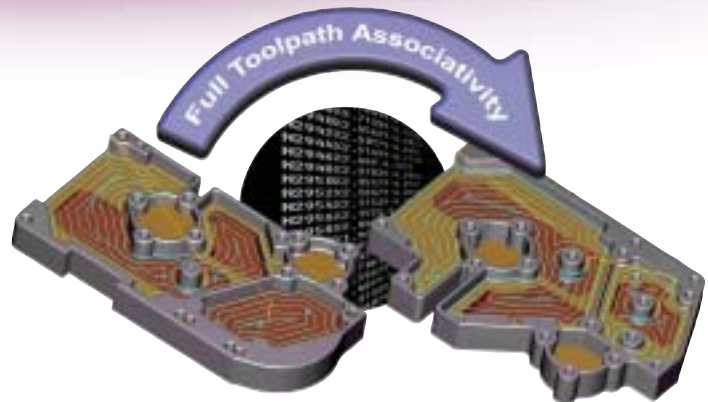


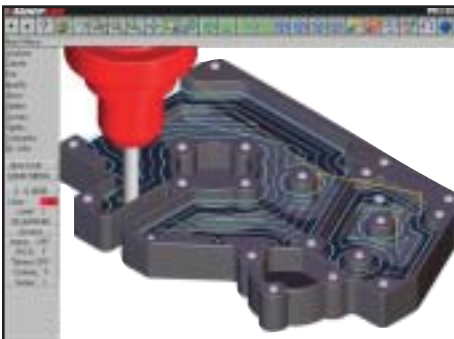
Mastercam

Mill Level 1



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 and surface modeling.
- Extensive editing and analysis tools ensure your model is accurate.
- Associative dimensions update as you change your model.
- Dynamic rotation, panning, and zooming in multiple views.
- 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.

Associative Toolpaths

- Fully associative geometry and toolpaths let you modify the geometry or machining parameters and immediately get an accurate, updated toolpath.
- Store a library of commonly used operations to automate machining. For example, you can spot drill, peck drill, and tap a new series of holes simply by importing one stored operation.

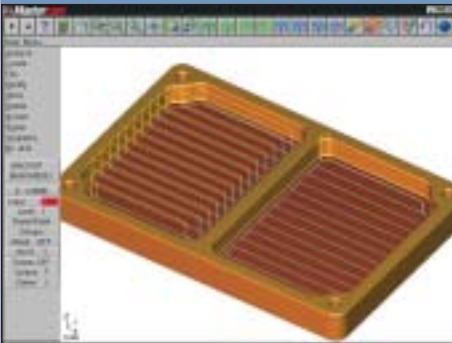
Efficient Toolpath Management

- Mastercam's Operations Manager stores all your job's operations in one place. Quickly create, edit and verify your toolpaths, or copy and paste parameters, toolpaths and tool definitions from one operation to another.
- Merge, translate, reverse, rotate, mirror, copy, cut and paste segments of your toolpath.
- User-customizable tool and material libraries automatically calculate feeds and speeds.
- Graphical toolpath editor gives you point-and-click on-screen editing tools.

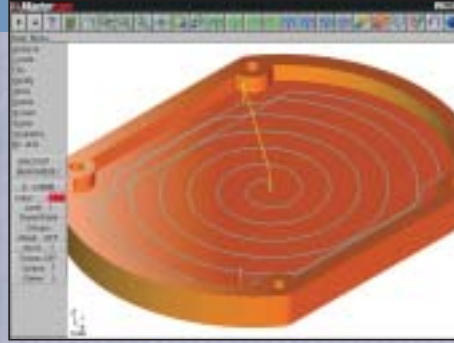
From simple to complex the toolpaths you need

Here are just some of the toolpa

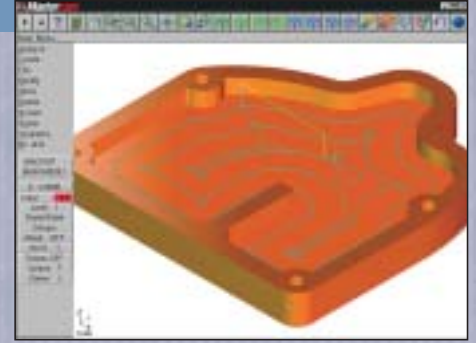
Pocketing



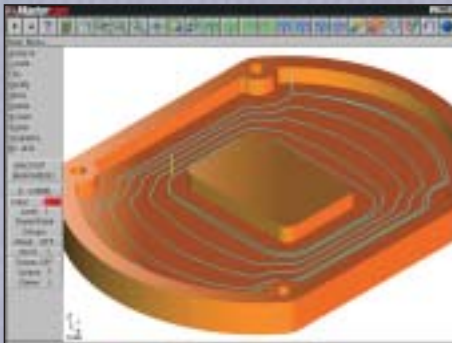
One-way and zigzag pocketing



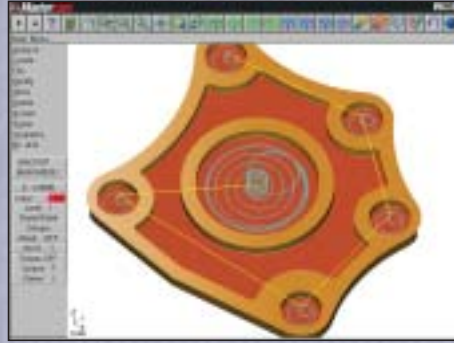
Spiral pocketing



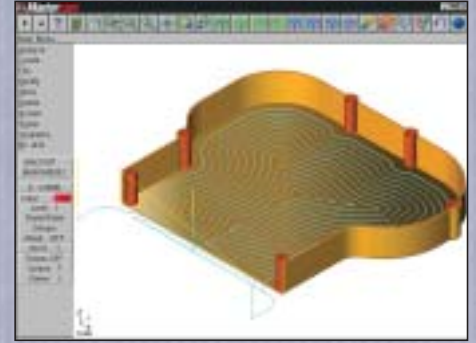
"Clean Corner" pocketing



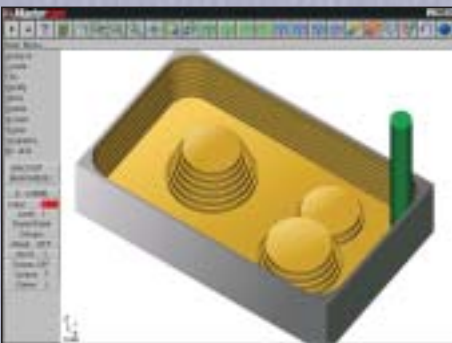
"Morph" pocketing



Automated circle milling



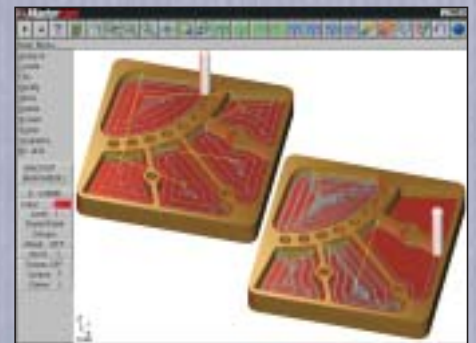
Open pocket machining



Tapered wall pocketing and islands of different heights.



Automated pocket remachining with a smaller tool

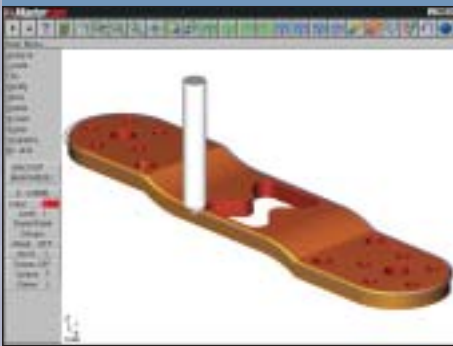


Machine pocket by pocket or complete Z-level cuts

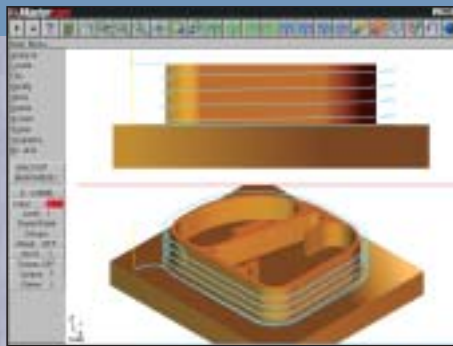
Mastercam has all it takes to get the job done.

Features that Mastercam offers...

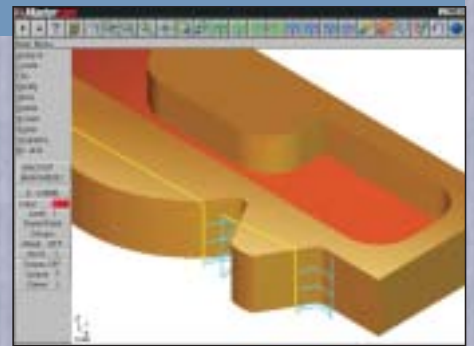
Contouring



3D contouring

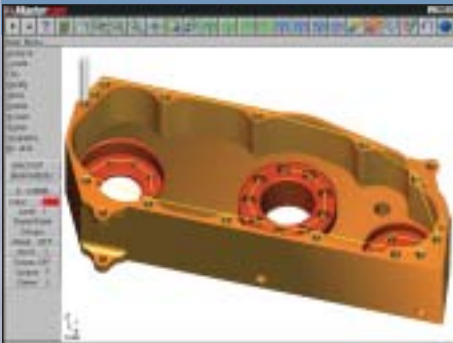


Smooth ramp contouring



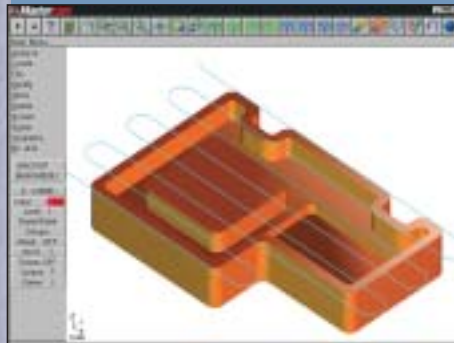
Contour remachining

Drilling

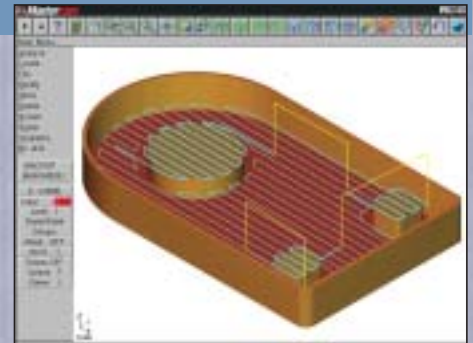


Fully optimized drilling

Facing

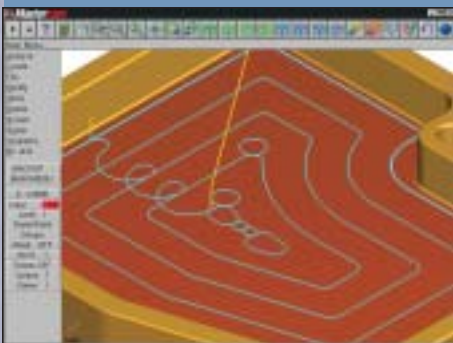


Automated part facing



Island facing

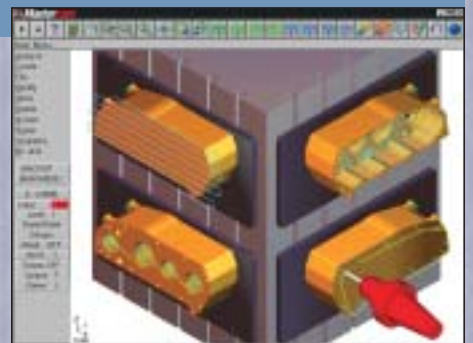
And More . . .



Specialized high speed pocketing



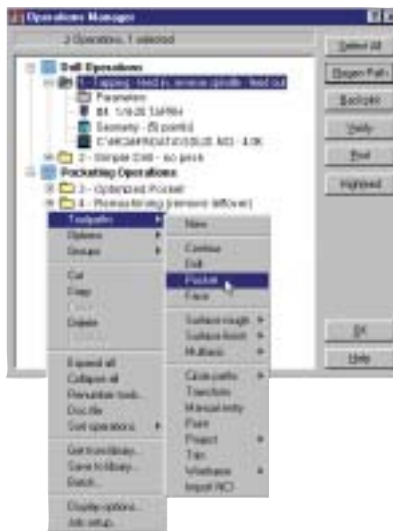
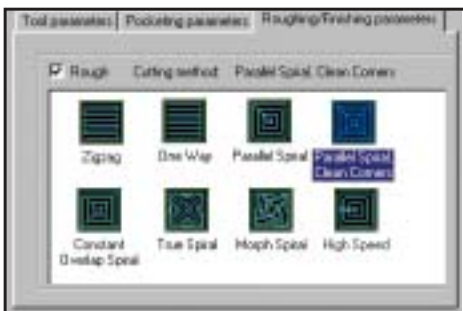
High-speed 'trochoidal' machining prevents the tool from being fully buried



Easy tombstone programming

Pocketing and Contouring

- Pocketing styles include high speed, zigzag, one way, true spiral, constant overlap spiral and "morph" pocketing, each with optional finish passes.
- Machine open pockets without creating additional geometry.
- Entry strategies include plunge, helical or ramp entry. Output helix moves as arcs or lines.
- Clean-corner and constant overlap spiral pocketing eliminate material when the tool turns a sharp corner.
- Contour and pocket remachining use a smaller tool to automatically clean out material left from previous operations.
- Wear compensation for contour and pocket finish passes lets you adjust for tool wear at the machine tool without creating a new toolpath.
- Click and drag a machining start point to anywhere on your model.
- Automated slot milling.
- Choose separate taper angles for contours, pocket walls, and pocket islands, including islands of different heights.
- Smart pocket depth control for thin-walled pockets lets you machine depths without retracting or machine all cuts in a single area before moving to the next area.
- Facing cleans stock from the tops of islands or the entire part.
- Program separate lead-in and lead-out for contours and pocket finish passes.
- Choose multiple roughing and finishing passes and multiple depth cuts for any contour.
- Easily machine 2D and 3D contours including parametric and NURBS splines.



High-Speed Machining

- New High Speed pocketing options including full "trochoidal" machining which "nibbles" out the entire pocket, ensuring the tool is never buried. You can also set the maximum percentage of tool contact on any single cut.
- Pocketing loops eliminate sharp motion.

Flexible Drilling

- Automatically identify and pre-drill multiple operations at their plunge points.
- Automatic drilling countersink depth calculation.
- Optimize drill routines to minimize tool travel.
- With the addition of the Solids option, Mastercam automatically recognizes and programs drill holes on solid models, complete with pre-drill operations.

Dependable NC Verification

- Watch your part being cut from stock with Mastercam's solid toolpath verification. The tool and holder are checked and displayed during simulation.
- Step through the program with toolpath backplotting and get an estimate of machining time. Dynamically view vital tool and operation details at any point.
- Quickly verify 2D toolpaths with a pixel paint of the full tool diameter.

Smart Feed Rate Optimization

- Automatically vary feed rates based on volume. More material and the cutter moves slower; less material, the cutter moves faster.
- Automatically ease the tool into and out of corners.
- Create optimization libraries for all your machines.

Practical NC Tools

- Choose multiple machining areas with a single selection.
- Full approach and retract control.
- Easy "tombstone" programming.
- Wrap a toolpath around a diameter with rotary axis substitution.
- "Safe zones" help ensure safe tool retract in tombstone and rotary work.
- Significantly reduce the size of a program with toolpath filtering.
- Custom tool geometry can be stored with the part file for easy retrieval.
- Fully customizable setup sheets.
- Pick from a library of dozens 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 puts frequently used functions at your fingertips.
- Context-sensitive on-line help provides instant access to tips and instructions.
- Detailed tutorial.

System Requirements

- Pentium-based PC with CD-ROM drive.
- Graphics capability of 800 x 600 resolution and 256 colors.
- Windows compatible mouse
- Windows NT 4.0 or higher, Windows 98, 2000, ME or XP.
- Minimum 64 Mb RAM; Minimum 350 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