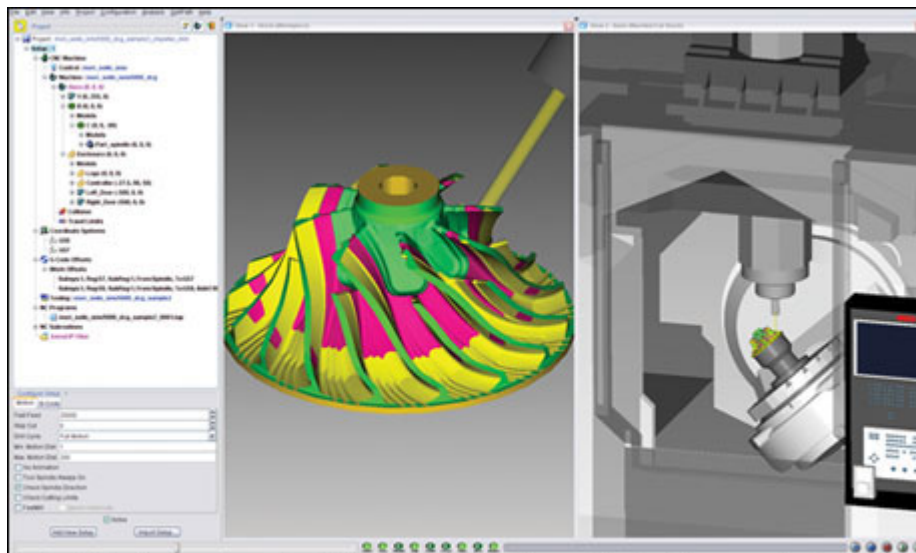

Manufacturing ENGINEERING

February 2010 Vol. 144 No. 2

CAD/CAM Software Drives Innovation

Latest software advances include new parallel-processing techniques, improved simulation tools, and enhanced support for complex multitasking machine tools

Patrick Waurzyniak, Senior Editor



Vericut 7.0 simulation software from CGTech offers new motion emulation logic that more closely mimics real motion on CNC machines.

As manufacturing emerges from the economic maelstrom, factory-floor managers can look toward the latest CAD/CAM and simulation software systems for cutting-edge tools to help improve their manufacturing productivity. With the latest CAD/CAM packages, manufacturing operations can add a host of new features including parallel processing techniques, better simulation capabilities, and support for complex machining on newer

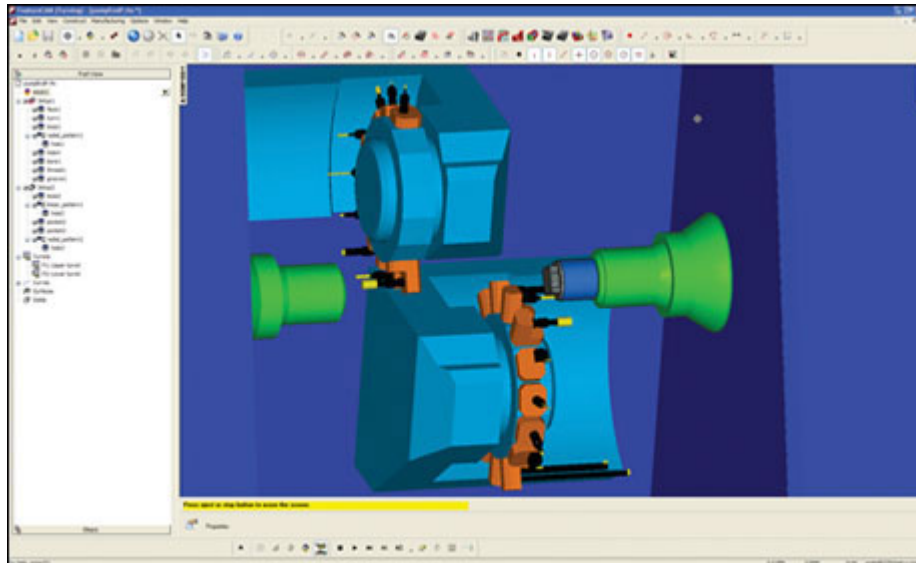
multitasking machine tools.

Parallel processing holds the promise of measurably speeding up programming tasks for NC programmers. In the past year, several CAD/CAM software developers have added parallel and background processing with support for multicore processors, allowing programmers to significantly speed up CAM programming tasks with parallel and multithreading capabilities (see the article "[Programming in Parallel](#)" in the October 2009 issue of *Manufacturing Engineering*).

"With the prevalence of multicore computers, many CAD/CAM suppliers are now offering multithreading, executing different threads simultaneously, as a way to gain significant performance gains in machining and cycle times," notes Glenn McMinn, president, Delcam North America (Windsor, ON, Canada), a unit of Delcam plc (Birmingham, UK). "Because Delcam owns all its machining code, we are free to optimize any of our algorithms, and furthermore, we're able to include this new technique in a number of our software programs. In 2010, the releases of Power-MILL, FeatureCAM, and ArtCAM all provided significant performance improvements based on multithreading."

In Delcam's PowerMILL 10, the company has developed two techniques that take advantage of the latest in multicore processing, McMinn adds. "The first technique involves rewriting machining algorithms so they could be performed in parallel. The second technique involves calculating the toolpaths using a background process, called 'queued' [line up], so you can continue to use PowerMILL in the foreground while toolpaths are calculating in the background. Users can prepare data faster and eliminate any waiting. Similarly, the user can be setting the parameters needed to machine one area of a part, while toolpaths are being calculated in the background for machining a second area with a different strategy. Productivity is also increased because the computer can continue with a series of calculations during the user's breaks or even overnight.

"With multithreading, we can speed up the majority of toolpath calculations in raster machining; other strategies using this code include constant Z, 3-D offset, area clearance, interleaved constant Z, optimized constant Z, and boundary calculation," McMinn states. "In addition, if the user's computer is suitable, parallel processing happens automatically on some algorithms. The typical PowerMILL customer is concentrating on the manufacture of high-precision parts and some of the toolpaths can be compute-intensive. Overall, we feel that Delcam's interpretation of multithreading can benefit the user, since we concentrate on speeding up individual toolpaths, not just dedicating a separate process to each toolpath calculation."



Migrating its parallel processing technology, Delcam's FeatureCAM 2010 also includes multithreading capability derived from the company's PowerMILL system.

Initial testing on PowerMILL 10 with quad-core machines show a savings of up to 60% in toolpath calculation times, McMinn notes. Delcam has added the programming techniques to FeatureCAM 2010 with support for multithreading when generating 3-D toolpaths, and a number of new options are now available for turned features and for multiple-turret machines.

In addition, the company's PartMaker Version 2010 has added the capability to perform five-axis simultaneous milling on multiaxis lathes, based on proven technology used in PowerMILL. "PartMaker Version 2010 is the most significant release made to date, and will provide every PartMaker user an immediate productivity improvement," asserts Hanan Fishman, president of Delcam's PartMaker division.

"CAD/CAM software has typically followed the machine tool technology, meaning CAD/CAM developers typically develop software tools to automate certain machine tool functionality," Fishman adds. "That trend will certainly continue, but to some degree, we are seeing a convergence in technology. Machine builders are making more complex machines, and they're finding markets for these machines because of available CAD/CAM technology that can drive them. Thus, you're finding new technologies where we you have not seen them before, with five-axis mills integrated into production multiaxis turning centers, and lathes integrated into sophisticated milling centers."

Advanced simulation tools allow CNC programmers to accurately simulate toolpaths prior to cutting metal. With the latest Vericut 7.0 NC simulation, verification, and optimization package, CGTech Corp. (Irvine, CA) has implemented new motion-emulation logic that more closely mimics real motion on the CNC machine, notes Bill Hasenjaeger, CGTech product manager.

"When simulating manufacturing processes with software, it's important to get as close

as possible to emulating the physical workshop's equipment," notes Hasenjaeger. "For CNC-controlled machinery, one obvious component is how well the simulation emulates the way the CNC control moves the machine. As CNC machine simulation technology has matured, the engineers who rely on it realize that simulating from something other than the actual CNC program is very nearly a waste of time. Maybe that's too harsh, but the ability to simulate from the exact 'Gcode' NC program file that will run on the machine is a necessary component of any trustworthy CNC simulation.

"Some of the more subtle and important features in CNC simulation software is to provide the ability to ensure shop standards are met," he adds. "Modern CNC simulation software must do much more than simply move a virtual machine—it must be able to check the NC program based on workshop standards, which means it must be easily configured to meet those standards. A simple example of meeting a shop standard is the software's ability to be configured to compare the tool offset number to the current tool number, to ensure the NC programmer activated the proper offset number, per workshop standards. While it is perfectly valid for the CNC control to use any offset number for any tool, many shops standardize on a tool-offset number scheme to ensure the operator sets up the machine the same way each time, thus reducing the chance for errors at the machine. If the NC program mistakenly activates offset 23 when the operator expects it to activate offset 32 [based on shop standards], serious career-threatening problems may ensue."

The most important task of CNC machine simulation software is to ensure the NC program produces the correct workpiece, Hasenjaeger points out. "This means that beyond CNC control emulation, machine motion kinematics, program syntax-checking, and other NC program error checks, the software must produce a measurable and accurate workpiece—and it must do so reliably under all circumstances. The trend continues to be to make the simulated workpiece as accurate a representation of the real physical workpiece as possible."

With Vericut 7.0, CGTech focused on customer requests for ease-of-use improvements. "We modified the user interface to create a more natural and obvious sequence to the most common user actions," says Hasenjaeger. "We implemented completely new motion-emulation logic that more closely mimics real motion on the CNC machine. As most computer users know, any computer representation of a physical thing is always an approximation. Where our perception of reality is of a smooth continuous world, the computer represents it with millions of bits of digital data. The goal in any computer representation is to make the digital steps small enough that we perceive them as smooth and continuous. But realistic representation comes at a price—speed and memory."

Increased five-axis machining and the use of accurate simulation can help make manufacturers more competitive, notes Vynce Paradise of Siemens PLM Software (Plano, TX), a unit of Siemens Industry Automation Division (Nuremberg, Germany). "We're seeing more emphasis on variable/five-axis machining with greater use of five-axis machining technology in areas previously reserved for three-axis or 3+2 axis

machining, like mold and die," Paradise notes. "There's also been a greater focus on accurate machine tool simulation, from the G-code rather than basic toolpath data, driven by the use of more complex, advanced machine tools."

In Siemens' latest NX CAM 7.0, the company has added its Synchronous Technology for fast part model preparation irrespective of the CAD source, Paradise notes. "This can save the NC programmer a lot of time in working with CAD models that need to be adjusted or edited to get them ready for NC programming. Models imported from other systems into NX, or created with a complex parametric structure, can be hard to edit. Programmers may even start from scratch and remodel. With NX Synchronous Technology model, editing is fast and easy regardless of source or model type or structure."

The updated NX CAM also features interactive parallel NC processing, using multiple processors where available, for faster programming on large jobs. This approach allows the programmer to carry on programming while the system computes toolpaths and automatically runs parallel processes using available computing resources, he adds. NX CAM also offers on-machine probing, plus a Generative Feature Based Automation function driven by model-based PMI (Product and Manufacturing Information, such as tolerances and surface finish). Another update is that NX CAM produces optimized output for Siemens Sinumerik controllers. "This ensures that the advanced functions of the Siemens controller can be utilized using postprocessors built using the NX Post Builder application. NX CAM has additional commands that allow the programmer to activate advanced Sinumerik functions, and the new postprocessors will automatically set additional output parameters to optimize performance at the machine tool. This leads to greater efficiency at the machine tool, and less time spent hand editing code to add special commands."

Flexible CAD/CAM software capable of handling virtually any model type is in high demand now from most shop-floor users, notes Mark Summers, president, CNC Software Inc. (Tolland, CT), developer of Mastercam software. "Models are coming to shops in so many different formats now," notes Summers. "Reliance on blueprints continues to dwindle. We are also seeing the need to use many nontraditional formats, such as scanned data from lasers and reverse-engineered data. Software must have the ability to not only use these model formats, but also manipulate the data so as to prepare these models for manufacturing. There is an increasing need for flexibility, automation, and superior finishes. Often, shops need these three components at the same time, while looking for faster ways to get parts off the machine, and make the most of the tooling and machines they already have."

Most shops want software used in conjunction with the right tool/machine combinations that can improve their surface finish beyond what was expected of them in the past. "Software packages need to take advantage of tool and machine technologies, and produce tool motion that is tailored to take advantage of these specific technologies," Summers states. "Another trend is 'high cube machining,' which stresses efficiency over speed. There is more material removal, but not by moving the cutter faster, which is the

traditional way of removing material as in high speed. The focus is material removal rate, not just moving the cutting tool faster."

In the latest Mastercam X4, CNC Software has added a host of new enhancements and features including Toolpath Refinement, which provides dramatically superior finishes on complex parts, notes Summers. "This new utility gives you greater control over the toolpath motion in order to achieve a superior finish. We've also added Dynamic Mill, which gives you the ability to utilize the entire flute length of the cutting tool while maintaining more consistent cutting conditions. This saves cutting time while reducing wear on your tooling and machines. Remember 'high cube, not necessarily high speed.'

"Systems with multiple processors/cores are becoming common, and the trend toward using additional processors and cores will continue. Multithreading gives you the power to segment complex tasks in a multicore computer, delivering faster processing. Another benefit is that the user can continue working in Mastercam as the multithreading manager is processing."

More CAM multicore support is available in the latest Esprit 2010 CAD/CAM package from DP Technology Corp. (Camarillo, CA), notes Chuck Mathews, DP Technology vice president. "The trends in the CAM business are most commonly driven by the latest CAD data formats, new machine tools, controls, and cutting tools, as well as the latest operating systems and computer hardware," Mathews states. "All of these require CAM vendors to continually update their software in order to keep pace. Direct requests from customers for more specific productivity-enhancing functionality for part programming, such as simulation and post processing, also influence CAM vendors.

"At the present time, the computer technology trends that stand out are the changes brought on by expanding use of multicore computers and the Windows 7 operating system," he adds. "CAD-to-CAM data exchange is evolving to include more design, features, and tolerances, expanding its role beyond the geometry. Manufacturing technical trends are primarily driven by increasing demands for more sophisticated postprocessing, and machine simulation to keep pace with the increasing complexity and number of mill-turn, Swiss-turn, and five-axis machines installed around the world."

Esprit 2010 has been optimized to fully embrace the capabilities of the latest multicore computers, an especially valued time-saving characteristic for three-axis and fiveaxis programming, adds Mathews. The updated Esprit includes the new multithreaded stock automation engine, which can benefit users by starting the part-program simulation at any time and at any point in the program. "This new feature eliminates the need to simulate all operations from the beginning or to save the results of previous simulations," Mathews notes. "Now, Esprit customers can start a simulation with the stock that results from the completion of all previous operations without first waiting for the simulation of the previous operation. The use of multithreading technology—with which stock calculations are done in the background—won't slow the programmer

down."

High-efficiency material removal technology is a trend in the industry, notes Bill Gibbs, president, Gibbs and Associates (Moorpark, CA), which recently signed an agreement to use the VoluMill toolpath optimization engine from Celeritive Technologies (Cave Creek, AZ) with its GibbsCAM CAD/CAM system. "Most improvements in productivity relate to more efficient toolpaths for cutting parts faster," Gibbs notes. "Cutting parts faster remains a significant cost factor in a job shop. Multitasking machines [MTM] continue to do well. Our MTM software and post-processor sales have not diminished during the recession."

With the latest GibbsCAM, users have many new enhancements to help improve productivity. The package features Windows 7 compatibility and a new Windows 7 part file preview function; improvements in part roughing, and Advanced 3D toolpath; a new plunge rough function optimized for insert drills; and VoluMill Solids integrated in GibbsCAM, Gibbs notes. The system also has improvements in five-axis milling and new tool types, impeller strategies, adaptive cuts to morph, and a new spiral cutting method, as well as a new MTM synch interface.

An integrated version of CAMWorks 2010 from Geometric Technologies Inc. (Scottsdale, AZ), a unit of Geometric Ltd. (Mumbai), will also soon be available for running directly under SolidWorks. The updated CAMWorks 2010 adds more than over 50 customer-driven features and enhancements for machining faster and more accurately, including advances to feature recognition, improved multiaxis machining, and new cutting strategies for two to five-axis milling, turning, and wire EDM.

WANT MORE INFORMATION?

[CGTech Corp.](#), P: 949.753.1050

[CNC Software Inc.](#), P: 860.875.5006

[Delcam plc](#), P: 877.335.2261

[DP Technology Corp.](#), P: 805.388.6000

[Geometric Technologies Inc.](#), P: 480.222.2255

[Gibbs and Associates](#), P: 805.523.0004

[PartMaker Inc., a division of Delcam plc](#), P: 877.335.2261

[Siemens PLM Software](#), P: 800.498.5351