The cost benefits of deploying multitask machine tools are undeniable, with multitask machines offering the ability to perform many machining operations on parts with just a single setup. By employing multitask machining (MTM), manufacturers can quickly produce high-quality, highly complex parts while eliminating most of the labor involved with machine setups, thereby making North American manufacturers much more competitive with offshore manufacturing operations. As MTM equipment prices have fallen and the machines become more popular, many CAD/CAM software developers have released new or updated software packages with enhanced MTM support.

BETTING BIG ON MULTITASKING

Newest CAM releases offer better multitasking, multiaxis support, and usability enhancements

Patrick Waurzyniak
Senior Editor
“There are some very, very important trends that have been going on—and they’re all being driven by economics,” said Bill Gibbs, president of Gibbs and Associates (Moorpark, CA), developer of GibbsCAM software. “We’re very happy with the reshoring of work coming back to the US. By far and away the biggest factor in all of this is increasing levels of automation and high-productivity CNC equipment, particularly with multitask machines, which continue to be the fastest-growing segment of CNC manufacturing. This trend continues to be the primary way to remove the cost of labor out of a manufactured part, and once we reduce the labor, then there’s no advantage to having it manufactured in Asia.”

New MTM Programming

An early advocate of multitask machining, Gibbs and Associates introduced its first MTM programming solution a decade ago. At IMTS 2012, the company demonstrated its next-generation GibbsCAM MTM programming for complex MTM machines including five-axis rotary head and Swiss-style machines. GibbsCAM’s easy-to-learn user interface walks users through complex part programming, including multiple spindles and turrets cutting parts simultaneously, and the program allows users to see accurate simulations of parts being machined, with full visualization of any errors like tool interference and gouges before parts get to the shop floor.

“Of all the bets I made with my business, this is the one I got right 10 years ago when I said the most important thing for us was to get ready for the advent of the multitask machine,” Gibbs said. “What this offers the customer is the ability to put material in one end and get a complete part out the other end. Each of those setups requires human labor, and that also means there’s lot of opportunity for human error and cataclysmic problems to happen.”

Today’s multitasking machine tools are extremely complicated, requiring CAM software capable of programming and accurately simulating simultaneous operation of multiple spindles and turrets. “CNC machines are getting much more advanced and software has been lagging behind,” noted Vivek Govekar, CAMWorks R&D Head, Geometric Technologies (Scottsdale, AZ, and Mumbai). “What we are seeing is that the machines have developed much faster than the software. There are not many software packages that can handle four turrets at a time, and very few can do two turrets.”

With its CAMWorks 2013, also previewed at IMTS 2012, Geometric Technologies introduced a system that includes the new Synchronous Machining Module (SMM) that greatly simplifies programming of complex mill-turn machines, Govekar noted. CAMWorks 2013 also provides what the company calls the first cost-effective true G-code machine simulator, giving users the ability to create “first time right” programs for mill-turn and other complex multiaxis machines.

“More programmers want to try the program on virtual machine simulation
before putting it on the machine,” Govekar added. “With its multifunction capabilities, CAMWorks can program combined mill-turn machining, along with the ability to synchronize two turrets to machine concurrently, and CAMWorks offers a wide array of APIs (application programming interface) to automate the entire programming process.” CAM users also want to automate the programming and machining to have quicker turnaround time and less dependence on people, he added. “It’s not about automating a part of the programming, like automatic feature recognition; they want the entire flow to be automated.”

“Multitasking makes machining on today’s high-powered turning centers and full-functioning machining centers simpler and easier than ever before,” said Gary Hargreaves, CNC Software director of business development and international

“Multitask machines continue to be the fastest-growing segment of CNC manufacturing.”

The virtualization and emulation of highly complex machines is always a moving target, as machine tool technology is often ahead of CAD/CAM software developers, according to NC Kishore, senior manager, Industrial Equipment, Dassault Systèmes’ Delmia brand (Auburn Hills, MI, and Velizy-Villacoublay, France). “It’s always been that way, especially with the rapid pace of new developments in high-performance, multifunctional mill-turns and other complex machines,” Kishore said. “The new trend is toward having an automated machining cell, with a combination of a robot and a machine, in one integrated system where robots hold workpieces, do load and unload, and a whole lot of other automation.”

In its upcoming Mastercam X7 release, CNC Software Inc. (Tolland, CT) plans to add multitasking capabilities to the CAM system with the industry’s largest installed base in terms of seats.
“Machines that eliminate multiple setups, manual part handling, redundant fixturing, and reduce idle time boost production and maximize profitability.

“We expect to see an increase in multifunction machines and horizontal milling machines,” Hargreaves said. “The prices are more affordable to shops now and the technology is allowing shops to expand their businesses to become more profitable. Mastercam’s Multitasking product will be something users can take advantage of to reduce idle time, boost production, and maximize profitability. MT streamlines the programming process with intelligent job setups that are keyed to the exact machine in the shop.”

Finding the Right CAM System

With technology advancing at an exponential rate, manufacturers need to make sure they’re using the right CAM software. “More than ever, it is technological advancement that keeps manufacturers ahead of the competition and investing in more advanced and complex machinery. However, with complex machinery comes inherent programming challenges and, in many cases, these machines are being programmed with ineffective CAM software,” said Dave Bartholomew, product manager, DP Technology Corp. (Camarillo, CA). “Companies and users are realizing that software changes are needed and want a CAM solution that not only supports the latest generation of machine tools, but one that is a full-spectrum CAM system capable of programming all of their CNC machines.”
With the high expense of running machine tools, all CAM users are looking for ways to optimize machining time through increasing toolpath and cutting efficiency, Bartholomew said. With the new Esprit CAM software, DP Technology has added the new ProfitMilling strategy, a high-speed material-removal cycle that allows taking significantly deeper, faster and more efficient cuts while reducing wear on the tool and machine. ProfitMilling is patent-pending and is designed for two, three, four and five-axis roughing cycles. The system bases its toolpath calculations—and roughing strategies—on the current stock state, which eliminates inefficient toolpath movement and air cutting, he added.

“Another productivity-boosting area is machine-tool simulation, which entails more than just catching programming errors before they go out to the machine,” Bartholomew said. “It’s a means of performing realistic simulation of an exact replica of the machine tool, with the inclusion of actual machine-tool kinematics in the calculation of toolpaths. This new technology will overcome many limitations innate to CAM systems, and gives the CAM user more accurate information to use, which increases performance.”

In addition to the new ProfitMilling strategy, among the many upgrades featured in ESPRIT 2013 are new options for cutting grooves faster and more efficiently, built-in collision avoidance in multi-axis milling, and a completely new, editable and customizable engraving machining cycle.

“Each of those setups requires human labor, and that also means there’s lot of opportunity for human error.”
Some key trends affecting CAD/CAM are coming from developments in related technology, such as cutting tools that can operate at higher speeds in harder materials and increasingly complex machinery configurations, especially in mill-turn equipment, according to Glenn McMinn, president, Delcam North America (Salt Lake City, UT). “These developments offer potential benefits in increased productivity and shorter delivery times. However, these benefits can only be achieved with CAM software that can take full advantage of the new technology.

“Most CAM systems now support recent computer hardware developments such as 64-bit operation, multi-threading and background processing, and most customers have upgraded their hardware to take advantage of these developments,” McMinn said. While the current focus is on exploiting developments in machine tools and cutting tools, another opportunity to boost productivity comes from the closer integration of machining and inspection. “This technology can vary from simple routines to assist automated machining, to probing cycles to check the position and orientation of the part before machining starts, or sophisticated adaptive machining options for efficient finish-machining of near-net shape parts from casting or forging.”

Among Delcam’s latest enhancements to its five-axis PowerMILL software is the lead and lean angles can now be defined relative to a contact normal, McMinn noted.
“In addition, better control is now possible for the tool-axis definition when using the To or From a Point, Line or Curve strategies. These improvements will make it easier for users to control the contact angle between the cutter and the part, and to set the optimum cutting conditions for efficient machining and quality of surface finish.”

“In many cases, these machines are being programmed with ineffective CAM software.”

New options have also been added to give smoother five-axis machining, McMinn said. “A new look-ahead capability will remove any sudden tool-axis changes during automatic collision avoidance and instead ensure that a smooth transition takes place,” he added. In addition, FeatureCAM now offers the ability to smooth the azimuth and elevation axes independently and help remove excessive five-axis movement. These options stabilize the rotary axis movement by maintaining a fixed tool axis as far as possible, he said, thus replacing constant five-axis movement with a series of 3+2-axis segments having full five-axis transitions between them.

Improving CAM Automation, Usability

Users of CAD/CAM software are increasingly using advanced modeling intelligence to make key tasks easier and faster, noted Vynce Paradise, director, product marketing, Siemens PLM Software (Plano, TX), developer of NX CAM software. “For example, editing 3D models in preparation for NC programming is a key task and can take a lot of time even with standard 3D modeling capabilities. The latest modeling technol-
CAM users need ease of use and smart applications that can reduce user inputs, he added. “For example, many traditional systems have very basic methods to define machining areas and volumes on prismatic parts. The parts look simple but the programming using typical methods is tedious. Often users have to make many selections to pick and define boundaries or to create planes in 3D space that are to contain the areas to be machined,” Paradise said. Users expect this to be more automated, to require less picks and for the result to be associative to the part and to the machining sequence. “In NX we have invested in smart programming for prismatic parts that uses a lot more 3D model technology to automatically define the volumes to be machined with far fewer user selections.”

The latest NX CAM 8.5 features volume-based milling, which is standard in all of NX CAM’s base milling packages. It allows the user to rapidly and easily select and define volumes to be machined, and it understands and references the model of the uncut (or stock) material between each operation, Paradise noted. The system shows the programmer a preview of the intended cut pattern before he decides to accept and compute the toolpath, he said, and customers report up to 10 times faster programming on prismatic parts.

“It’s programming the machine in the 3D life-like environment exactly as it behaves on the shop floor.”

In the latest release of Delmia’s V6R2013X portfolio, Delmia has added a suite of new enhancements to its all-in-one programming and simulation tools for machining, which includes Extended Milling Machining, Milling Machining, NC Machine Builder, NC Machine Simulation, Prismatic Machining, and Turning Machining. The update includes expansion of its capabilities of emulating controllers such as the Siemens 840D and other leading brands, noted Delmia’s Kishore. The
portfolio for NC programmers comes with an integrated data management and collaboration capability that adds value to programmers by saving time and cost.

“With our latest releases, it’s programming the machine in the 3D life-like environment exactly as it behaves on the shop floor,” Kishore said. “To complement programming the machine for those complex parts, we have automated capabilities, like the process templates and catalogs that could be reused to save programming time. There is now support for dual-core machines, enabling multiprocessing where the processing time is significantly improved.

“When our latest releases, it’s programming the machine in the 3D life-like environment exactly as it behaves on the shop floor,” Kishore said. “To complement programming the machine for those complex parts, we have automated capabilities, like the process templates and catalogs that could be reused to save programming time. There is now support for dual-core machines, enabling multiprocessing where the processing time is significantly improved.

There are also some usability improvements, where we’ve eliminated a number of clicks and improved automation so the programmer can do his job with the least amount of clicks,” Kishore added. “This again is an ongoing effort, in terms of improving the usability all the time, based on the customer feedback, and we’ve come a long way integrating these products to deliver the best value-added experience in machine programming.”

NC Simulations Crucial

With today’s machine tools, obtaining accurate NC simulations is critical to success, and the sharing of key machine details between suppliers has sometimes been a barrier, notes Bill Hasenjaeger, product marketing manager, CGTech (Irvine, CA), developer of the Vericut software package for NC simulation, optimization and verification. “Accurate NC program simulation, especially in the context of machine simulation, requires accurate representative 3D geometry models of the hardware involved in the machining process,” Hasenjaeger said. “In the past some of these
suppliers have been reluctant to share detailed 3D geometry models of their products, fearful that they may be giving competitive suppliers access to trade secrets.”

That reluctance to share data has been slowly changing in recent years, especially with a greater emphasis on the importance of NC program simulation in the CNC machining processes, he added. “In fact, some manufacturing organizations require suppliers to provide 3D models of CNC machines, cutting tools, and fixtures at the time of purchase. Sometimes models are only supplied directly to the end-user, under a licensing agreement, but more and more 3D models are simply being provided on suppliers’ websites. At CGTech we’re also seeing machine tool builders becoming more likely to share models directly with us, for use on specific end-user customer projects.”

Today’s simulation users typically want software that’s faster and easier to use, Hasenjaeger said, but also they’re looking for the best ways to improve the design of their machining processes. “End users want an easy way to create and share descriptions of their manufacturing processes, with the downstream workshop, with upstream designers, and with peers in their domain,” he said. “Vericut can automatically create videos and electronic reports of the machining process, which can be easily viewed by others outside the manufacturing department.”

In the latest Vericut 7.2, CGTech has continued to improve Vericut’s ease of use and to improve the simulation process effectiveness, he said. The latest iteration adds features that simplify the most common user actions, Hasenjaeger said, and the company has invested significant developer hours to increase speed by more thoroughly taking advantage of multiple processors and background processing. Another new feature is the ability to optimize NC programs from a saved simulation, he said. “Users can adjust and re-optimize without rerunning the simulation, and a new interactive panel displays cutting conditions immediately, anywhere in the machining process.” ME

Want More Information?

CGTech
Ph: 949-753-1050
Web site: www.cgtech.com

CNC Software Inc.
Ph: 860-875-5006
Web site: www.mastercam.com

Dassault Systèmes/Delmia
Ph: 248-267-9696
Web site: www.3ds.com

Delcam plc
Ph: 877-335-2261
Web site: www.delcam.com

DP Technology Corp.
Ph: 805-388-6000
Web site: www.dptechnology.com

Geometric Technologies Inc.
Ph: 480-222-2255
Web site: www.geometricglobal.com

Gibbs and Associates
Ph: 805-523-0004
Web site: www.gibbscam.com

Siemens PLM Software
Ph: 800-498-5351
Web site: www.siemens.com/plm