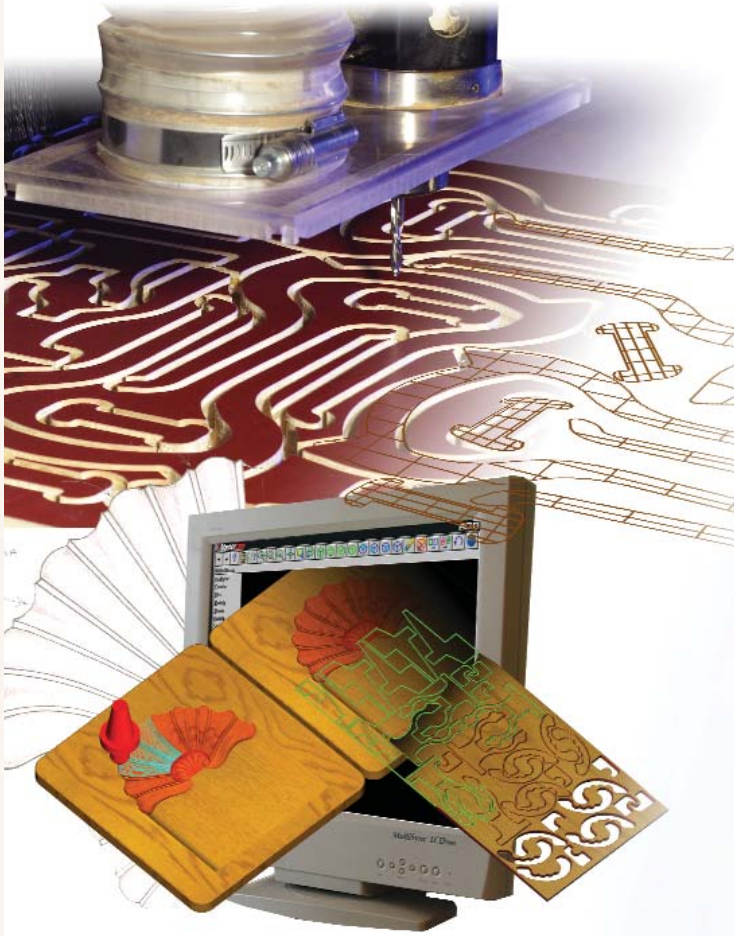
Mastercam Notes
Customer Newsletter

First Quarter 2003
Vol. 12 No. 1

Mastercam Router V9 Cuts a New Path



Mastercam Router Version 9 is hitting the streets. With a series of easy-to-use new CAD/CAM tools and powerful enhancements, this updated software provides greater speed, more flexibility, and new functionality.

But Mastercam Router V9 is more than just a collection of industry-leading CAD/CAM programming tools. It's software specifically designed to meet your changing needs. These new features are based on input and requests directly from our customers. We talked to everyone from machine tool manufacturers to shop operators, and incorporated the changes they said would make their work easier.

This article highlights a few of the new features and explains how they can benefit your shop.

(Continued on pg. 2)

Contents

Product News.....2-8
Customer Profile.....9
Tech Tips.....10-11
Mastercam in the Wood Industry.....12
Marketing News.....13
Third Party Advertisements.....14-15
What's New.....16

cnc software, inc.

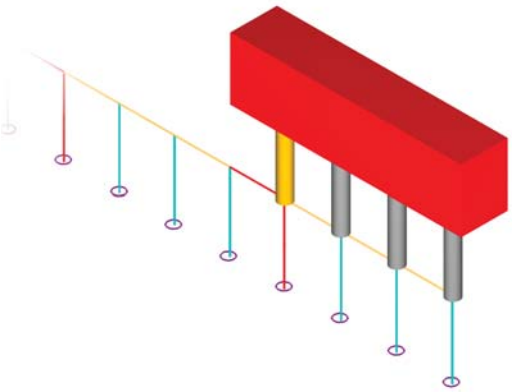
Mastercam Router Cuts a New Path

(Continued from pg. 1)

New Feature: Block drilling

What it does: Mastercam Router V9 can control block or "gang" drills, including straight and "T" block drills. Mastercam easily optimizes the drilling routine, dropping the correct drills according to the drill pattern that you are machining.

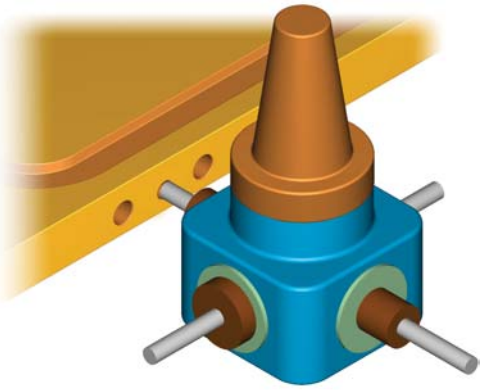
How this benefits your shop: Easy, complete control over block drilling routines means faster turnaround and more efficient use of your shop's time.



New Feature: Aggregate head programming

What it does: In addition to Mastercam's wide variety of tool shapes and types, Router V9 has the added flexibility of Right Angle (RA) and Compound Angle (CA) head control. This lets you take toolpaths you created from any view and apply RA and CA heads to them.

How this benefits your shop: Efficient RA and CA head control saves time by allowing you to do more machining in a single setup. It may also allow you to take on complicated jobs that were too difficult or time-consuming with multiple setups, but are much easier when you don't have to remove the piece from the router bed.



New Feature: VBScript support

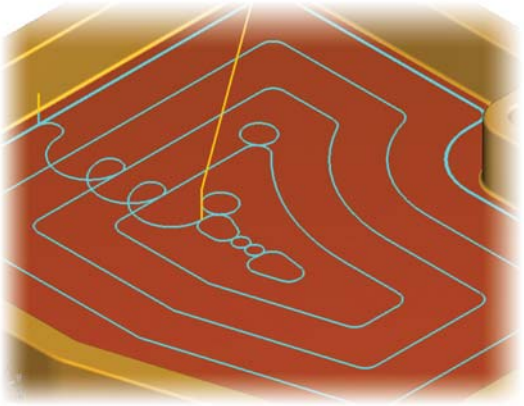
What it does: Mastercam Router V9 includes new software utilities that are written in Microsoft Visual Basic®, a widely used programming language.

How this benefits your shop: VB programmers are common, so Mastercam's ability to run VB scripts creates opportunities for an even wider variety of focused utilities that can make your job easier. For more information about VB scripts and how they can benefit you, see "Easily Automate Repetitive Tasks With VBScripts" on page 7 of this newsletter.

New Feature: High Speed Machining

What it does: High Speed Machining (HSM) has become increasingly popular in the past few years. One of the main goals of HSM is smoother tool motion with very few sharp movements. Mastercam Router V9 gives you HSM abilities including 2D high speed pocketing.

How this benefits your shop: Your router servos can be damaged by fast, sharp-angle moves that traditionally occur during tool direction changes. HSM reduces sharp tool movements with specialized motion, so there is less wear on the cutting tool and your router.



New Feature: Specialty software tools

What they do: Mastercam includes a variety of specialized design and programming tools for unique tasks. Examples include Mastercam's door designer (which can create nested, ready-to-cut doors from a Microsoft Excel® spreadsheet) and an automatic stair stringer creator. Of special note are Mastercam's automated links to various cabinet design software packages that automate nesting and programming of complete cabinet designs. For more information, see "New Cabinet Links for Mastercam Router" on page 6 of this newsletter.

How this benefits your shop: Specialty tools like these have two very distinct benefits: they save you time and make you money. The time you save by using Mastercam's automated links to cabinet software can make a job much more profitable. And the extra capabilities of these design and programming tools can make the difference in winning a crucial contract.

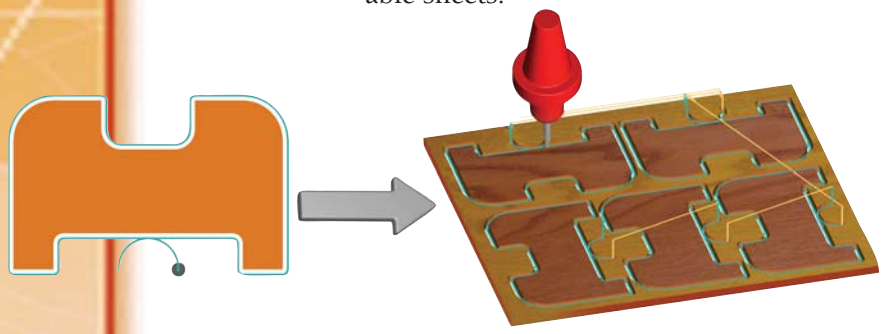
These are just a few highlights of Mastercam Router V9. For a complete list of Mastercam Router's functions, visit www.mastercam.com/router or contact your local Mastercam reseller and see what new trails Mastercam Router can help you blaze.

Advanced Nesting - Making the Most of Your Material

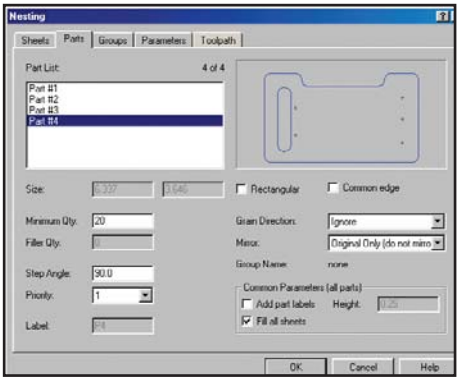
Mastercam Router includes basic nesting, but there are times when you want more sophisticated nesting for a higher yield. Mastercam's optional TrueShape nesting optimizes your material use for greater yields and an improved bottom line. With the TrueShape nesting add-on, you can efficiently nest interlocking parts with a wide variety of options:

- Nest your parts or your toolpaths.
- Advanced automatic interlocking nesting.
- Custom sheet definition.
- Maintain grain direction of any part.
- "Filler" part nesting to reduce waste.
- Mark a series of parts as a "Group" to ensure the entire set is always nested.
- Set minimum number of parts needed.
- Assign priority to make sure that the parts you need the most are nested first.
- Combine parts into "clusters" that will be nested in the same orientation to each other.
- Create and save a library of fully definable sheets.
- Save sheet scrap for later use.
- Full control over part spacing and sheet margins.
- Directly import and place AutoCAD®, DWG and DXF files.
- Automatically separate different parts into levels that you can turn on or off.
- Program your entire nested sheet at one time.

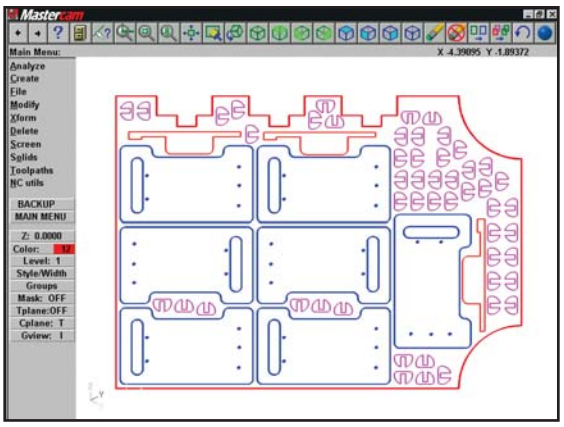
For more information about purchasing the TrueShape nesting add-on, contact your Mastercam reseller.



Mastercam's toolpath nesting lets you program a part once, exactly the way you want it, then nest that operation so that each part is cut with identical motion. Toolpath nesting is available in both Mastercam's basic and advanced nesting.



Mastercam's advanced nesting gives you high yield with just a few simple steps.

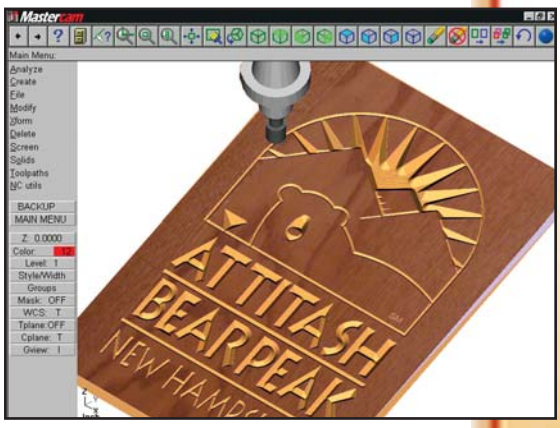


You get a wide variety of nesting tools, including nesting parts in scrap from a previous session.

Mastercam's Engraving Helps Carve Out New Opportunities

Mastercam's optional Engraving gives you the effect of classic hand-carved art using your CNC machine. Sharp inside corners give the finished piece a crisp edge which is unattainable with conventional machining. Mastercam's Engraving add-on gives you a wide variety of choices:

- Standard "chiseling."
- Embossed images.
- Combined chiseled/embossed images.
- Emboss "reversed" images.
- Wrap engraving toolpaths to surfaces or between curves.
- Open pixel-based artwork as engravable geometry (.BMP, .GIF, .TIF, .PCX, .EPS and more).
- Easily engrave any TrueType® font.
- Inverse embossing for mold work.
- Engrave remachining cuts an area with a large tool and automatically removes remaining material with a smaller tool.
- Use constant-volume depth cuts for more consistent tool load.
- Use V-groove cutters with any angle.
- Engrave with flat cutters to remove floor stock.



Easily produce a wide variety of parts with the look of classic, hand-carved art.

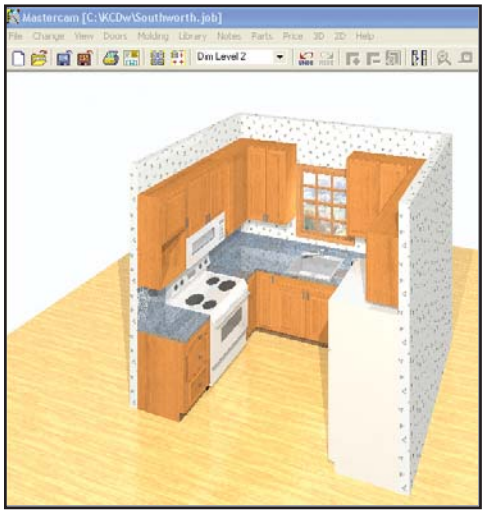
For more information about purchasing the Engraving add-on, contact your Mastercam reseller.

New Cabinet Links for Mastercam Router

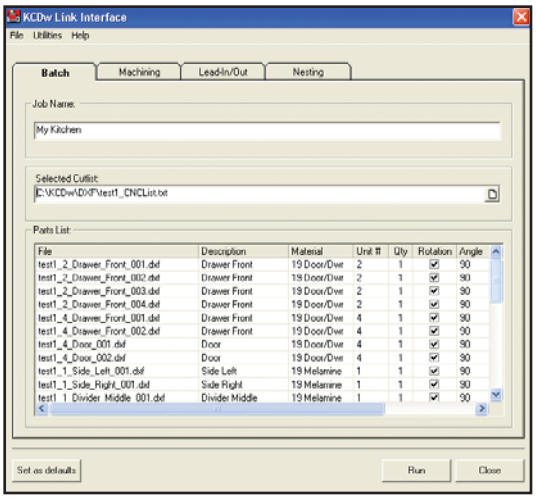
With the growing popularity of cabinet design software, CNC Software introduces a Cabinet Link for Mastercam Router. This new link automates the sorting, optimization nesting, and cutting of cabinet components from many of the major cabinet design software packages such as KCDw, Pattern Systems, and a variety of others.

Since Mastercam can read any DXF file, any package with the ability to output DXF files can be supported by Mastercam's new link. Here's an example of Mastercam's link with the popular KCDw kitchen design software:

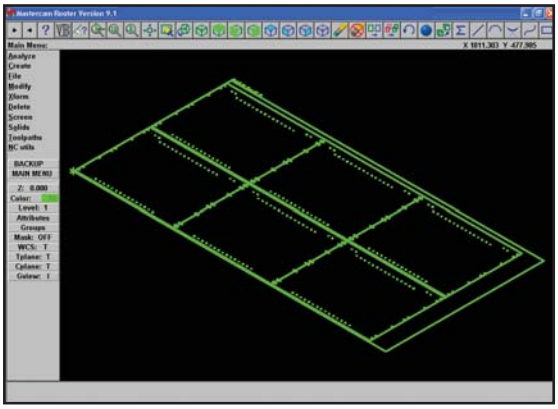
1) The kitchen is designed in KCDw. Once the design is complete, the user can select the construction method (dowel, dado, frameless, etc.), and output the individual components of the kitchen as DXF files. These DXF files are all referenced in a single text file called a "cutlist."



2) Open Mastercam and launch the KCDw Visual Basic Link. A dialog box will appear. Simply select the cutlist that was output by the cabinet software, and Mastercam's link will read in the cutlist.

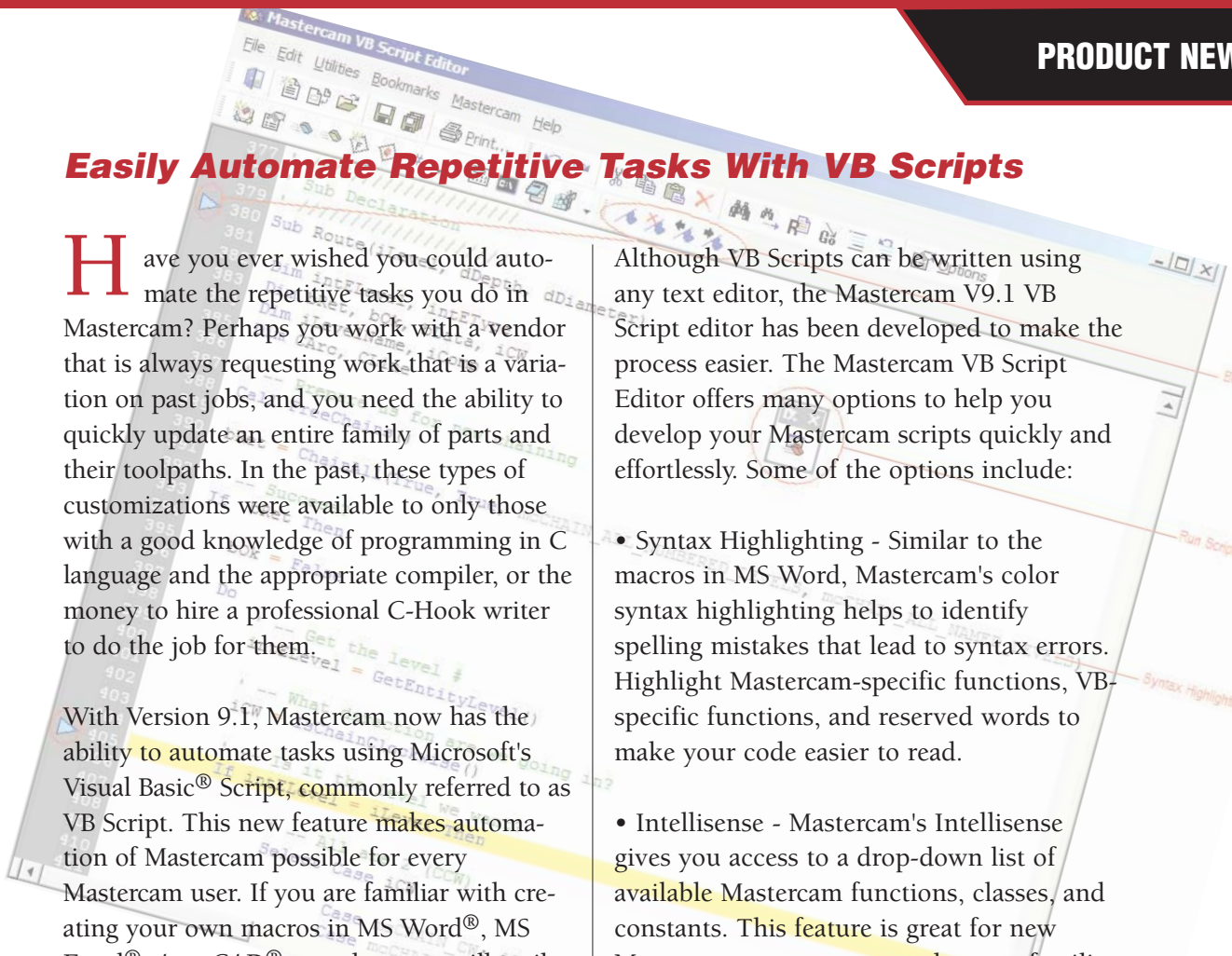


3) The user can then input default machining and nesting parameters that can be saved for future jobs. Once the parameters are input, simply run the link.



Once run, Mastercam will automatically sort by material, toolpath, nest, label, and output NC code for every component of the kitchen design, leaving the operator with nothing to do except load the machine and run the job.

Easily Automate Repetitive Tasks With VB Scripts



Have you ever wished you could automate the repetitive tasks you do in Mastercam? Perhaps you work with a vendor that is always requesting work that is a variation on past jobs, and you need the ability to quickly update an entire family of parts and their toolpaths. In the past, these types of customizations were available to only those with a good knowledge of programming in C language and the appropriate compiler, or the money to hire a professional C-Hook writer to do the job for them.

With Version 9.1, Mastercam now has the ability to automate tasks using Microsoft's Visual Basic® Script, commonly referred to as VB Script. This new feature makes automation of Mastercam possible for every Mastercam user. If you are familiar with creating your own macros in MS Word®, MS Excel®, AutoCAD® etc., then you will easily be able to develop Mastercam scripts. This is an invaluable tool for automating repetitive tasks, family of parts applications, and making complex tasks easier.

Here's a scenario that could become a reality for a customer with a specific product, like entertainment centers, that vary in size and material. The order gets taken by phone or via e-mail. As the order entry is completed, a VB Script grabs the sizes, quantities of the pieces, and materials that make up the product. This information is used by Mastercam to create the parts, create toolpaths, nest the parts, and send the information to the router on the shop floor. How can someone who knows Visual Basic do this?

Although VB Scripts can be written using any text editor, the Mastercam V9.1 VB Script editor has been developed to make the process easier. The Mastercam VB Script Editor offers many options to help you develop your Mastercam scripts quickly and effortlessly. Some of the options include:

• Syntax Highlighting - Similar to the macros in MS Word, Mastercam's color syntax highlighting helps to identify spelling mistakes that lead to syntax errors. Highlight Mastercam-specific functions, VB-specific functions, and reserved words to make your code easier to read.

• Intellisense - Mastercam's Intellisense gives you access to a drop-down list of available Mastercam functions, classes, and constants. This feature is great for new Mastercam programmers to become familiar with all the available functions. Currently there are more than 200 functions available.

• Bookmarks - Bookmarks can be used to mark lines of code in the editor so that you can easily return to them later.

• Commenting - Code can be commented out or uncommented by selecting the lines you want and clicking the applicable menu item.

• Advanced search and replace - Search up and down, by word, or by expression and replace. Or search for a specific line.

For a more detailed document covering the use of the Mastercam VB Script editor, visit www.mastercam.com/VBScript.

Integrating the Integrex

Multitask machines (those that can do different types of machining) are a hot item. You've probably seen machines like Mazak's Integrex and Mori Seiki's MT at trade shows. They always look great on the show floor, but how would they work on *your* shop floor?

These machines use a milling spindle with an index or simultaneous B-axis, C, X, Y, and Z axes on both main and sub-spindles. They can do impressive work, but they need to be controlled properly. That's where Mastercam comes in. In addition to the milling, turning, and specialty tools that we've developed, we also have an Integrex post processor that you'll find it at many Mazak outlets worldwide, helping them turn out top-notch parts. Visit your local Mazak dealer or go to www.mazak.com and find out how these machines can expand your business.

If you want to explore out other multitask machines, be sure to visit Mori Seiki's MT (www.moriseiki.com), Okuma's MacTurn (www.okumaamerica.com), and Hitachi's HiCell (www.moriseiki-hitech.com).

New Partnership Announced

CNC Software and Northwood Designs have entered into a strategic partnership to provide the benefits of the MetaCut Utilities (MCU) to Mastercam customers. Starting with the release of Mastercam V9.1, CNC Software will be including Northwood's MCU Light suite of toolpath utilities with Mastercam and will begin selling the upgrade to MCU - Level 1.

MCU is available in two levels. The "MCU Light" version bundled with Mastercam, and "MCU-Level 1," which has additional features including 3-axis G-code verification, advanced graphical edit, revisions management, and compare.

To introduce this new partnership, Northwood Designs has announced an upgrade program available only to Mastercam users, and only until May 15, 2003. Until May 15, Mastercam users can upgrade to MCU - Level 1 for the special price of just $499 – a substantial savings over the normal price of $600.

For more complete details about the features and abilities of MCU, visit the MetaCut web site: www.MetaCut.com. To purchase upgrades or stand-alone copies of any MCU product, contact your Mastercam dealer.

"We see this partnership as a real benefit to our customers. MCU Light provides the most powerful G-code editing, backplotting, and analysis we've seen for the CNC market and the availability of a low cost upgrade to stand-alone G-code verification is a great bonus."

- Mark Summers, President, CNC Software, Inc.

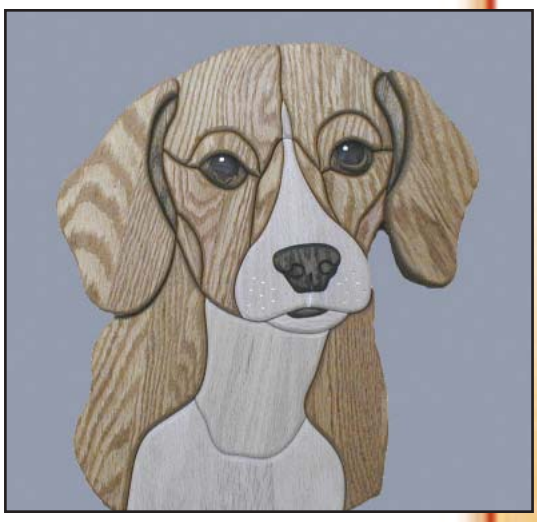
From Hobby To a Profitable Business

CAM Joins Art, People, Pets, and Fine Woods

Like many novel businesses, Kevin Edgar's Wooden Nature began as a hobby of making a mosaic or puzzle from different woods. Each piece is individually cut, shaped, sanded, and finished, then glued to a sturdy backing. The result is a 3D art piece. Word spread about Kevin's band-sawed and hand-sanded creations, and he soon embarked on a search for the optimum technology to increase the production rate of his creations. "I have a passion for the art form and for working in wood," he says, "and it was very important to me that I select a method of design and manufacturing that allowed me to create the quality of handmade artwork at production quantities."

After two years of researching and rejecting production possibilities, Kevin zeroed in on CNC routing as the best of all possible techniques to reach his goals and satisfy his quality requirements. Kevin bought a CNC 28" gantry machine and a seat of Mastercam. Learning on his own in his spare time, he began to import the 2D images he drew and to generate the toolpaths within a couple of hours. "One nice thing about Mastercam is that I can go from a photograph to a completely machined piece in two to three hours, so I have new pieces on the market almost instantly."

"Although Mastercam's toolpaths run exactly as programmed, I may find I've positioned a piece differently from what I'd really like, or I want to alter a shape," Kevin notes. "Since Mastercam files and their toolpaths are associative, I don't have to re-toolpath every time I make a change in a design. It's done automatically." Once Kevin has successfully completed a stock design, his crew takes over for production runs.



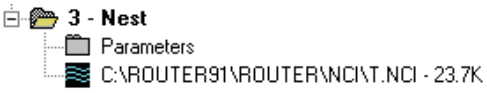
Wooden Nature has been growing exponentially. Kevin says, "It's growing because customers appreciate our artistic abilities and craftsmanship at prices that remain affordable thanks to CAD/CAM production. The reactions we receive from customers make us proud of our quality."

"One nice thing about Mastercam is that I can go from a photograph to a completely machined piece in two to three hours, so I have new pieces on the market almost instantly."

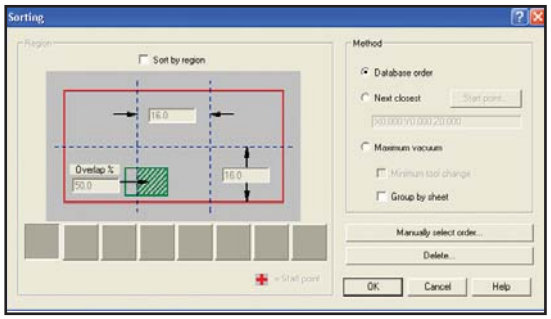
— **Kevin Edgar,** Owner
Manitowoc, WI

Manual Sorting in Nesting

Mastercam's new toolpath nesting is a powerful and efficient way to optimize toolpaths for greater stock yields and more efficient toolpath sorting. Many sorting methods are available to provide functions like maximum vacuum and minimizing tool changes, but there are times when you would rather manually select the toolpath order. This is when Manual Sorting can be used. This can be misleading at first, because when you initially nest toolpaths, the "Manual Sorting" button is inactive. A nesting operation must already exist before you can change the toolpath order manually.



Go into the parameters of an existing nested operation and enter the sorting parameters on the third tab.



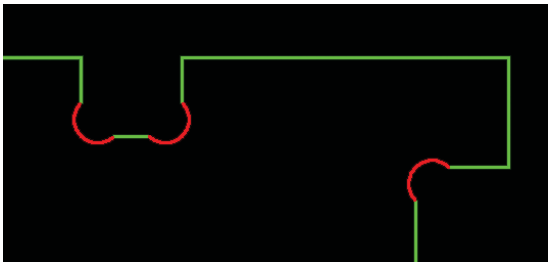
Notice that the "Manually select order" is now active. Simply click the button and select the order in which you'd like the toolpaths sorted by clicking on the bitmaps. The red bitmap is where Mastercam started the previous sorting.

Clearance Fillets

This function is designed mainly for the upholstered furniture industry. These days, the frames of most upholstered furniture are made out of plywood, and are assembled in a "jigsaw"-like fashion.

Because of this assembling process, clearance fillets must be used wherever parts mate so that the parts fit flush with each other.

In Mastercam's Fillet menu, turn the "Clearance" option to "Y" for yes. Now select geometry as you normally would to create a fillet. Please note that only lines can be selected using the clearance option.



Preventing Skips with 4-axis Toolpaths

Q: I run 4-axis toolpaths with bull-nose cutters. Occasionally the tool seems to "skip" to the previous or next cutting path, then back again. It doesn't gouge – the tool just seems to make an odd motion. What can I do to prevent this?

A: This happens when the surface normals cross from leaning left to leaning right relative to the rotary axis. For example: When doing turbine blades, the twist in the surface causes the vector to swap sides.

If the vector crosses from leaning left to leaning right, the contact point on the tool moves to the other side of the flat on the tool end. That's the "skip."

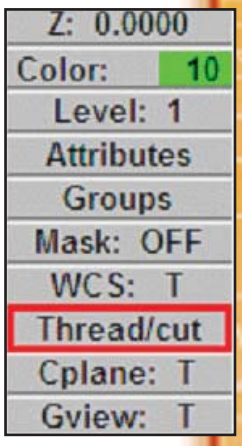
An easy solution is to apply some lead/lag to the 4-axis toolpath. This forces the contact position to "roll" to the front or back so the vector leaning left or right causes very little tool center motion.

New Thread/Cut Pointers For Easier Wirepath Definition

Wire Version 9.1 is now capable of using special Thread / Cut points for easier wirepath definition. New point styles include:

Thread **Cut**

These point styles can easily be created from the new menu *(shown to the right)*.



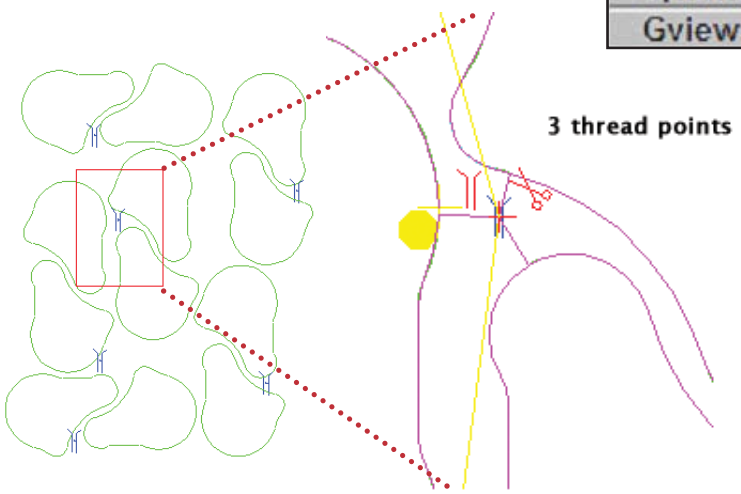
This example illustrates how to use multiple thread points, stacked on top of one another, to use the same position to thread three contours. This method will automatically pick up the same thread position for each of the contours and only cut the wire once all three contours are completed.

In this illustration, there are several new "thread" style points that will be used to thread all twelve contours. For this example, we will focus on the highlighted point to the right. In the magnified view, you can see the toolpath automatically chose this thread position to enter all three of the closest contours.

To accomplish this:

- Window chain the entire set of contours and points.
- Have three "thread points" created in this one position that will apply to all three contours.

The end result with toolpath is below:



Mastercam and the Woodworking Industry

Mastercam Router was quietly introduced to the woodworking and plastics industries 18 months ago as a newcomer in a growing market. After 18 years in the metalworking industry, the opportunity to serve a new market in addition to our metalworking customers became an exciting new challenge for everyone at CNC Software.

The challenge with any new software product is trying to figure out what most people need and expect. Our router-specific features came from suggestions in the field from people that use Mastercam and CNC routers every day for a variety of different applications.

For example, the geometry nesting feature was well-received, but users told us they wanted to nest toolpaths as well. Aggregate head support, block drilling, and a Visual Basic interface were deemed as "must have" items for the second release of Mastercam Router. Router users wanted special software tools to create custom stair stringers and cabinet doors. Simple but useful tools like "laminate machining" where the cutter slides up or down in the Z axis during the cut so the entire cutter gets used rather than dulling one small portion of the cutter - all came from vocal customers who want to make their jobs more efficient. You've been kind enough to tell us what you need, and in turn we will listen and deliver.

Mastercam Router Version 8 sales were modest the first year, but steady growth with this new product and the newly released Version 9 Router promises to become an important part of our business, which reinforces our commitment to this product. Our decision to serve this market was based on research indicating that businesses that purchase CNC routers have needs that are a bit different from our average metalworking customer. But the core of our nearly 20 years of CAD/CAM development allowed us to start with a solid base upon which to develop. Our competent development staff, coupled with my interest in occasionally making sawdust and Paul Martin's (Router product manager) 20-some-odd years of finish carpentry and furniture building experience are proving to be the right ingredients to create a "bread and butter" software tool for every shop with a CNC router.

The router market feels reminiscent of the metalworking industry of almost 20 years ago when the majority of shops did not yet have CNC equipment but everyone was looking! The excitement of watching shops succeed in taking the step from manual equipment to their first CNC machine is clearly one of the rewards of a job in this business.

You probably know that the modern world has many software products from many different countries and electronic files via e-mail are replacing blueprints in many shops. You may want to import a complex art file from a customer across the ocean or import and break down a set of cabinets from a customer down the street. We know Mastercam is not the only CAD/CAM system in use, so we have designed it to accept and import as many file types as we think you will typically run into out in the field. The ability to understand and get along with as many software products as possible is a philosophy important to our success, and hopefully yours too! Enjoy using Mastercam and please know that we welcome your comments and suggestions!

Mark Summers
President, CNC Software Inc.

Your Vote Counts

We would like to thank all of you that took the time to cast your vote in the "printed vs. electronic" newsletter debate. We have compiled the results, but we don't feel that we have heard from a big enough percentage of our customer base to make an informed decision. We would like to give everyone one more chance to let their opinion be known. At the end of this quarter, we will come to a decision regarding the future of this newsletter and we will make an announcement in the second quarter 2003 issue.

If you haven't already taken the time to cast your vote on whether you would prefer to receive Mastercam Notes in printed or electronic format, please do so today. Simply log on to **www.mastercam.com/newsupdate** and answer three quick questions in the online survey. The most important part of this process is giving us your e-mail address. If we do decide to go with an electronic format instead of the printed version, it is crucial that we have your email address. If you enjoy receiving this newsletter and don't want to miss out on future issues, please take a moment to provide us with your information.

Trade Shows

Mastercam will be exhibited at the following trade shows. Brown type indicates woodworking shows. **Bold** type indicates a CNC Software corporate show. All other shows are supported by local Mastercam dealers. More show listings can be found on our Web page at www.mastercam.com.

Carolina's Expo
February 20-21, 2003
Greensboro Coliseum
Greensboro, NC
Booth #2210-2211

NDES
March 3-6, 2003
McCormick Place
Chicago, IL
Booth #6514

ITEA (Educational)
March 13-15, 2003
Nashville, TN
Booth #801

Westec
March 24-27, 2003
Los Angeles Conv. Ctr.
Los Angeles, CA
Booth #3364

Mid Atlantic Job Shop Show
March 25-27, 2003
South Jersey Expo Ctr.
Cherry Hill, NJ
Booth #N19 & G14

Solid Modelling 2003
March 26-27, 2003
Birmingham, UK
Stand #76

Fort Wayne SME
April 16, 2003
Allen County Memorial Coliseum
Fort Wayne, IN
Booth #324-325

AMTDA Indianapolis Tool Show
April 29-30, 2003
Indianapolis, IN
Booth #420

Moldmaking Expo
April 30 - May 1, 2003
International Expo Ctr.
Cleveland, OH
Booth #407 & 409

Rapid Prototyping
May 13-15, 2003
Hyatt Regency
Chicago Riverside Ctr.
Chicago, IL
Booth #1109

Eastec
May 20-22, 2003
Eastern States Expo
Better Living Center
W. Springfield, MA
Booth #5231

LIGNA - World Fair for Forestry & Wood
May 26-30, 2003
Hannover Germany

Atlantic Design Show
June 2-4, 2003
Jacob Javits Conv. Ctr.
New York, NY
Booth #1233

NPE - National Plastics Expo
June 23-27, 2003
McCormick Place
Chicago, IL
Booth #11008

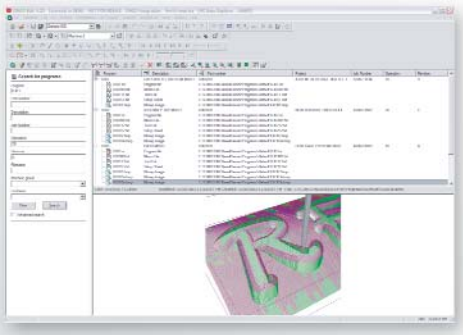
AWFS
July 31- Aug. 3, 2003
Anaheim Conv. Ctr
Anaheim, CA
Booth #1263

Mastercam
CD-ROM Training
Design with Solids
Lathe
FMMT
Mill
Why use FMMT?
- Tutorial lessons, step-by-step examples and detailed diagrams clarify Mastercam's features for the new or experienced user.
- The user of FMMT is always in control of what to learn at any time. There is never a need to follow a specific sequence, and accessing any function is as simple as point and click.
- Training and help for Mastercam is available to you at all times at your fingertips.
- With the existence of Mastercam and FMMT on the same computer system, the user can view a specific example and then toggle to Mastercam to gain hands-on experience using the sample geometry files provided with the CD.
- FMMT runs independently of Mastercam and can be used on any computer system that meets the hardware requirements.

FMMT is available for Mastercam Version 9!

FASTech, Inc.
Findlay Advanced Systems Technologies
Visit our web site at www.fastechinc.net for a demonstration

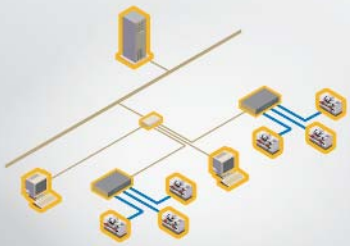
FASTech, Inc. 1750 Westfield Dr. Findlay, OH 45840 Tel. (419) 425-2233 Fax. (419) 425-9431
Mastercam Sales /Training / Technical Support / Multimedia / Business Card CD-ROMs / Consulting



NC-Base
GET ORGANIZED

Are you managing your CNC programs?
NC-Base is the only product designed specifically for CNC program management. Organize and manage part programs, setup sheets, tool lists, image files, and other production documents quickly and easily.

Available as a stand-alone product or integrated with DNC-Max!
From the makers of **CIMCO Edit** !



DNC-Max
GET CONNECTED

Looking for the right DNC solution?
Whether you are looking for a wired serial, wired Ethernet or wireless DNC solution CIMCO has best solutions available anywhere. From remote request and auto-receive to client/server and remote access, CIMCO has the best solution for your shop.

50 Leominster Rd., Suite 14, Sterling, MA 01564, USA
Phone (978) 728-0014, Fax (978) 728-0013

Virumvej 28, DK-2830, Virum, Denmark
Phone +45 45 85 60 50, Fax +45 45 85 60 53

You can rely on CIMCO as your single source for CNC editing, CNC program managment, and DNC solutions.

Visit our website for more information
www.cimco-integration.com

CIMCO Integration

ADVANCING INDUSTRY THROUGH TECHNOLOGY AND TRAINING

Since 1991, Prism Engineering has served as the Mid-Atlantic's predominant source for expert level technical support and award-winning training services for Mastercam.

Prism offers :

Mastercam 2D
Mastercam 3D
Mastercam Solids
Mastercam Lathe
Mastercam Adv. Lathe
Mastercam Wire
Mastercam 4-Axis Wire
Mastercam Router
Mastercam Post
Mastercam 5-Axis

For more information:
888.441.9696 or
www.prismeng.com

PRISM ENGINEERING

Mastercam Systems & Training Centers
Pennsylvania • New Jersey • Delaware
Maryland • Washington, DC

www.onlineatc.com

StepWISE for Mastercam

ATC

Stepwise for Mastercam

NOW AVAILABLE IN SPANISH

Mastercam Books and CD's

Online Training

The Origional step by step no nonsense inexpensive Tutorials, CD's and Online Training for every level of user from the novice to experienced professional

Advanced Technical Curriculum
P.O. Box 4085 Brandon, MS. 39047
Tel: (800) 443-2041 Fax: (601) 992-2162
www.onlineatc.com

WHAT'S NEW

Time Is Running Out

CNC Software's first quarter sales promotion is valid through March 31, 2003. This quarter we are offering the True Solids 5-Axis Verify option for Mastercam Mill at a reduced price. If you are using any type of Mastercam 5-axis toolpath, this is a valuable and necessary tool.

Mastercam's included basic 5-axis verification lets you verify your part from one view at a time, with no further ability to manipulate the verified model. With the added power of the True Solids 5-Axis Verify add-on, you can rotate and zoom in on your part at any point in the verification process. Orders must be received before the end of the business day March 31 to be honored at the promotional price. Call your Mastercam Reseller today for more information or to place an order.

The current sales promotion will always be posted in the "What's New" section on the Mastercam homepage. Check often so you don't miss out! www.mastercam.com

CNC Software, Inc.
National / International
671 Old Post Road
Tolland, CT 06084
(860) 875-5006
Fax (860) 872-1565
BBS (860) 875-9189
www.mastercam.com

Educational Division
5717 Wollochet Dr. NW
Suite 2A
Gig Harbor, WA 98335
(253) 858-6677
Fax (253) 858-6737
www.mastercamedu.com

Mastercam is a registered trademark of CNC Software, Inc.
Windows/Windows NT is a registered trademark of Microsoft Corp.

cnc software, inc.
671 Old Post Road
Tolland, CT 06084

Change service requested

Prst. Standard Mail
US Postage
PAID
Hartford CT
Permit 1754

Mastercam Notes
is published quarterly. Please write, e-mail, phone or fax us your comments or questions.
Contact: Paula Apro, Editor
mcnotes@mastercam.com

Printed on recycled paper