please visit www.mastercamrouter.com. ☒



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
ProDrill is a product of Tecnomim S.L. and Moldplus S.A.
Solutions that make a difference! www.moldplus.com
info@moldplus.com

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 DS SolidWorks Corporation.

DESIGN

Many New Enhancements for Mastercam Design

Whether your shop gets files from an outside source or designs them in house, modeling tools is an essential element in prepping a file for machining. Mastercam X5 Design delivers efficient new tools to the entire Mastercam product line.

Assigning Colors to Solid Faces:

This new capability allows users to change the colors of entire features or individual solid faces of a Solid Body. It also highlights individual solid faces during selection, helping you to see clearly what you have selected.



Change the color of entire features or individual solid faces of a Solid Body.

Solid Patterns:

You can create a pattern of solid features by selecting a source and producing multiple copies of it on a part in a grid (rectangular pattern) or around a center point (circular pattern). You can also use the AutoCursor to place copies (manual pattern).



Create a pattern of solid features and create multiple copies on a part.

Solid and Surface Analysis Tools:

Analyze draft angles and curvatures of a surface to identify undercuts and minimum radius with two new Analyze functions: Analyze Draft Angle and Analyze Curvature. These new functions shade areas of the model with user-specified colors and support both surfaces and solids.

Drill and Tap table values are just a mouse-click away:

You can specify diameters quickly and without consulting paper drill charts. Standard letter and number drill and tap sizes have been added to all Mastercam calculable fields.

Improved Bolt Circle Creation:

Bolt circle has been enhanced to give you the ability to create bolt circles:

- in a 2D plane or wrapped around a diameter
- about a full circle or along a total sweep angle
- Preview and, if desired, flip the results
- Remove individual circles
- Create a center point



For more information on Mastercam X5 Design, please visit www.mastercamdesign.com. ☒

ART



New Features in Mastercam X5 Art

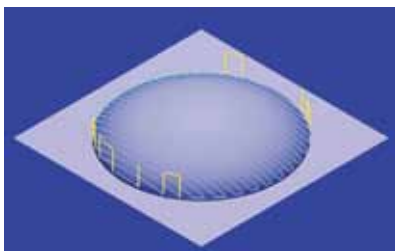
New Art Toolpaths:

Mastercam X5 includes four new Art toolpath options: Parallel Spiral, Spiral Rectangular, Spiral Circular, and Spiral Follow Boundary.

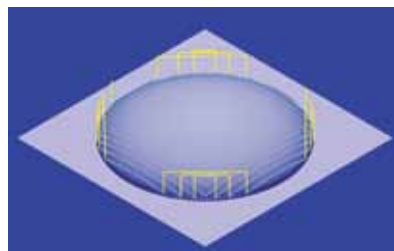
All these new toolpaths can contain the toolpath to a user-defined boundary, as well as support internal islands and pockets (nested boundaries). This allows you to increase productivity by cutting only what is necessary. Additionally, the new spiral toolpaths can cut either from the inside-out or from the outside-in. And while these toolpaths have broad application, they are especially useful for cutting harder materials like those used in the coining, engraving, and paper embossing industries.

Laser-cut Sculptural 3D Surfaces:

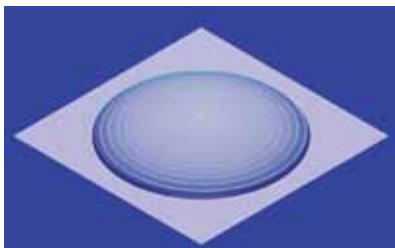
We think of lasers as 2D cutting devices for only engraving or pocketing, but they can also quickly and easily cut sculptured surfaces from simple 2D artwork. You simply export an Art model with one click to a perfect 256-level grayscale image of the Art model as a BMP, JPG, or TIFF file. This perfect grayscale image is passed to the laser control software, where the 3D image can be cut. The laser will change focus/intensity on each pixel, utilizing multiple passes to cut the 3D image from the material. ☒



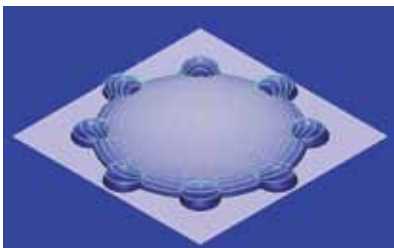
Parallel Spiral



Spiral Rectangular



Spiral Circular



Spiral Follow Boundary

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!

Powerful and easy to use **ELECTRODE** maker features

www.moldplus.com
info@moldplus.com

Mastercam
Tips For Manufacturing LLC **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 or...
www.TipsForManufacturing.com
Questions? Call Us at... (480)-986-9528

ADD-ONS

Robotmaster X5

Robotmaster® seamlessly integrates robot programming, simulation, and code generation inside Mastercam, delivering quicker robot programming. Robotmaster is ideal for all applications including trimming, 3D machining, deburring, polishing, dispensing, grinding, and more.

Jabez Technologies introduces Robotmaster X5. Expanding on powerful optimizing tools, Robotmaster X5 has set a new standard for programming robots with the same ease and functionality as CNC milling machines.

Common robot programming challenges typically require intricate user intervention or costly manual editing for producing error-free programs. Robotmaster X5 provides innovative new tools to effortlessly optimize robot programs producing error-free robot paths, avoiding singularity zones and robot/tool/workpiece collisions, working around each joint and reach limitations, and optimizing tool orientations along the entire trajectory.

Robotmaster users get the cost-efficiency and flexibility of robots and the ease of programming with CAD/CAM tools typical of CNC manufacturing. Optimized trajectory programming is generated directly from CAD/CAM data in a single, fully integrated software solution, all off line and without teaching points.

For more information on Robotmaster, please visit www.robotmaster.com.



Master3DGage Offers Affordable & Portable Rapid 3D Inspection Solution

Master3DGage gives machine shops an affordable and portable CMM solution. Powered by Verisurf inspection software, a six-axis portable CMM offers a rapid, fully automated 3D inspection process that improves product quality, manufacturing productivity, and shop profitability. Master3DGage is sold and supported exclusively through select Mastercam Resellers worldwide.

“For the first time, machine shops will have a precise 3D model-based inspection system that can quickly verify parts directly on a CNC machine or anywhere else in the manufacturing process,” said Tom Shelar, president of CAD/CAM Consulting Services, Inc., a leading Mastercam Reseller. “Implementing portable inspection into the manufacturing process identifies issues quicker, reduces scrap, and significantly improves productivity.”

Rapid Three-Step Inspection

- 1. Align:** Align a manufactured part to 3D CAD model by probing part to corresponding alignment targets on 3D CAD model.
- 2. Inspect:** Inspect a manufactured part in real time by manually probing or by following automated inspection plans.
- 3. Report:** Report inspection results are automatically generated in HTML and Excel formats.

The Master3DGage rapid inspection process improves manufacturing productivity in several ways. First, it eliminates dependence on 2D drawings, dozens of hand tools, and gages. Second, it eliminates the need to route products away from the manufacturing process to fixed inspection areas. Third, it completes a fully digital 3D design, simulation, manufacturing, and inspection workflow.

GagePlate

The new GagePlate system is a major innovation that dramatically expands the capabilities of the Master3DGage portable rapid 3D inspection solution. Quality engineers can now use the affordable Master3DGage to inspect large parts without sacrificing accuracy.

“Adding the new GagePlate System to the Master3DGage enables quality engineers to inspect much larger parts for a fraction of the cost of larger portable or stationary CMMs,” said Ernie Husted, president of Verisurf. “Master3DGage’s light weight, stable base, battery power, and WiFi connectivity make it easy to move around and is ideal for inspecting large parts placed on the GagePlate.”

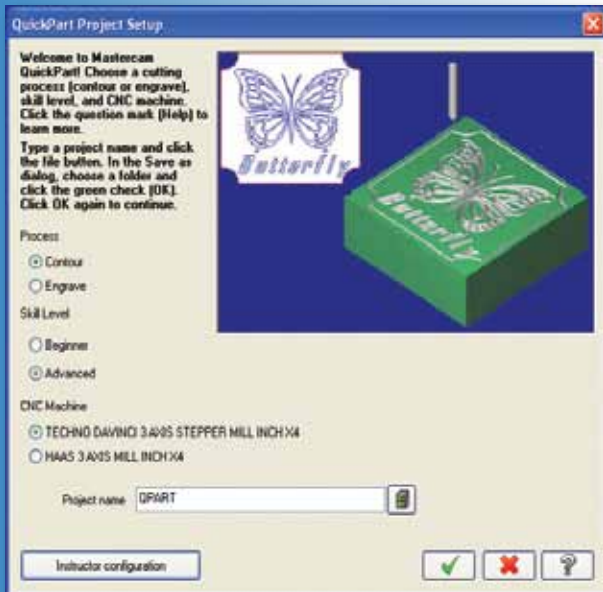
The GagePlate system automates realignment of the Master3DGage around large parts under inspection, while referencing one coordinate system. The precision-calibrated plate rapidly scales and aligns the Master3DGage, enabling the inspection of large parts without the accumulative errors associated with leapfrogging, or the costs associated with large-scale CMMs.

Master3DGage software can also precisely compensate for temperature-related measurement variation. It automatically scales and compensates the known Coefficient of Thermal Expansion (CTE) of the part under inspection with the CTE of the calibrated GagePlate. ☒





EDUCATIONAL NEWS



QuickPart takes a student step by step through a basic CAM application.

Mastercam QuickPart: A Favorite Learning Tool Returns

CAD/CAM can be challenging to learn, and even more challenging to teach. Mastercam QuickPart gives students a fast, easy way to design and cut parts while learning the principles of CAD/CAM, and have fun doing it. Mastercam QuickPart is easy to use, and provides a path for instant success.

QuickPart takes a student step by step through a basic CAM application. It uses Mastercam functionality to let a student first design a part and then create toolpaths to cut the part. From material selection to stock size, to tool selection, a student will learn the methods involved in manufacturing with CAM software.

Mastercam QuickPart is a standalone product that has been developed on the Mastercam X4 Demo/HLE platform. ☒

Wildest Parts Contest

The challenge is simple... create a bold, original idea using Mastercam.



Mastercam's Wildest Parts

Competition challenges students to design and make something that is wild! We are looking for high quality parts that either haven't been done before or have never been done like that before. We want you to break the mold, think outside the box, and cut outside the lines.

We are looking for eye-catching parts. We have designed a display case for our show booths, specifically for the Wildest Parts Competition. This case travels to the top educational shows, and people from all over the country stop to look at the parts. We are looking for parts that will demonstrate the many impressive and creative ideas that can be made with Mastercam.

For more information on the Wildest Parts Competition, please visit www.mastercam.com/wildestparts. ☒



OTHER NEWS



What's New at Mastercam University™?

All Mastercam University classes have been updated to the Mastercam X5 platform, so be sure to check them out if you want to get up to speed with the new release.

Associate Level of Certification:

New to Mastercam University is an Associate level of Certification, and it's available for the following classes:

- Mill/Design and Toolpaths
- Advanced Mill/Design and Toolpaths
- Lathe/Design and Toolpaths

Mastercam University has a pre- and post-test in each of its modules. When a student taking the course gets a score of 80% or higher on every test, they will achieve the Associate level of Certification. This is a way to acknowledge that the student did very well in the course.

To achieve that Associate level, you must have a post-test score of at least 80%, your final test score must be 80% or better, and you must have completed all of the material in the Mastercam University course.

Free Promotional Course:

The Mastercam Enhancements course is now available for free (to Maintenance customers) on Mastercam University. This course consists of the first five lessons only of the first module of the V9 to X5 Transition course. This course will be available only on Mastercam University, and no code is needed to run it. Contact your Mastercam Reseller for more information.

For a syllabus of the complete V9 to X5 Transition course, please visit <http://tinyurl.com/Xtransition>.

For more information on Mastercam University, please visit www.mastercamu.com and www.mastercam.com/Support/Training/MastercamU. ☒



Get Integrated PSE!

Post-processing
Simulation
*Emulation
*using G-Code

Supports Multi-Axis
Mill & Mill / Turns!

ICAM

icam.com/EASY-QUOTE JIMTOF 2010-E5031

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



OTHER NEWS

Don't Miss Out on Mastercam X5. Update Your Mastercam V9 System NOW!

You might not have realized it, but your Mastercam Version 9 software was released nearly a decade ago. A decade of technological development is a very long time. Computers have significantly evolved during this time, and today's CAD/CAM technology has taken advantage of those improvements. By not keeping your Mastercam system current, you are missing out on all the advances — not the least of which are dramatically increased processing speed, ease of use, shorter cycle times, and machining flexibility.

Mastercam X took a huge technological leap forward, and Mastercam X5 continues to build on that powerful platform.

If you are a customer running Mastercam Version 9, now is the time to update your software. With the release of Mastercam X5, Mastercam Version 9 will no longer be supported and you will not be able to update it without buying at full price. By updating now, you avoid having to repurchase the software. Act now and you can save up to 70% off the latest version of Mastercam. If you don't act soon you will lose your valuable investment in Mastercam! ☒

Time is Running Out



Mastercam. Version 9

www.mastercam.com/update



The New iMastercam Store

iMastercamstore.com™

We are pleased to announce the opening of the new iMastercamstore.com, a new site for fans of Mastercam worldwide to purchase Mastercam clothing and other items.

This new online store has a selection of polo shirts, jackets, sweatshirts, T-shirts, hats, work shirts, travel mugs, mouse pads, and more. We now have Tap/Drill charts at an amazingly low price of \$5 US. There is also a **Closeouts** section where you can find great bargains on quality items in limited sizes.

iMastercamstore.com accepts all major credit cards including VISA, MasterCard, American Express, Discover, JCB, and in the near future, will also accept PayPal™ payments.

To introduce the new store, we are offering a special promotion: Enter promotion code **GIVING5** during check out and receive 5% off your entire order, excluding shipping and tax. This offer is only valid until November 30, 2010, so act now.

If you have questions, suggestions, or requests for the new iMastercamstore.com, you can send them to support@imastercamstore.com. And please do not hesitate to contact us if there are any items or sizes you are looking for but cannot find on the site! Contact us at (877) 815-3172 (US only) or (860) 282-1632 (worldwide) Monday - Friday, 9 AM - 4:30 PM EST. ☒

Mastercam® is a registered trademark and iMastercamstore™ is a trademark of CNC Software, Inc.
©CNC Software, Inc. 2011. All rights reserved.



CNC Software will be showcasing Mastercam X5 all around the world. Here is just a sampling. To see a complete list, please visit www.mastercam.com/events.

November 2-5
SEMA
Las Vegas, NV
Booth # 10328

December 1-3
IMIS
Indianapolis, IN
Booth # 2514

December 2-4
ACTE
Las Vegas, NV
Booth # 719

December 9-11
PRI
Orlando, FL
Booth # 4591



Mastercam LIVE!



It's Mastercam X5 LIVE and it's coming to a location near you. This worldwide rollout delivers information-packed sessions demonstrating what Mastercam X5 can do for you and your shop. Check back often as new events are being added frequently. And you can always check with your local Mastercam Reseller to find out when they have an event coming up if you don't see your city on the schedule.

For more information on Mastercam LIVE, check your 4th Quarter Customer Newsletter, or check out the website www.mastercam.com/x5rollout.

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