





Cutting Anything. Everywhere.



Powerful Part Modeling

Mastercam's streamlined CAD makes design work easier than ever. Each piece of geometry 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.
- Fitting tool to help place parts between clamps and fixtures.
- Associative dimensions update as you change your model.
- Advanced analysis tools to help modeling and programming.
- Solid modeling is available as an optional Add-On.
- 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 Siemens NX, CATIA®, Pro/E, and more are also available.
- Special free Mastercam Direct Add-Ons put a Mastercam launch button in your SolidWorks, Solid Edge, or AutoCAD Inventor toolbars.

CAD model



Intelligent Machining

Precision Milling

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

Mastercam's File Tracking and Change Recognition offers an easy way to identify CAD edits and revise your toolpaths. Create a checklist of CAD files and Mastercam alerts you when a new version is available while identifying the changed areas for easy toolpath updating.

Machining Stock Model

A precise machining stock model delivers a variety of benefits—viewing and verifying work as it progresses, performing stock model comparisons, and easily choosing existing stock for rest machining. Mastercam gives you the power to create accurate, fully-associative stock models at any point in your machining process. You can reference the model within that project or save it as geometry for other uses. You can even quickly bring in outside CAD models to use as starting stock models for added flexibility.

Confidence at the Machine

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 want it. The Mastercam Simulator offers a single streamlined interface for solid-model verification and toolpath backplotting with a powerful set of analysis tools and information. Mastercam Simulator opens in its own window letting you work and adjust your project as the tool motion display continues.



The Mastercam Simulator delivers a variety of toolpath verification options and information.

Machine Simulation

Mastercam's machine simulation shows the entire machine tool and workpiece in action. Easily check and verify the cutter path, axis motion, retracts, table moves, and any other elements that impact how the part will interact with the machine. This vital information gives you the confidence that what you see is what you get.

Stock model

OCK MODE

COMPARE

* Source: CIMdata, Inc.





"The accuracy of our medical parts is" crucial. Patient care depends on it. And we depend on Mastercam."



Wayne Gruver Lead Toolmake Imaae Moldina Denver, CO



Full Machine Simulation delivers a practical view of how your toolpaths interact with

2D Toolpaths

Contouring, Drilling & 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) automatically programs a solid model's pockets, contours, and drilling routines, including new slug cutting and hole mapping.
- Standard pocketing styles include zigzag, one way, true spiral, constant overlap spiral, "morph" pocketing, and open pocketing.
- Suite of entry methods including plunge, helical, ramp, profile, medial, or custom including trochoidal entries.
- Contour and pocket remachining use smaller tools to automatically clean out material left from previous operations.
- Specialized support for ISCAR® High Efficiency Machining (HEM) tool set.
- **Region Chaining** delivers a fast and easy way to adjust your 2D high speed machining areas.
- 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.





Feature Based Machinina makes it easier to machine prismatic solids by automating the programming process.



The new Tool Manager delivers a fast, efficient way to organize and create your tooling and tooling assemblies.

Toolpath Spotlight: Dynamic Milling

Cycle time and tool wear are shops' constant concerns. Mastercam's dynamic milling is designed to improve both of these with a single powerful technique.

Dynamic milling creates a constantly adapting toolpath with consistent cutting conditions providing smoother, safer motion. This motion is easier on your machine and can effectively use the full flute length, greatly extending cutter life and often eliminating the need for multiple depth cuts. Optimized cut ordering, specialized motion to keep the tool down, and other elements combine to deliver parts faster than ever before.



Creates an active toolpath, delivering more consistent cutting conditions and using the entire tool flute length.



Roughing, Finishing & 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:

- Cut multiple surfaces, solid models, and mesh entities (STL data).
- 3D Toolpath Refinement provides unsurpassed control on surface cuts, delivering superior finishes and optimized cycle times.
- High Speed OptiRest uses Mastercam's new stock model to identify and efficiently machine areas that need to be roughed with a smaller tool.
- 3D "projected" machining creates a consistent, smooth finish that follows the natural curves of the geometry. In addition 3D HST offers toolpath projection based on curves, points, or NCI data.
- Constant scallop machining maintains a consistent finish on sloped and flat surfaces alike by using a consistent 3D stepover.
- Smart hybrid finishing creates a single toolpath that changes cut methods as the slope of the model changes.
- Constant-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.
- Automatic roughing of critical depths.
- Full check surface support.

Go to www.MastercamMill.com for more.



Hybrid finishing intelligently blends two efficient cutting techniques in one toolpath.





Conventional Finishing

Refined Finishing

3D Toolpath Refinement delivers a dramatically superior finish.

Toolpath Spotlight: **Optimized Roughing**

Mastercam's suite of innovative, fast roughing toolpaths—OptiCore, OptiArea, and OptiRest—are all designed to remove large amounts of material quickly using concepts from our popular dynamic milling motion.

Large, aggressive cuts are followed by fast, smaller up-cuts, safely delivering a fully roughed part in dramatically less time—as much as 60% faster or more. Like our dynamic milling toolpaths, these roughing techniques promote longer tool life.









OptiRough removes bulk material faster and with more consistent tool wear.

e • Aerospace • Automotive • Medical • Energy • Consumer •



"Mastercam's dynamic toolpaths have allowed us to push our machine tools and reduce cycle times as much as 50%."







Streamlined multiaxis programming tools make projects easier than ever before.

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: cut pattern, tool axis control, and collision avoidance.

Some highlights of Mastercam's multiaxis machining:



Specialized toolpaths support fast, efficient 5-axis drilling.



Mastercam's oscillating 5-axis curve and swarf motion promotes even wear, greatly extending tool life.



with or without supporting surfaces or



splitters, leading and trailing edges, hubs, and more.





with smooth motion.

removes material left from a previous rough pass.

Faster Turnaround & Superior Finish

High Speed Machining

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 HSM toolpaths designed to help you make the most of this strategy. Best of all, you don't need a dedicated high speed machine to use HSM—any machine can benefit from this powerful technique.

Feed Rate Optimization

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 slower, 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.



and drilling.

- "Rail" swarf cutting delivers fine cut control

Specialized motion ensures clean cuts on



- 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 for applications such as trimming vacuum-formed parts.
- Easy 4-axis rotary, rolldie, and 5-axis drill programming.
- Minimum tilt control helps prevent tool motion that would cause tool holder collisions.
- Create full 5-axis motion from a 3-axis toolpath.
- 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.
- Mastercam Blade Expert delivers specialized tools for efficiently cutting multi-bladed parts and hubs.
- Mastercam Port Expert offers dramatically faster head port programming and smoother toolpath motion.



High speed pencil tracing leaves a precise finish while being easy on your machine and tooling.



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

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





Specialized Options

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

- Focused 5-axis programming tools tailored to cutting multi-bladed parts and cylinder head ports.
- Complete programming for complex machining robots.
- In-process probing and inspection for easy set-up/validation of parts and cutting tools as well as in-process adjustments.
- 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 EDM electrode creation, including a library of definable stock sizes and materials.
- See a full list at <u>www.Mastercam.com/Products/AddOns</u>.





671 Old Post Road Tolland, CT 06084 USA

(800) 228-2877 • (860) 875-5006 • Fax (860) 872-1565 www.mastercam.com • mcinfo@mastercam.com



N	Aastercam Mill	Level 1	Level 2	Level 3
CAD				
Create and dimension live wireframe geometry		Х	Х	Х
Read/write IGES, DXF, SAT, Parasolid, EPS		Х	Х	Х
Read native AutoCAD, SolidWorks, Solid E	dge	Х	Х	Х
Read native CATIA, Pro/E, Unigraphics (UC	5 NX)	Optional	Optional	Optional
Live surface modeling		Х	Х	Х
CAD File Change Recognition		х	Х	Х
Solid modeling		Optional	Optional	Optional
САМ				
In-process stock model		Х	Х	Х
Fully associative toolpaths		Х	Х	Х
Dynamic Milling		х	Х	Х
Feature Based Machining		Х	Х	Х
Automated Feed Rate Optimization		Х	Х	Х
Contouring, pocketing, and drilling		х	Х	Х
2D High Speed Machining		Х	Х	Х
3D contour cutting, trimming, and remac	hining	х	Х	Х
On-screen toolpath verification		Х	Х	Х
Fast, easy tool management		Х	Х	Х
Machine and control definition		Х	Х	Х
Engraving		Х	Х	Х
Raster to Vector image conversion		Х	Х	Х
Single and limited multisurface roughing			Х	Х
Single and limited multisurface finishing			Х	Х
Full multisurface and solid roughing				Х
Full multisurface and solid finishing				Х
Full multisurface and solid "cleanup" mach	hining			Х
Full multisurface and solid High Speed Ma	achining			Х
Full multisurface and solid optimized roug	hing			Х
Full multisurface and hybrid finishing				Х
5-axis drilling and curve machining		Optional	Optional	Optional
Simultaneous 4- and 5-axis machining				Optional

System Requirements

- Processor: 32-bit or 64-bit Intel or AMD w/SSE2 support.
- Operating System: 32-bit or 64-bit Windows 7 or Windows 8.
- System Memory: 2GB (32-bit OS) or 4GB (64-bit OS).
- Hard Disk Space: 40GB, 1GB free.
- **Graphics:** Minimum 1280 x 1024 resolution monitor, 128 MB graphics memory, OpenGL driver support.
- Microsoft Products: Microsoft IE v6.0 or higher, Excel and Word 2007 or higher.