

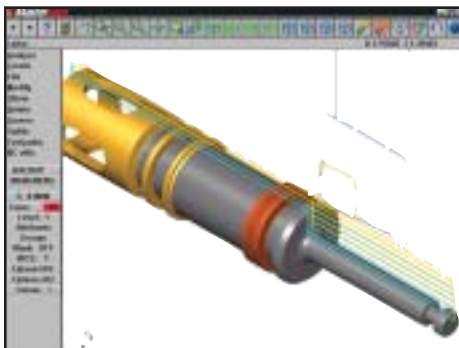
# Mastercam<sup>®</sup>

## Lathe



### 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 and apply them to a part, and Mastercam adapts them to the new model. Fast, easy and productive. The way programming should be.



### Logical Part Modeling

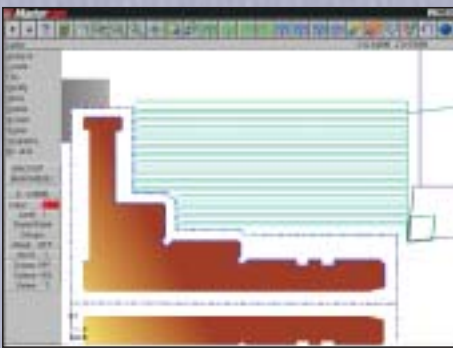
- Easy 2D and 3D geometry creation with complete wireframe & surface modeling.
- Extensive editing and analysis tools ensure your model is exact.
- Associative dimensions update as you change your model.
- AutoCursor™ snaps to commonly used points to simplify construction.
- New Work Coordinate System makes it easy to reorient your View, Construction, and Tool planes, allowing you to work on a model without having to move it in space. These new planes 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®, CADL, STL, VDA, and ASCII. Direct translators for STEP, Catia®, and Pro-E® are also available.

### Designed for Ease

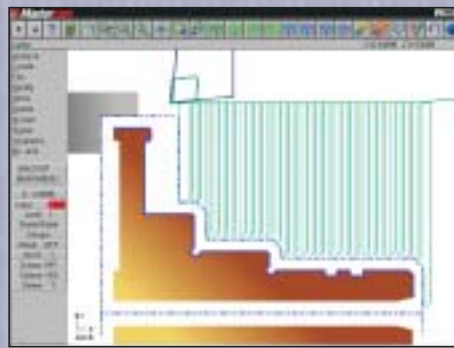
- Fully associative geometry and toolpaths let you modify the geometry, tooling information or machining parameters and immediately get an accurate, updated toolpath.
- Mastercam's "Quick Toolpaths" let you program parts completely with just a few simple clicks.
- Mastercam's Operations Manager gives you a single place to create, modify and analyze all aspects of your toolpaths.
- Store a library of commonly used operations to automate machining. For example, you can rough and finish a new part simply by importing one stored set of operations and applying them to a new model.
- Tool auto-selection displays appropriate tools or allows full access to Mastercam's tool library.

# Software that works the quick, and with an eye

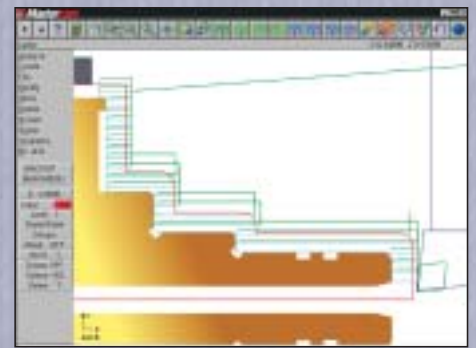
Here are just some of the toolp



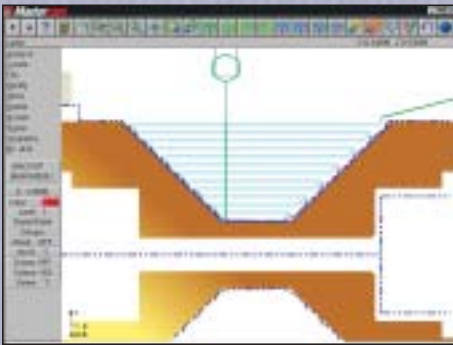
"No Plunge" roughing



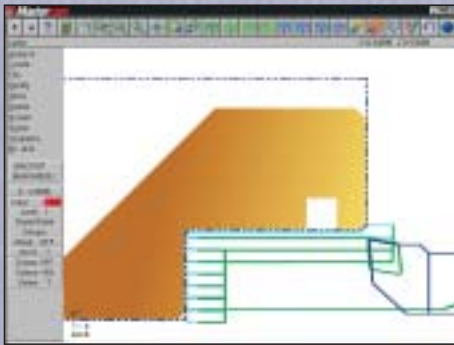
Downward roughing



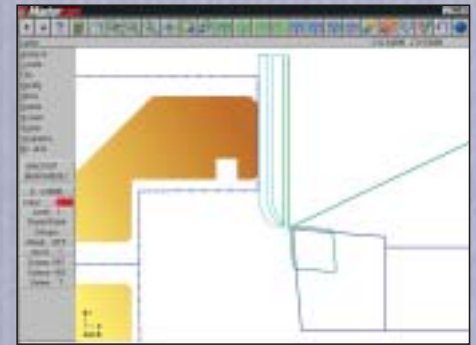
Roughing to an outer boundary (casting)



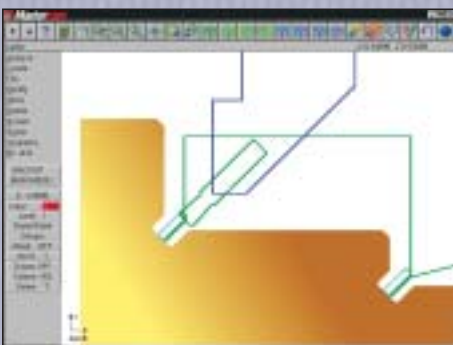
Zigzag Roughing



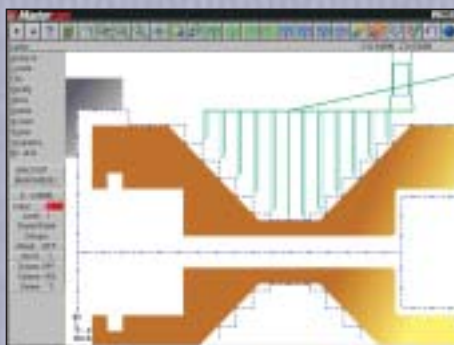
ID Roughing



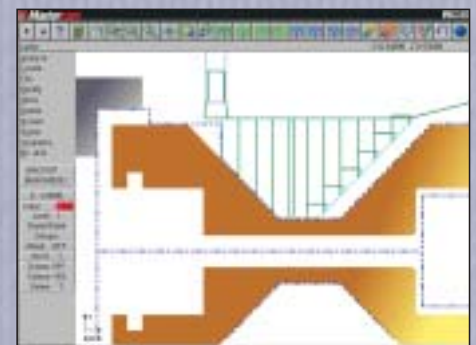
Automated facing from the center line out



Multiple grooving styles, including angular



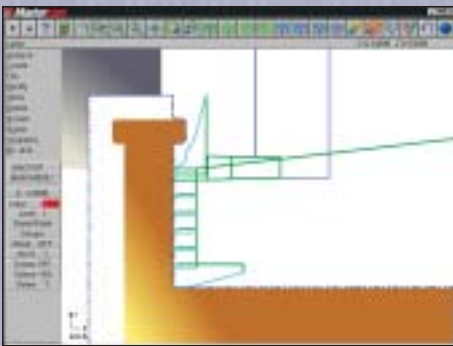
Step grooving



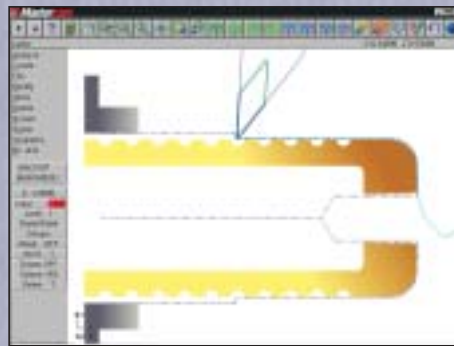
Stepless grooving

# way you think — clear, ve on the bottom line.

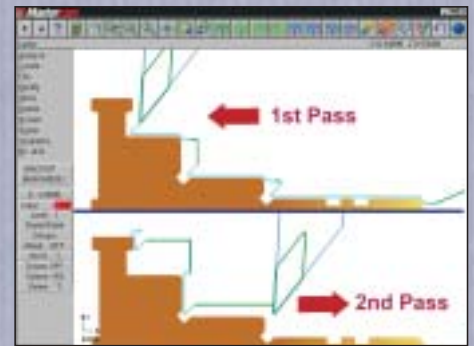
aths that Mastercam offers . . .



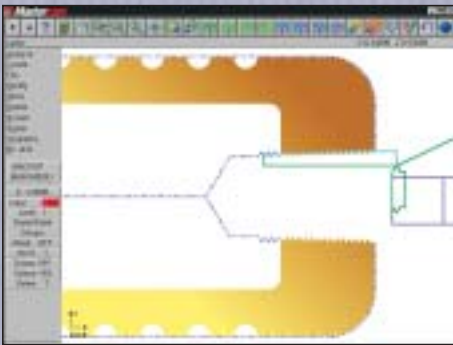
Face grooving



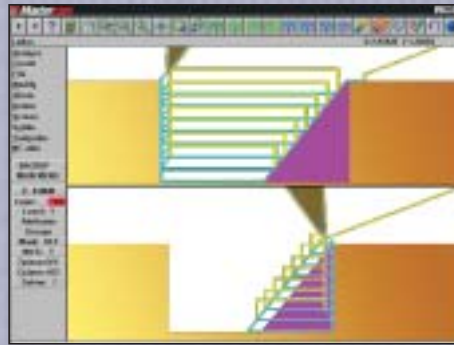
Finishing with automated "no plunge" option



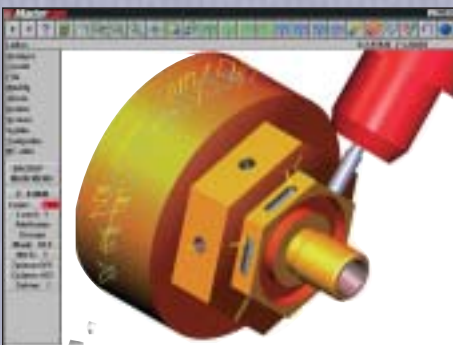
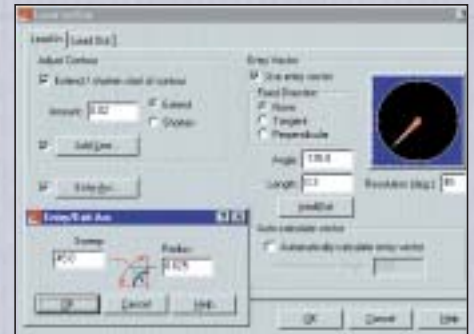
Multiple contouring for complete finishing



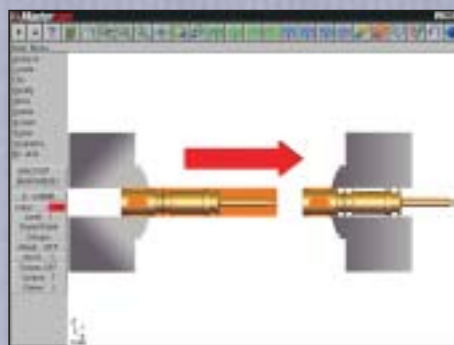
Fast thread machining



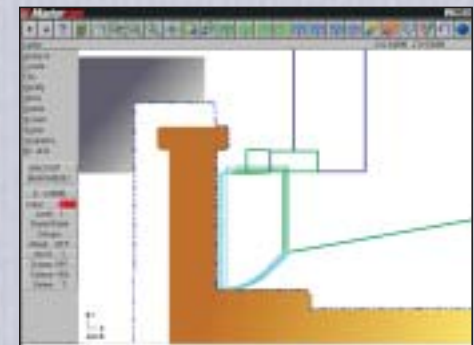
Stock recognition knows what is left after each operation



Full C-axis machining when combined with Mastercam Mill



Fast, easy stock transfer or part flip



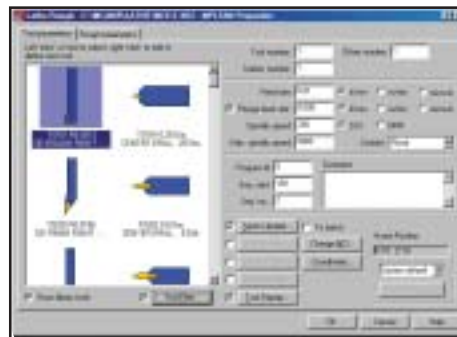
Clear, easy-to-follow prompts simplify actions such as lead-in / lead-out control

## Powerful Lathe Programming

- Click and drag a machining start point to anywhere on your model.
- Intelligent roughing of inside and outside diameters, including roughing to a boundary for castings.
- Optimized facing including rough and finish pass routines.
- 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 rate for desired finish quality.
- Groove at any angle 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.

## C-axis (Mill/Turn) Programming

- Full C-axis programming when combined with Mastercam Mill.
- Face and Cross contour.
- Face and Cross drill 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 drill.
- Machine C-axis contours with three-dimensional or flat geometry.



**Mastercam displays a tool list appropriate for your toolpath type. You can also select from the entire tool library or create your own tool.**

## 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.

## Dependable NC Verification

- Quickly check toolpaths as they are created with a cross-section of the part in progress.
- View or save a 3D model of your part at any machining stage.
- Step through the program with insert, holder and toolpath and dynamically view all the vital information at any point in the toolpath.
- Verify your part being cut with Mastercam's solid toolpath verification.

## Practical Tools for Full Control

- 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.
- Pick from a library of post processors or have one customized.

## Easy to Learn and Use

- Logical point-and-click interface lets you program the way you think.
- Fully customizable icons put frequently used functions at your fingertips.
- Context-sensitive, on-line help provides immediate access to Mastercam tips and instructions.
- Detailed tutorial.

## System Requirements

- IBM-compatible personal computer with a Pentium™ processor
- Microsoft®, Logitech® or compatible mouse
- Windows™ NT® 4.0 or higher, Windows 98, 2000, ME or XP.
- Minimum 32 MB RAM (64 is recommended for better performance); 50 MB free disk space in addition to Windows requirements

# Mastercam®

When Second Best Won't Cut It.

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