

Mastercam X³ Mill



Also available Mastercam Router, Mastercam Art, Mastercam Wire and Mastercam Lathe


Fast. Flexible. Reliable.

X3

CNC Software, Inc. is committed to bringing environmentally friendly and socially responsible practices to our industry.

The brochure you are reading has been condensed to decrease paper consumption. These are some of Mastercam's most important highlights, but there's much more information online.

Throughout the brochure, you'll notice some special indicators.

 = Video available online that will give you more details.

Blue text = More descriptive information linked in the electronic brochure.

For complete information on Mastercam Mill, visit www.MastercamMill.com.

A Few Simple Clicks

Powerful Part Modeling

Mastercam's streamlined CAD engine makes design work easier than ever before. Each piece of geometry you create is "live," letting you quickly make modifications until it's exactly what you want. Some of Mastercam's modeling tools include:

- Easy 2D and 3D geometry creation with complete wireframe and surface modeling.
- Remove trim boundaries and fill trimmed holes.
- Automatic parting line calculation for mold making.
- Associative dimensions update as you change your model.
- Solid modeling is available as an optional add-in.
- Built-in data translators for IGES, Parasolid®, SAT (ACIS solids), AutoCAD® (DXF, DWG, and Inventor™ files), SolidWorks® (including history tree), Solid Edge®, STEP, EPS, CADL, STL, and more. Direct translators for CATIA®, Pro/E, and more are also available.
- Special free Mastercam Direct add-ins put a Mastercam launch button in your SolidWorks, Solid Edge, or AutoCAD Inventor toolbars.

Intelligent Machining

As the world's most widely-used CAM software*, Mastercam is dedicated to making your entire process easier from start to finish. Here are just a few of the things Mastercam offers to help you make the most of your time.

Capture Your Machining Knowledge

Mastercam's full associativity gives you the power to capture your work and build on your experience. Once you program a part – no matter how complex – you can modify any element of the job, and immediately get updated toolpaths without starting over. You can also build a library of your favorite machining strategies. Choose any of your saved operations, apply them to a part, and Mastercam helps adapt them to the new model. It's fast, easy, and productive – the way programming should be.

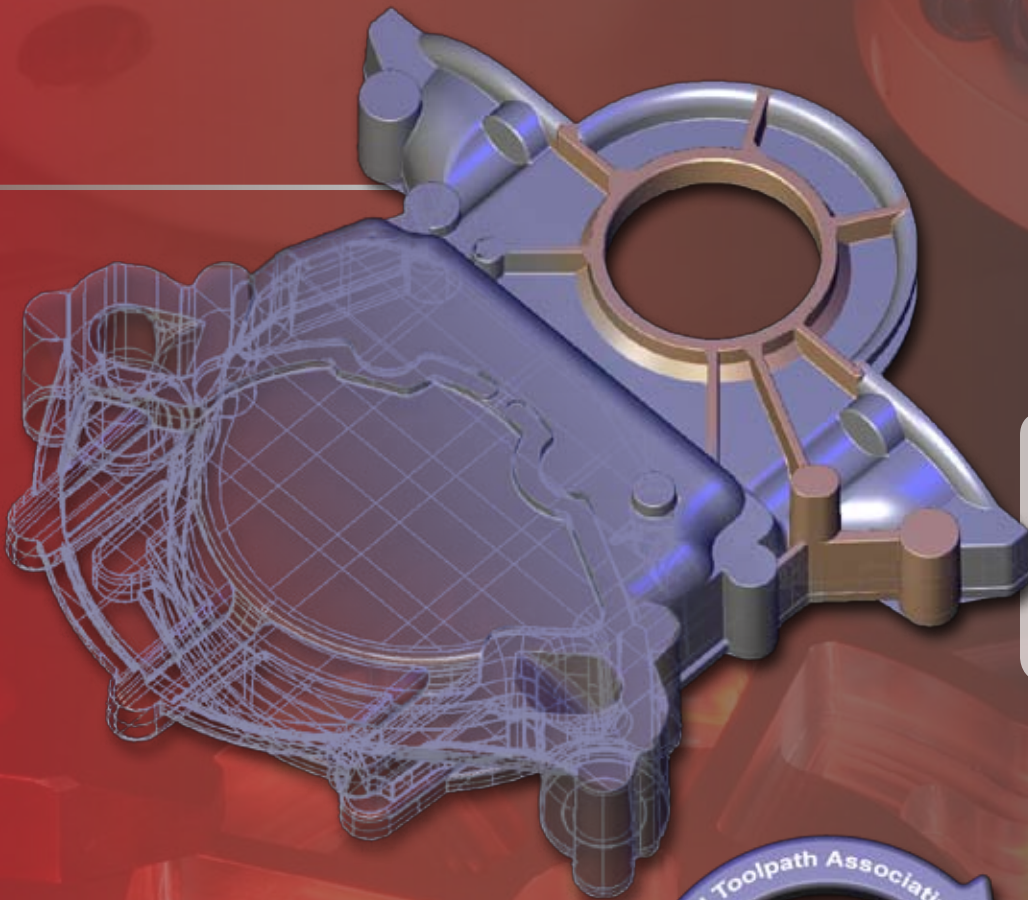
CAD File Change Recognition and Toolpath Updating

Mastercam's File Tracking and Change Recognition give you an easy way to identify CAD edits and updated cutter paths. Create a checklist of files including Mastercam, SolidWorks, AutoCAD, or other CAD files. Mastercam alerts you when new versions of the files are available and identifies any changes. With a few mouse clicks, you can immediately locate and program these changes, saving valuable time. Plus, you will always be sure your CAM program is based on the latest version of the CAD model.

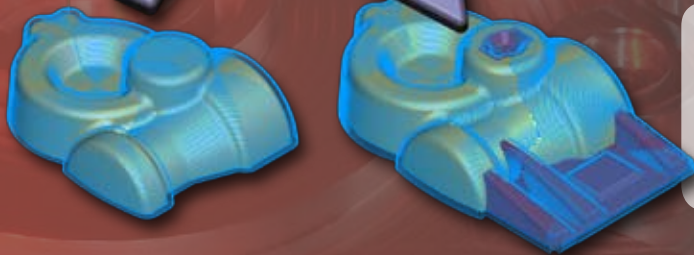
Dependable Toolpath Verification

Knowing your results before committing tool to material is crucial. Mastercam gives you several ways to ensure that your part will come off the machine exactly as you intended. These include solid-model verification with tool and holder checking, along with toolpath backplotting. These tools deliver vital information about every aspect of the tool motion so you know for sure that what you see is what you get.

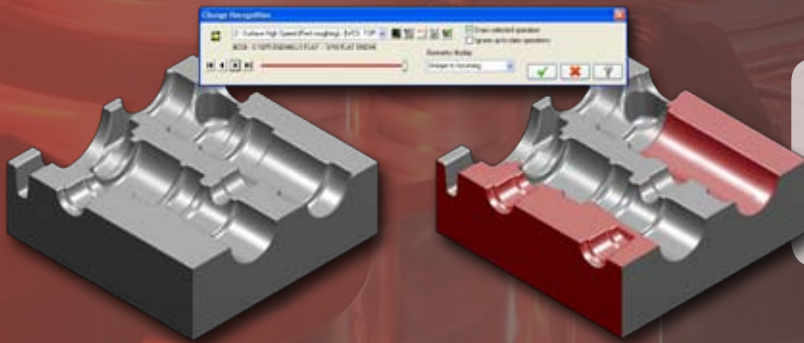
* Source: CIMdata, Inc.



Mastercam includes powerful wireframe and surface modeling tools, perfect for creating everything from simple to complex models. Mastercam also gives you easy tools for prepping incoming CAD files.



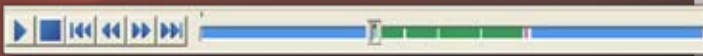
Mastercam creates a smart link between a part and its toolpaths. This associativity means that if there are any changes to the part, you can easily update the toolpath.



Mastercam's Change Recognition quickly identifies changes to a CAD file, allowing you to easily adjust the toolpath for the updated model.



"Safe zones" help ensure safe tool retracts in all toolpath types. This is especially helpful when used with machines with complex motion.



"Play" your program with toolpath backplotting and get an estimate of machining time. A quick color-coded slider bar lets you view operations at any stage.

X3



2D Toolpaths

Contouring, Drilling, and Pocketing

2D machining ranges from the very simple to the very complex. Mastercam delivers all the tools you need for these operations. Highlights include:

- **Feature Based Machining** (FBM) evaluates a solid model and automatically programs pockets, contours, bosses, and drill routines.
- New **2D high speed** toolpaths create smooth motion for longer tool life and better surface finish.
 - Standard **pocketing styles** include zigzag, one way, true spiral, constant overlap spiral, "morph" pocketing, and open pocketing, each with optional finish passes.
 - Choose plunge, helical, or ramp entry.
 - Contour and **pocket remachining** use smaller tools to automatically clean out material left from previous operations.
 - Ability to click and drag a machining start point to anywhere on your model.
 - Automated slot, circle, and thread milling.
 - Controlled engagement facing removes stock using a consistent tool load.
 - Automatic identification and pre-drilling of multiple operations at their plunge points.
 - Automatic drilling and countersink depth calculation.
 - Visit www.MastercamMill.com for a complete list.

Feature Based Machining automatically programs prismatic solids, saving a dramatic amount of programming time.

3D Toolpaths

Roughing, Finishing, and Cleanup Machining

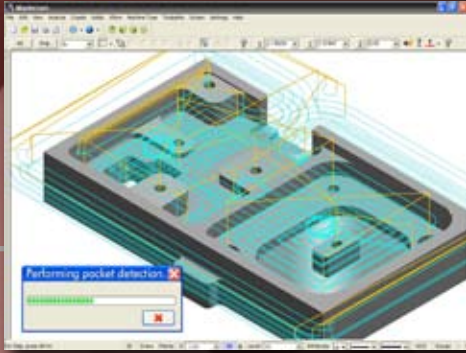
Operations that quickly deliver a clean and precise finished part are essential to efficient NC programming. Here are just a few of Mastercam's popular 3D machining techniques:

- Ability to cut multiple surfaces, solid models, or a combination of both. Rough cut with constant-Z contours or pockets.
- **Constant-Z rough "remachining"** identifies and machines areas that need to be roughed with a smaller tool.
- 3D "projected" machining creates a consistent, smooth finish while following the natural curves of the geometry.
- **Constant scallop machining** maintains a consistent finish on sloped and flat surfaces alike by using a consistent 3D stepover.
- Flowline machining cuts single or multiple surfaces using their natural shape to define the cutter path and deliver a smoother finish.
- Full check surface support.
- **Hybrid leftover machining** changes the cut method as the slope of the model changes – all within a single toolpath.
- **Constan-Z rest milling (remachining)** identifies and machines areas and critical depths that need to be cut with a smaller tool.
- **Pencil tracing** walks a tool along the intersection of surfaces to clean out hard-to-reach areas. You can perform single or multiple passes for precision cleanup.
- Go to www.MastercamMill.com for more.

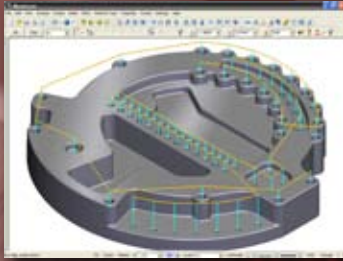


"Mastercam has been central to the success and growth of Roush Yates racing."

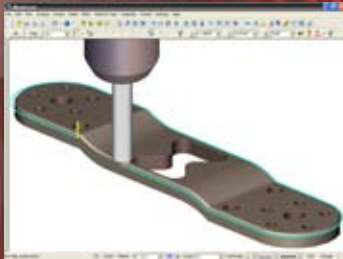
Doug Yates
President
Roush Yates Racing Engines



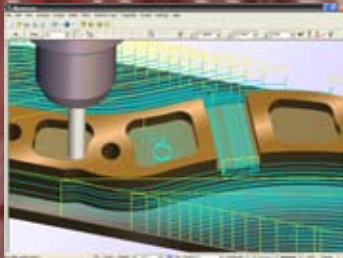
Mastercam's new Feature Based Machining makes it significantly easier to machine prismatic solids by automating the programming process.



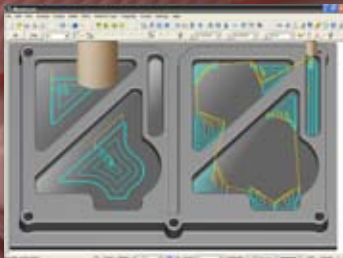
Fully optimized drilling.



Complete 2D and 3D contouring, with cleanup machining.



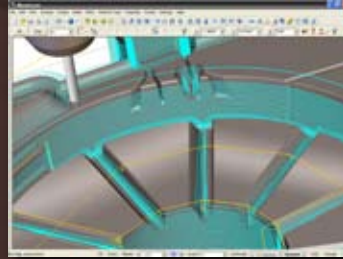
Mastercam's 2D high speed toolpaths deliver a finer finish and increased tool life.



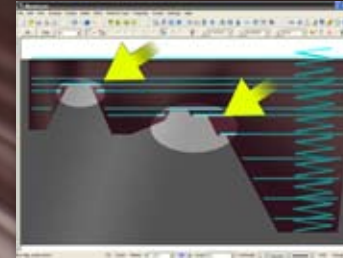
Automated pocket remachining with a smaller tool.



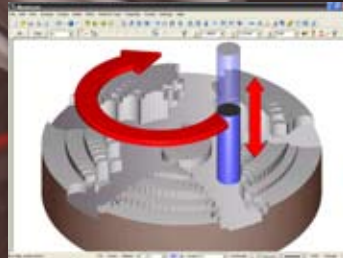
Easy tombstone programming.



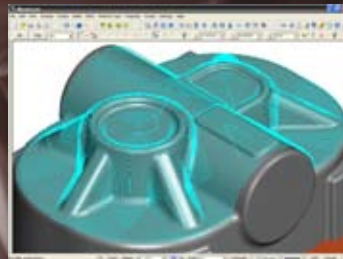
Automated rough remachining with a smaller tool.



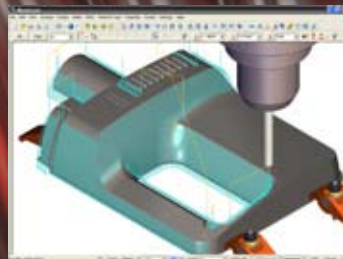
Automatic roughing of critical depths.



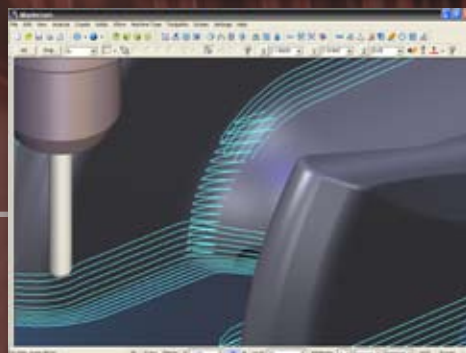
Advanced plunge roughing in any pattern.



"Unlimited Pencil Finishing" machines the entire part in logical segments.



Remachining stock cleanup in steep and shallow areas.



Hybrid remachining changes strategies in a single toolpath.



A Wide Range of Strategies

Multiaxis Machining


Multiaxis machining can dramatically increase a shop's competitiveness. Mastercam offers a wide range of multiaxis machining strategies. With Mastercam, you have complete control over the three crucial elements of multiaxis machining: toolpath types, tool motion, and tool axis.



"We use Mastercam's high speed machining to hit the sweet spot where machining is accurate, fast, and least abusive of the machine. We recently shaved 26 hours off three runs of a part, turning it from a low-profit to a high-profit job."

John Graney
Surface Solutions Inc
Ipswich, MA

Some highlights of Mastercam's multiaxis machining:

- **Multisurface 5-axis** roughing and finishing (including depth cuts), plunge roughing, and flowline machining and drilling.
- Swarf fanning and swarf machining over multisurface floors, plus "rail" swarf cutting for added control.
- **Machine 5-axis curves** with independent definitions of tool side angle and lead/lag angle.
- Create 5-axis contour toolpaths around surface edges for applications such as trimming vacuum-formed parts.
- Easy 4-axis rotary axis and roldie programming, and 5-axis drilling.
- Create full 5-axis motion from a 3-axis toolpath.
-  Special options for machining cylinder heads and converting probe data to machinable geometries.
- **Advanced gouge checking** and a 5-axis "safe zone" around the part.
- Complete control over the tool axis, lead/lag, entry/exit, and tilt. These simplify even the most difficult multiaxis jobs.
- See a complete list of multiaxis tools at www.MastercamMill.com.

Faster Turnaround and Superior Finish

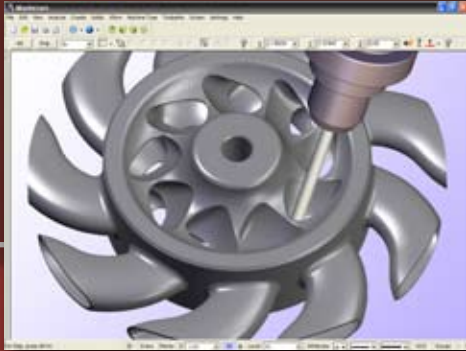
High Speed Machining/Feed Rate Optimization

High Speed Machining (HSM) is a powerful cutting method that combines high feed rates with high spindle speeds, specific tools, and specialized tool motions aimed at producing ultrasmooth movement and cutting action. HSM can deliver faster turnaround and a superior finish. Mastercam includes High Speed Machining toolpaths designed to help you make the most of this powerful technique, even if you don't have a high speed machine.

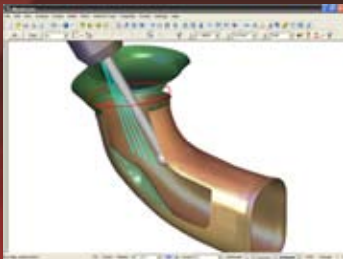
Mastercam also delivers another powerful way to get the most out of your machines. As every shop owner knows, running an entire job at a single feed rate reduces efficiency. Running the same job at varying optimal feed rates can save time and money, and reduces tool wear. Our **Feed Rate Optimization** feature enhances any 2-axis or 3-axis toolpath based on the volume of material being removed and machine tool limitations – more material and the cutter moves more slowly; less material and the cutter moves faster. Feed Rate Optimization will also automatically ease the tool in and out of corners. The result is efficient, varied feed rates tailored to each job.

*Feed rate optimization
can save up to 35% in
machining time over
non-optimized toolpaths.*

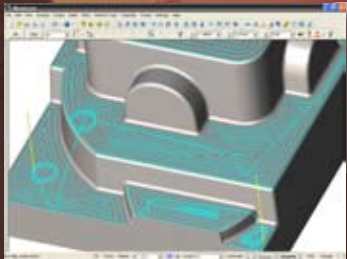




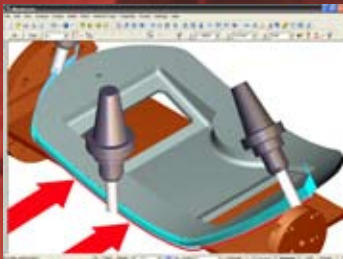
Efficient multiaxis roughing and finishing help ensure accurate cuts and short turnarounds.



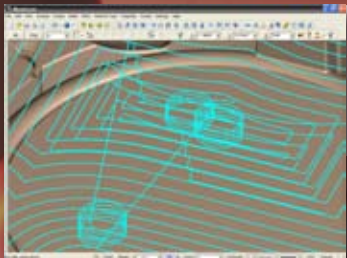
Shank containment simplifies working in confined spaces.



Automatically machine flat areas using new time-saving minimum retracts and smooth entry, exit, and cut motion.



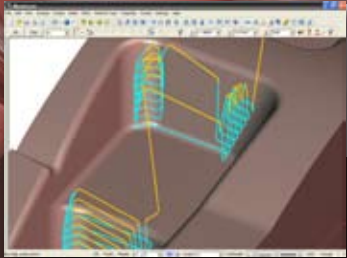
"Rail" swarf cutting lets you control the cut using a lower rail.



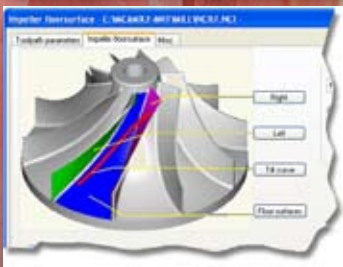
High speed area clearance removes bulk material from the inside out with smooth motion.



Specialized toolpaths support fast, efficient 5-axis drilling.



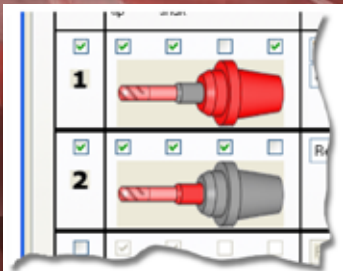
High speed rest roughing smoothly removes material left from a previous rough pass.



Mastercam's advanced multiaxis machining lets you choose what type of project you are doing and adjusts the interface to show only what you need.



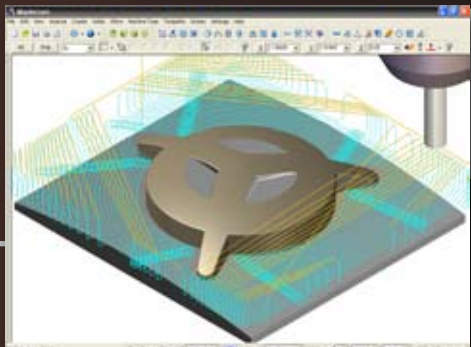
Constant volume removal: Mastercam slows the feed rate through deep material and speeds it up through more shallow stock.



Advanced gouge checking helps ensure safe cuts in even the most complex operations.



Smart cornering: Mastercam adjusts the feed rate around corners and in small radii for smooth transitions in tight areas.




Smart core roughing cuts from the outside-in. If it encounters a pocket, it automatically switches to inside-out.

Specialized Options

Very often, that little something extra – that one additional CAD or CAM tool – makes a specific job easier, faster, and more profitable. Mastercam offers a set of specialized add-in options for these occasions including:

- Automatic separation of surface model into core and cavity, including draft angle analysis and identification of problem surfaces.
- Use of point data to create surfaces or STL data for reverse engineering and manufacturing.
- Sophisticated tools for traditional blueprint and CAD – based inspection.
- Automated complex shutoff and parting surface creation.
- Automated EDM electrode creation, including a library of definable stock sizes and materials.
- See a full list at www.mastercam.com/Products/Addins.



Other powerful CAD/CAM packages available from Mastercam:

- Mastercam Lathe**
Fast, flexible CNC turning
- Mastercam Wire**
2-axis and 4-axis wire EDM programming
- Mastercam Router**
Quick and easy CNC routing
- Mastercam in SolidWorks**
NC programming within SolidWorks
- Mastercam Art**
Turn flat line art into artistic 3D work
- Mastercam Solids**
Powerful Parasolid®-based part modeling

System Requirements

- Processor: 1.5 GHz (minimum) 32-bit Intel®-compatible processor; 64-bit Intel-compatible processors are supported.
- Operating System: Windows XP, Windows Vista® (Business or Ultimate) including the latest service packs and recommended updates; .NET 2.0 framework.
- Memory: 1 GB minimum (2 GB if using Windows Vista), 1.7 GB available hard disk space.
- Graphics: 128 MB OpenGL-compatible graphics card (minimum).
- Monitor: 1024 x 768 at 96 DPI (minimum).
- Mouse: Windows-compatible 2-button or 3-button mouse (or with middle mouse wheel).
- CD or CD/DVD drive.

	Mastercam Mill	Level 1	Level 2	Level 3
CAD				
Complete customizability		x	x	x
Create and dimension live wireframe geometry		x	x	x
Read / write IGES, DXF, SAT, Parasolid, EPS		x	x	x
Read native AutoCAD, SolidWorks, Solid Edge		x	x	x
Read native CATIA, Pro/E, Unigraphics (UG NX)		Optional	Optional	Optional
Live surface modeling		x	x	x
CAD file change recognition		x	x	x
Solid modeling		Optional	Optional	Optional
CAM				
Feature Based Machining		x	x	x
Fully associative toolpaths		x	x	x
Automated feed rate optimization		x	x	x
Contouring, pocketing, and drilling		x	x	x
2D high speed machining		x	x	x
3D contour cutting, trimming and remachining		x	x	x
On-screen toolpath verification		x	x	x
Safety zones for all toolpaths		x	x	x
Machine and control definition		x	x	x
Engraving		x	x	x
Single and limited multisurface roughing			x	x
Single and limited multisurface finishing			x	x
Full multisurface and solid roughing				x
Full multisurface and solid finishing				x
Full multisurface and solid "cleanup" machining				x
Full multisurface and solid high speed machining				x
5-axis drilling and curve machining		Optional	Optional	Optional
Simultaneous 4-and 5-axis machining				Optional

Mastercam
cnc software, inc.

671 Old Post Road Tolland, CT 06084 USA
(800) 228-2877 • (860) 875-5006 • Fax (860) 872-1565
www.mastercam.com • mcinfo@mastercam.com