



Mastercam X² Lathe



Fast. Flexible. Reliable.

You've Got The Power



Mastercam X2 Lathe – Fast. Flexible. Reliable.

A friendly familiarity

As you open Mastercam, there's an immediate comfort level with the look of the interface. Plus, we provide a set of integrated guides to help you get down to business creating toolpaths and turning parts as quickly as possible. For instance, you can engage a learning mode designed for new users. And beyond the ease and comfort of the Mastercam interface is almost unlimited flexibility. You can customize its appearance and functionality. Even your keyboard and mouse shortcuts can be tailored to your personal preferences.

Do more with fewer clicks

We understand that saving programming time can get you turning parts faster, boosting productivity and your company's bottom line. There are numerous programming efficiencies designed into Mastercam. Multiple functions are consolidated within a single option, streamlining the number of programmer interactions. Mastercam's new smart Ribbon Bar appears only when needed, putting crucial tools at your fingertips at just the right moment.

Also with Mastercam, geometry you create remains "live" until you move on to the next task. You can modify its data points, adjust attributes, even change entity types without affecting other entities in your part until you're completely satisfied.

Reliable turning toolpaths

At CNC Software, we have built our reputation for more than 20 years on creating the highest quality, most robust toolpath generation system in the world. The primary benefit that users continually cite about Mastercam is that they have complete control over all aspects of tool motion. Delivering comprehensive, clear tool control is crucial to programmers and one of many strengths that sets us apart from the competition. With Mastercam X2, we continue our tradition of quick, powerful turning toolpaths, coupled with a streamlined operation that will simplify even the most complex jobs.

Plays well with others

Mastercam can use virtually any CAD source file in the world including SolidWorks®, Solid Edge®, AutoCAD® products, CATIA®, Pro/E®, and many more. And even better, it tracks new versions of these files, recognizes changes, and helps you update the toolpaths. In addition Mastercam is designed with an "open" architecture, allowing specialists around the world to create unique add-ins tailored for specific needs.

Local experts, local support

Our company has the longest tenured Reseller channel in the business, which is a tremendous advantage for our customers. As a Mastercam user, you have an exclusive Mastercam partner who is highly trained and experienced, ready to pass on his or her knowledge and to support you in every way possible. No other CAM software company has garnered such loyalty and strength. Our customers often remark about the significant role their Reseller has played in helping them achieve success and profitability with Mastercam.

The power of X!

The Mastercam X family of CAD/CAM takes another leap forward with Mastercam X2, bringing powerful new techniques and faster programming to your shop.

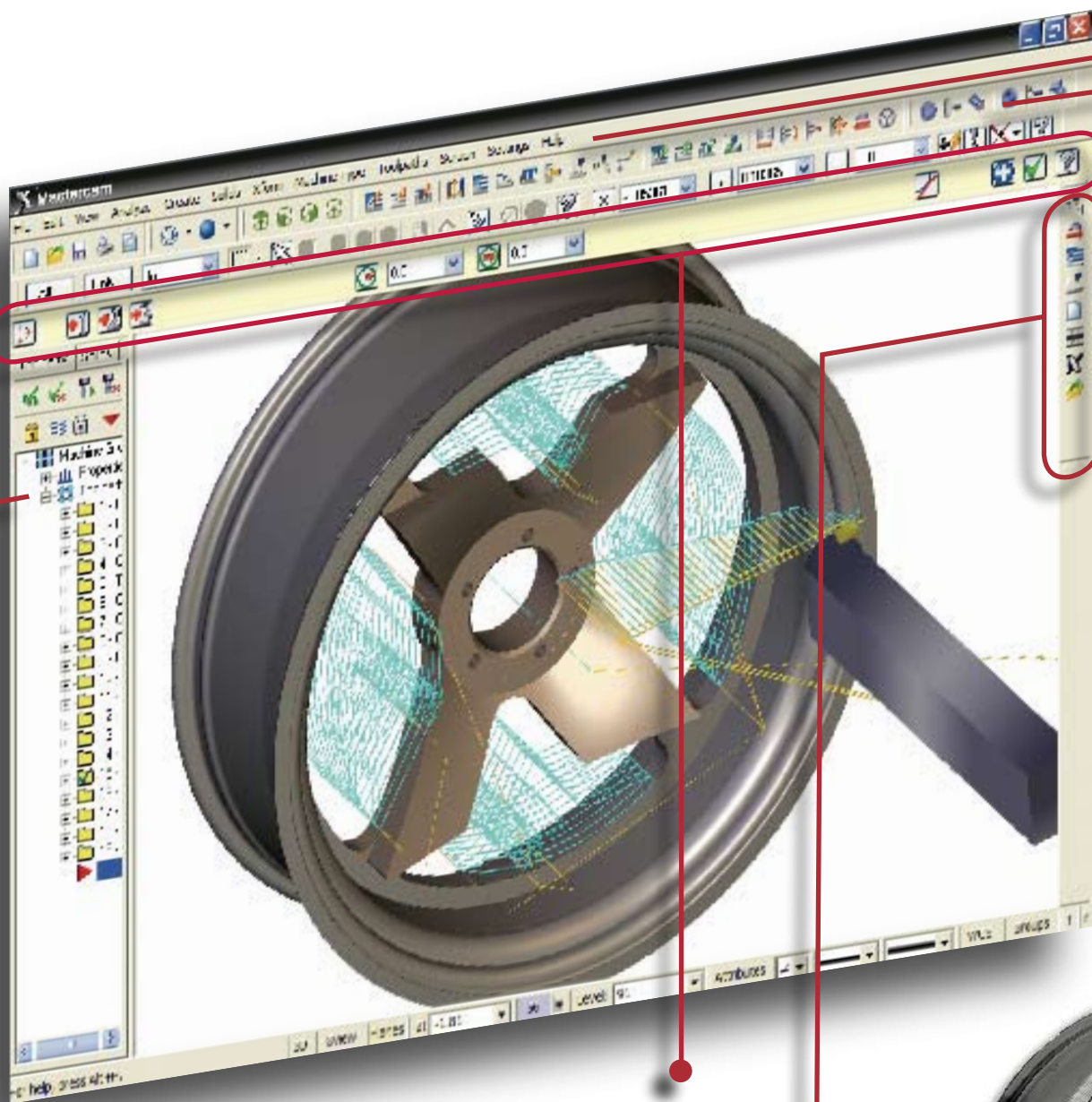
Mastercam X2.

CAD/CAM for the most important job – yours.



We established CNC Software, Inc. in 1983 to give shops a smarter, easier, offline method to program CNC machines. Today, Mastercam is the most commonly used CAM software worldwide* and remains the program of choice among CNC programmers. When you invest in Mastercam, you get the best in CNC programming backed by the support and service of today's leading developer and our worldwide certified Resellers. Mastercam X2 represents the next generation of our popular Mastercam X CAD/CAM software, delivering the most comprehensive turning package with powerful new toolpaths and techniques.

* Source: CIMdata, Inc.



Quick access to basic functions gets you familiar with the toolbar icons quickly – and lets you choose from a familiar text menu.

Fully customizable interface gives you instant access to the tools you need most.

Active Operations Manager gives dynamic access to all toolpath and solid model histories.

Smart Ribbon Bar automatically changes based on what you are doing.

Quick access history keeps your most recently used functions at your fingertips.



CAD and CAM

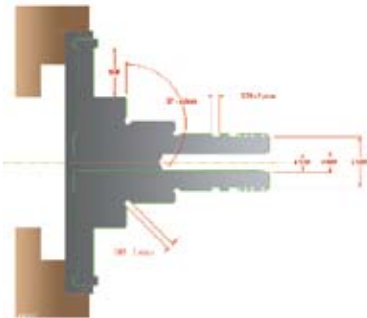


CAD Functions – A few simple clicks



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 modify it until it's exactly what you want. Further, with traditional functions consolidated into a few simple clicks, Mastercam simplifies the creation of even the most complex parts.



- Easy 2D and 3D wireframe and surface modeling
- Extensive editing and analysis tools ensure your model is exact.
- Associative dimensions update as you change your model.
- Customizable AutoCursor™ snaps to commonly used construction points.
- Simplified and improved work coordinate system makes it easy to work on a model without having to move it in 3D space. Your new planes and origins are easily transferred to your G-code.
- Built-in data translators for IGES, Parasolid®, SAT (ACIS solids), AutoCAD® (DXF, DWG, and Inventor™ files), SolidWorks®, Solid Edge®, STEP, EPS, STL, and more are built in. Direct translators for CATIA®, Pro/E, and more are also available.
- Special no-charge Mastercam Direct add-ins put Mastercam in your SolidWorks, SolidEdge, or AutoCAD Inventor toolbars.
- Solid modeling is available as an optional add-in.



CAD file change recognition & toolpath updating

Every shop faces engineering changes. Often, it can be difficult to track which file is the latest, what modifications have been made, and what programming changes are needed.

Mastercam X2 includes two powerful new tools – File Tracking and Change Recognition – that give you an easy way to identify these issues and streamline programming an updated file.

- Create a checklist of files to watch including Mastercam, SolidWorks, AutoCAD, or other CAD files. Mastercam alerts you when a newer version of any file on the list is available.
- Mastercam's Change Recognition identifies modifications to any type of CAD geometry with no file type limitations – any combination of wireframe, solids, or surfaces.
- Mastercam marks any toolpaths affected by the updated CAD file. You can quickly select the new geometry and update the toolpaths.
- Once you update an affected toolpath, automatically update any additional toolpaths that use that geometry.

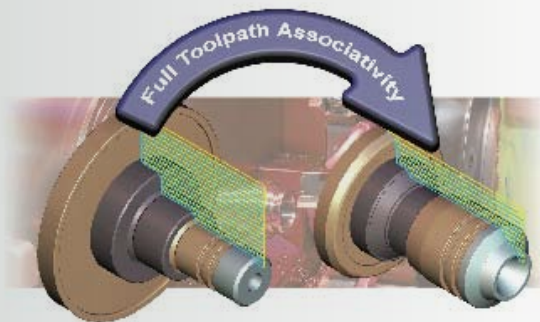


Now, with a few mouse clicks, you can immediately locate and program CAD file changes, saving valuable time.



Capture Your Machining Knowledge

Mastercam's full associativity gives you the power to capture your knowledge and build on your experience. Once you program a part – any part – you can modify any element of the job and immediately get updated toolpaths without starting over. And Mastercam's intelligent NC programming lets you build a library of machining strategies done the way you want them. Just choose saved operations, apply them to a part, and Mastercam adapts them to the new model. Fast, easy, and productive. The way programming should be.



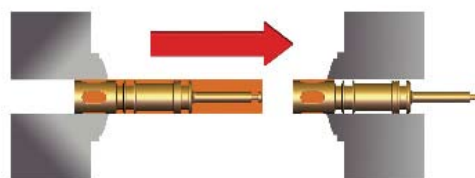
CAM Tools

Extensive Libraries

- Comprehensive insert libraries including Sandvik, Kenametal®, Iscar, and Valenite®.
- Extensive tool and tool holder libraries.
- Feeds and speeds calculated automatically based on user-customizable material and insert libraries.

Practical Tools for Full Control

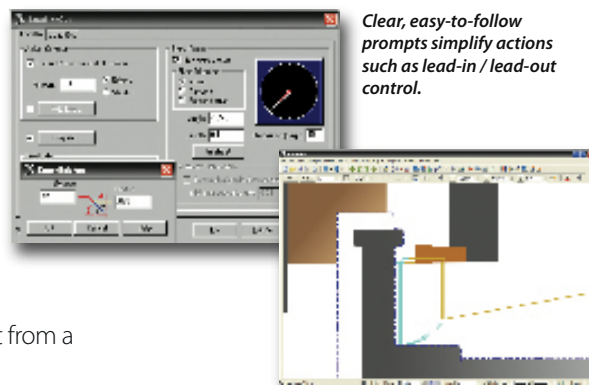
- Mastercam's Operations Manager gives you a single place to create, modify, and analyze all aspects of your toolpaths.
- Complete control over the cutting tool's approach and retract.
- Fast operations for manual part flip, stock transfer, and stock advance.
- Canned cycle and subprogram support.
- Significantly reduce the size of a program with toolpath filtering.
- Custom tool geometry can be stored with the part file for easy retrieval.
- Automatically generate customizable setup sheets.



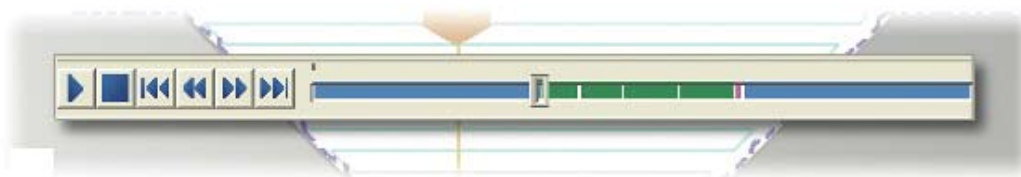
Fast, easy stock transfer or part flip.

Dependable Toolpath Verification

- Quickly check toolpaths with a cross-section of the part in progress.
- View or save a 3D model of your part at any machining stage.
- "Play" the toolpath, measure your part, and view vital information as it is cut from a solid block of material with Mastercam's solid model toolpath verification.



Clear, easy-to-follow prompts simplify actions such as lead-in / lead-out control.



View your toolpaths with a quick color-coded slider bar that lets you view operations at any stage.



Define Your Shop

Mastercam's Machine and Control Definition delivers a new way to make sure that your parts are done right the first time.

- Define features and characteristics for all the machines in your shop.
- Mastercam allows you to program only within the chosen machine's capabilities.
- Drag programmed toolpaths from one machine to another. Mastercam checks that they will run on the new machine, automatically making adjustments where possible.
- Mastercam tailors the interface to your selected machine's capabilities, and will even automatically load the toolbars that make you most efficient.

Toolpaths



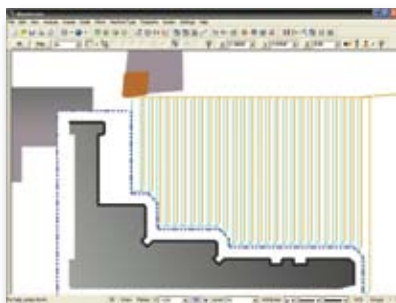
Turning toolpaths – Powerful Lathe Programming

Fast, Easy, and Precise Turning

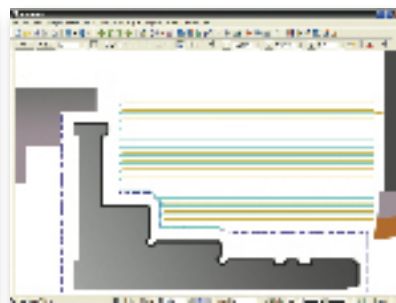
- Click and drag a machining start point anywhere on your model.
- Intelligent roughing of inside and outside diameters, including roughing to a boundary for castings.
- Optimized facing including roughing and finishing.
- Finish contouring (profiling).
- Automatically create a finish pass after a rough operation using the same tool.
- Automatically break corners during finish passes without changing your model.
- Set feed rates for desired finish quality.
- Grooving with multiple depth cuts and pecking.
- Groove from a point or multiple points without creating geometry.
- Easily define groove depth, width, wall angles, corner radius, and chamfer.
- Skip over a groove or cavity when roughing or finishing without creating additional geometry.
- Complete threading with multiple starts, diameter calculation, and thread tables.
- Boring, drilling, and point-to-point machining.
- Automatic gouge checking watches the front and back of the tool.
- Chuck, part, steady rest, and tailstock detection.
- Open and directly machine a solid model.
- Powerful tool angle control for machines with B-axis tooling arms.
- Mastercam's "Quick Toolpaths" let you program parts completely with just a few simple clicks.



Roughing



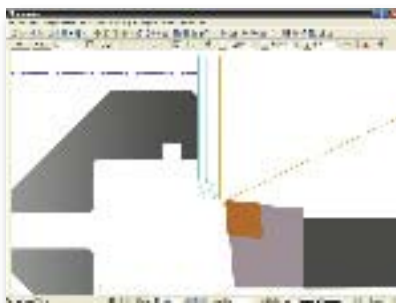
Downward roughing.



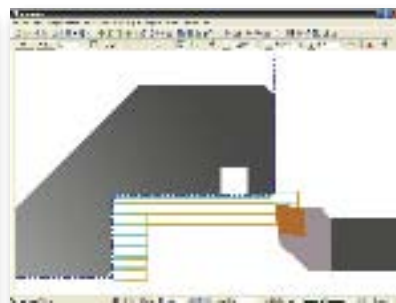
"no plunge" roughing.



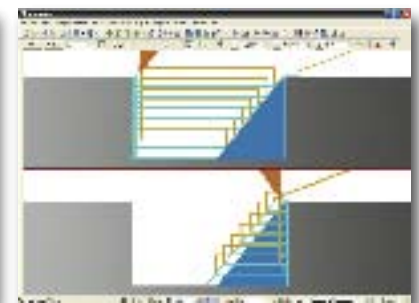
Roughing to an outer boundary (casting).



Automated facing from the center line out.



ID Roughing.



Stock recognition knows what is left after each operation.



C-axis (Mill/Turn) Programming

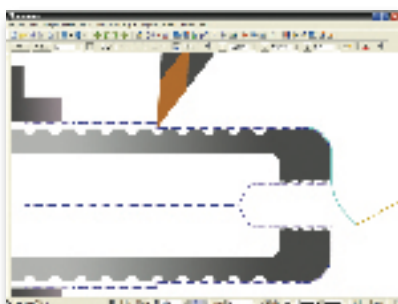
- Basic C-axis programming in Mastercam Lathe, with expanded options when combined with Mastercam Mill.
- Face and Cross contour.
- Face and Cross drilling including the ability to sort points in a clockwise or counterclockwise sweep (rotary) direction.
- Automatically sets tool plane and construction plane for face/cross contour and face/cross drilling.
- Machine C-axis contours with 3D or flat geometry.



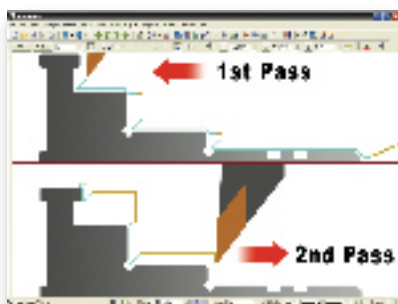
Full C-axis machining when combined with Mastercam Mill.



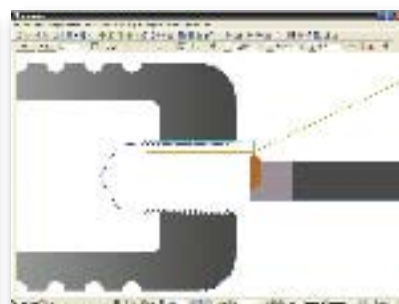
Finishing



Finishing with automated "no plunge" option.



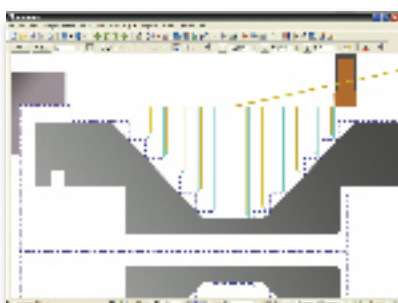
Multiple contouring for complete finishing.



Fast thread machining.



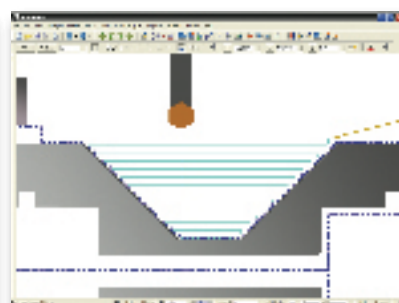
Grooving



Step grooving.



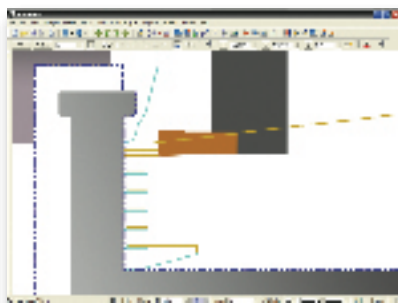
Stepless grooving.



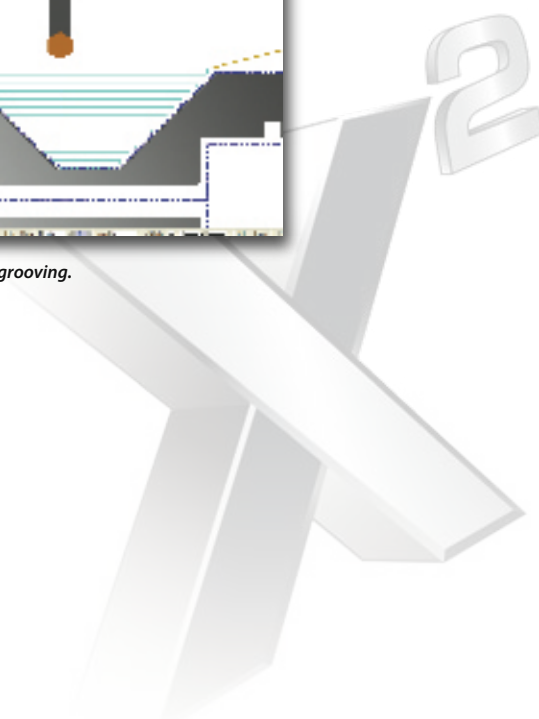
Zigzag rough grooving.



Multiple grooving styles, including angular.



Face grooving.



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’s “open” architecture allows specialists around the world to create a wide variety of specialized add-in options for these occasions. Ask your Mastercam Reseller for details.



Other powerful CAD/CAM packages available from CNC Software:

Mastercam Mill

Industry leading milling package

Mastercam Wire

2-axis and 4-axis wire EDM programming

Mastercam Router

Quick and easy router cutting

Mastercam Art

Turn flat line art into artistic 3D work

Mastercam Solids

Powerful Parasolid® based part modeling

System Requirements

- **Processor:** 32 bit, 1.5 GHz **minimum** Intel compatible processor (64 bit Intel compatible processors are supported).
- **Operating System:** Windows® XP, Windows® XP Pro 64 bit edition or Windows 2000 with latest service packs and updates.
- **.NET 2.0** framework.
- **Memory:** 512 MB RAM, 1 GB available hard disk space.
- **Graphics Card:** 64 MB OpenGL-compatible (minimum).
- **Monitor:** 1024 x 768 resolution (minimum).
- **Mouse:** Windows® compatible mouse.

	Lathe	When combined with Mastercam Mill Level 1	When combined with Mastercam Mill Level 3
CAD			
Complete customizability	x	x	x
Quickly create and dimension live wireframe geometry	x	x	x
Read / write IGES, DXF, SAT Parasolid & more	x	x	x
Live surface modeling	x	x	x
Solid modeling	optional	optional	optional
CAM			
Streamlined “Quick Toolpath” programming	x	x	x
Intelligent ID and OD roughing	x	x	x
Easy finishing	x	x	x
Optimized facing	x	x	x
Grooving	x	x	x
Threading	x	x	x
Canned cycle support	x	x	x
Automatic gouge checking	x	x	x
Boring, drilling, and point-to-point machining	x	x	x
Machine directly on a solid	x	x	x
Chuck, part, steady rest, and tailstock collision check	x	x	x
Fully associative toolpaths	x	x	x
Machine and control definition	x	x	x
Face and cross contour	x	x	x
Face and cross drilling	x	x	x
3D C-axis contour machining	x	x	x
All pocketing routines		x	x
All contour routines		x	x
All drilling routines		x	x
Full multisurface and solid roughing			x
Full multisurface and solid finishing			x
Full multisurface and solid “cleanup” machining			x
High speed toolpaths			x
Multiaxis machining			optional
Engraving		optional	optional

Mastercam.

cnc software, inc.

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