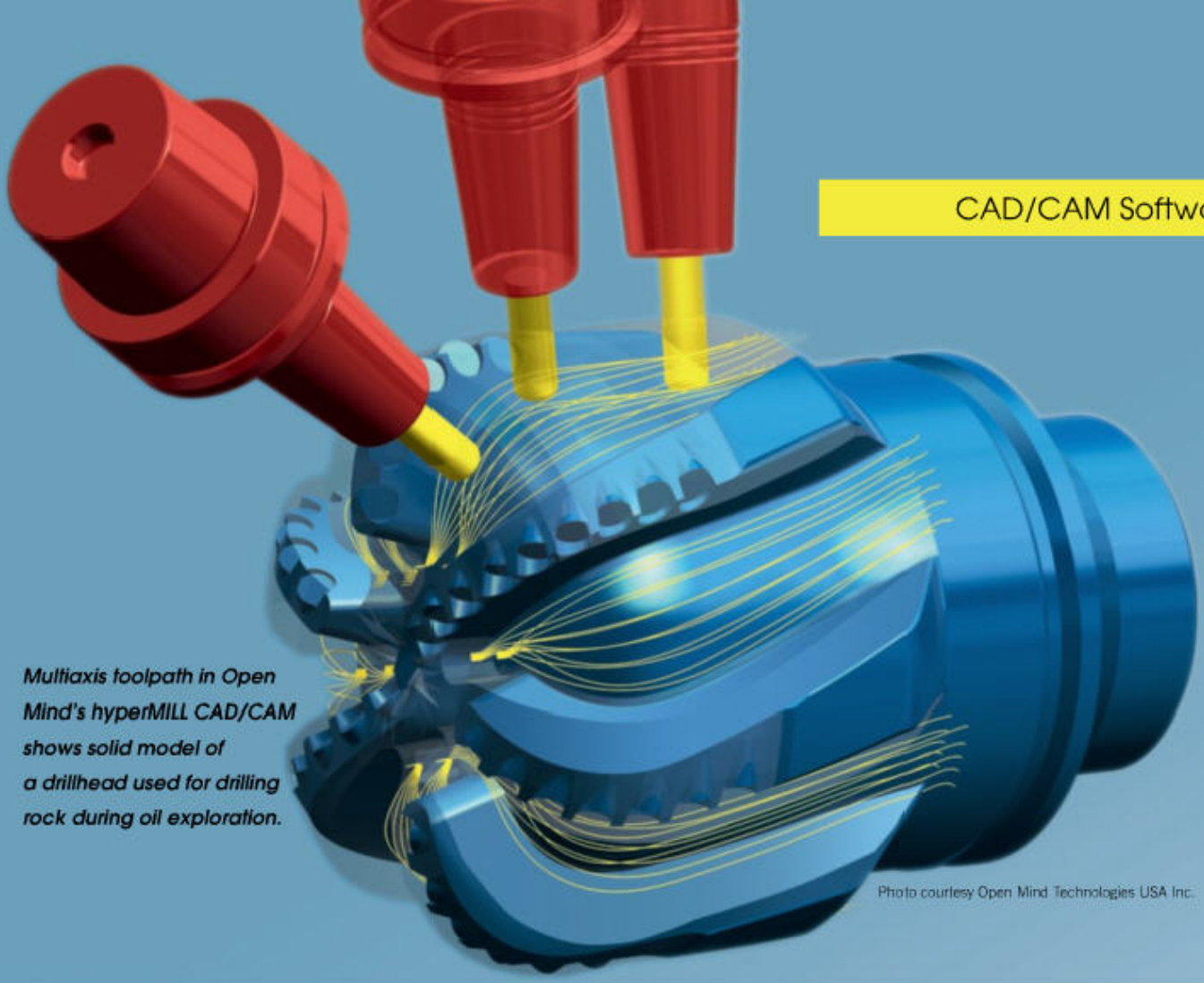

Multiaxis toolpath in Open Mind's hyperMILL CAD/CAM shows solid model of a drillhead used for drilling rock during oil exploration.

Photo courtesy Open Mind Technologies USA Inc.

Refining CAM Toolpath Production

Latest CAD/CAM solutions offer faster programming of increasingly complex machines

Patrick Waurzyniak
Senior Editor

To weather rough economic times, manufacturers seek the fastest, most efficient methods for programming factory-floor machines. With advances in new CAD/CAM and simulation solutions, manufacturers can employ the latest programming techniques to wring the most efficiency from multiaxis machines on the shop floor.

Automating programming tasks in the most efficient manner offers manufacturers a way to level the playing field and boost the bottom line, with CAD/CAM users employing the newest tools to streamline the process of programming multiaxis and multitasking metalcutting machines, as well as NC systems for cutting composite materials. "Today, our company is looking to offer a much higher level of automation," notes

Alan Levine, managing director, Open Mind Technologies USA Inc. (Needham, MA, and Wessling, Germany), developer of the hyperMILL CAD/CAM software package. "The automation is available through high-quality machining instructions with robust collision detection and avoidance that allow the programmer to get it right the first time, give meaningful feedback to the programmer, and make high-quality results come out of the software with a minimum of programming effort.

"We do a lot of work on the automation in the programming process," Levine says. "We want to ensure that we save time and money on the machine, and also on programming labor through utilities such as feature recognition, tool

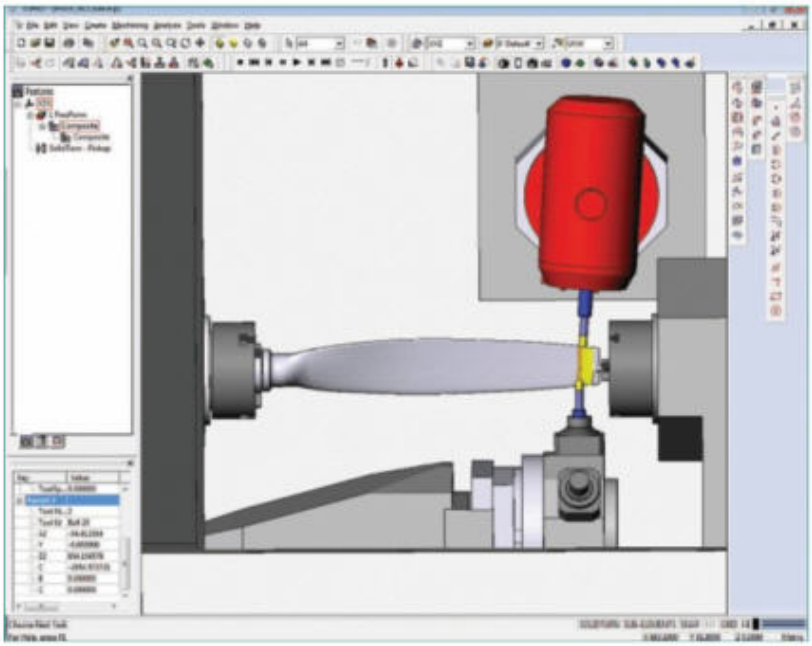

Photo courtesy DP Technology Corp.

New pinch-milling technique in DP Technology's Esprit 2011 CAD/CAM software balances out stresses on the workpiece, resulting in less deflection and greater material removal for faster cycle times.

database, and programming process database. Some shops produce prototypes and the programming tasks are driving the costs, and in another case it's a production shop, and it's the piece prices that are important."

At IMTS last year, Open Mind debuted its 64-bit version of hyperMILL that included a continued emphasis on adding more automation capabilities to the software that is used for 2.5 to five-axis milling with in-house-developed postprocessors. "There have been improvements at the toolpath end that everybody understands, smoother paths and much shorter retract distances, and smart retracts that are doing interference checks on connections," Levine notes. "In some cases, we can save 30–35% in roughing cycles just because of the way we approach a part, rather than the way we cut the part."

In addition, Open Mind's milling package now features a new optimization strategy for within its stock roughing, called hyperMAXX, employing the VoluMill toolpath layout engine licensed from Celeritive Technologies Inc. (Cave Creek, AZ). Reducing air time aids the standard roughing cycles in hyperMILL. "The hyperMAXX results offer meaningful further time savings on most parts. Roughing is critical for two purposes; one is time savings, and the other is leaving a nice stock condition for subsequent finish operations. You want to control remaining stock; if you have 0.020" [0.508-mm] stock allowance, but there's a few places where it's 0.080" [2.0 mm], then when you're finishing, you're going to get some funny noises, wear out cutters, risk damage to parts, and have degraded surface finish."

In five-axis machining, which is Open Mind's focus, the company offers its own in-house-developed machining strategies and postprocessors. "One thing we can do, which has a great benefit, is that during large five-axis calculations we can also predict not only cutter fit, but determine if it doesn't fit, and how to find a new position where it does fit," Levine adds. "We can end the calculation by stating and predicting the customer-desired minimum cutter stick-out length.

"In five axis, where everything's leaning and angled and you have to manage the head and the holder, some people don't know whether to stick out the cutter by an inch or 1.25" [25 or 31.75 mm], but with a little checkbox, we will make that calculation for them. The benefit is based on very simple mechanical engineering beam theory—if I can shorten that stick-out by 10%, I've increased the cutter stiffness by about 28–30%, so shortening a cutter is going to increase the life of the cutter, the feed rates you can achieve, and more. By having the software do it, the engineer doesn't have to try it and calculate how it crashes—we'll do that for them, with the appropriate safety buffers. The prediction of stick-out length may be 1.2014" [30.5-mm], but we will provide an engineering safety as the user will want a little more room than for the molecules of coolant."

New analysis tools in Esprit 2011 CAM from DP Technology Corp. (Camarillo, CA) are designed to help CAM programmers isolate problems with models that can negatively impact

machining, particularly in multiaxis and mill-turn machining operations. "We're working toward identifying any imperfections in the design geometry, because a slight imperfection in the design geometry that's not necessarily visible or pertinent from a design or FEA point of view can have a big effect on the surface finish on a machine tool," states Chuck Mathews, DP Technology vice president. "Those tools are designed to identify any discontinuity in the surfaces, any tangency issues, so we can correct them before we create the toolpath."

In Esprit 2011, the new tools—the zebra, porcupine, heat map, and toolpath analysis commands—provide graphic displays for the inspection of solid models. "A lot of the small imperfections are not visible while using traditional shading techniques," Mathews notes, "but with these techniques they become visible, and it stops you from cutting a workpiece that ends up being unacceptable because it's got a slight mark on it, maybe where the tangency is not quite continuous, for example. There are a whole series of healing tools that we use after this in order to correct the problems—you use our surface modeling tools to go ahead and repair the surfaces where you want."

For five-axis machining, particularly on five-axis mill-turns, the tools are critical due to mill-turn machines' sensitivity to geometry quality, he notes. "We've had the correction tools in there forever, but we wanted to place them on the front side to make it easier for them to demonstrate where the CAM guy feels there's a problem on the geometry," Mathews explains. "As we run the five-axis machines faster and faster and push the limits of the five-axis machines, machining becomes more and more sensitive to the quality of the geometry. It's also related to the mill-turn, when you do five-axis milling on a mill-turn machine, because of the kinematics; because of the configuration of those machines, they are more sensitive to imperfections in the geometry than a traditional five-axis machine.

"On five-axis mill-turn machining, imperfections occur on almost every model, but the level of sensitivity is going up tenfold," he adds. "A few years ago, we were talking

about gross errors that were relatively obvious, and those are all handled and the fixes work really well—everybody's improved their software. Now, it's like we've used a magnifying glass and we're looking 10-100 times closer at geometry. We're talking about problems in the geometry that are actually hard to identify without really good tools."

A five-axis pinch-milling technique also is new in Esprit 2011, Mathews adds. "Where in the past we might have used two lathe tools, and done a balanced turning operation where you've got the shaft, a tool on the top and a tool on the bottom, and you're doing material removal with both simultaneously, now we're talking about doing the same thing, but it's five-axis cutting," he explains.

"You've got a five-axis cutting operation on the top, and a five-axis cutting operation on the bottom—it's what we call five-axis pinch milling. It's almost like research, because there are very few existing machines that can do it. The limit to the machine is three axis in one place and five axis in the other. This is a type of milling where we're using two milling cutters simultaneously on opposite sides of the workpiece. There are some cases where it balances out the stresses on the workpiece, so you don't get a lot of deflection, and it's doing about twice the amount of material removal, so you're reducing cycle times.

Closer development with cutting tool manufacturers is a trend in CAD/CAM, says Bill Gibbs, president, Gibbs and Associates (Moorpark, CA), a subsidiary of Cimatron (Givat Shmuel, Israel), noting recent efforts with tooling companies such as Iscar and Sandvik, among others. "Recently, we have started working much more closely with Iscar, partially because Iscar is an Israeli company and Cimatron is an Israeli company. Of course, all the tooling manufacturers work closely with all of the CAM companies, but it's just an indication of the fact that we're not developing tooling and cutting science, so we want to partner with the companies that are in that field, to enable our software to best represent the latest technologies."

For example, Iscar has long been a specialist in the advantages of grooved-insert style turning. "Instead of just plunging with grooved tools on a lathe part, you actually turn it sideways with grooved tools. Gibbs was one of the first companies to develop machining cycles that followed the Iscar methodology for this kind of machining," he remarks. "What we've done recently is we've gone back to Iscar and we're having them review our cutting methods to match improvements they've made with their inserts and their own studies, so we're updating our grooved-tool roughing technology to keep up with the latest Iscar advances."

Development work continues on Gibbs' MTM (multitask machining) software, which the company demonstrated at IMTS. The rewritten MTM software will be available near the end of this year, Gibbs notes, with the redesigned package offering reduced time to develop postprocessors, generate CNC programs, and simulate machine motion.

Process trends in CAM continue to move toward single-setup cutting, reduced machine time, and better off-the-machine finishes, notes Mark Summers, president, CNC Software Inc. (Tolland, CT), developer of Mastercam. "The recent economic climate pushed these demands even more strongly, as shops needed to trim costs and make more with what they had," Summers says. "Direct machining of solids is—and will continue to be—a huge driver in CAM development. We're also seeing a surge in multiaxis machining, with many shops opting for this technique as it offers more long-term flexibility."

Last year, CNC Software also acquired the sole software assets of Jurasoft, developer of SylvieXpert, the Swiss-made CAM software, Summers notes. "We have spent the time finishing up the 2011 release of our renamed Mastercam Swiss Expert for the existing customer base, and have started to release this to our European customers," he says. "We have been making changes to the product to better support the US market due to the slight difference in programming styles." Swiss Expert will be demonstrated at PMTS and OrthoTec in the US, as well as in Switzerland, with the program being rolled out through certified resellers throughout the year.

Reuse of shared technologies across its CAD/CAM products enables Delcam plc (Birmingham, UK, and Windsor, ON, Canada) to offer users a wide breadth of functionality, according to Glenn McMinn, president, Delcam North America. "CAM customers are quite sophisticated," he says. "They require support across five-axis machining, HSM, multitasking, and machining solid models, while also requesting interfaces that fit their style of programming and specific machines.

"One of these shared technologies is the use of solid stock models to optimize toolpaths," McMinn notes. "It's important to be able to machine from solid models, but this is fairly common now. Delcam has been focusing on the use of solid-stock models to represent each machining stage and optimize toolpaths relative to those models. Different aspects of this functionality are available in both PowerMILL and FeatureCAM."

A number of CAM programs incorporate strategies based on the extent of cutter engagement to give more consistent loading on the tool and to allow higher feed rates, he adds.

Tighter integration of software, controls, and machine tools is a trend, notes Vynce Paradise, of Siemens PLM Software (Plano, TX), developer of NX CAM. "This is driven by the industry as vendors seek to provide increased value to customers in manufacturing by providing a complete solution," he says. "A preconfigured combination of CAD, NC programming, postprocessing, and simulation, along with the controller and machine tool, provides the customer with an optimized solution that helps obtain maximum value from the investment. We refer to this concept as the CAD/CAM/CNC Process Chain.

In its latest NX 7 CAM, Siemens introduced a multiaxis module targeted at the programming of multibladed rotational parts, such as impellers or blisks for turbine engines. "This module differs from traditional CAM software in that the functions put the user in the appropriate context for this kind of programming," Paradise explains. "The terminology, the menu commands, the required inputs and options all relate to the machining of these complex, multibladed components."

Improved NC simulation in Dassault Systèmes' (Paris) Delmia (Auburn Hills, MI) brand V62011x solutions added last year saves NC programmers time and effort. With its V6 solutions, Dassault's Delmia software has made programming machine tools much easier, notes NC Kishore, industry leader, channel solutions, Dassault Systèmes/Delmia.

"It's not just part programming anymore. It's programming of the machine tool," Kishore says. "We have made the machine simulation capability up-front in the part programming phase, so the NC programmer has access to an entire machine, with the kinematics defined, with a virtual controller, and the postprocessor. This helps simulate machine motion up-front rather than at the end, and it significantly reduces the number of iterations and lead-time."

In V62011x, Delmia has enhanced its mill-turn capability, Kishore notes. "It's not just a 2.5-axis lathe programming, it supports complex multiturret mill-turn operations with part transfer simulations. A couple of other usability aspects very welcomed by users is the immersive environment. In V5, we used to have a bunch of workbenches for a specific application; now in V6, we've consolidated them into one single workbench, called Machine Programming."

Complex machines demand simulation for visualizing machine processes. "Manufacturers continue to adopt complex machines," notes Bill Hasenjaeger, product manager, CGTech Corp. (Irvine, CA), developer of the Vericut simulation/verification/optimization software package. "A few years ago, two-spindle, two-turret mill-turn CNC machines were not very common. But today many manufacturers use multitasking machines with multiple tools and part-mounting stations on various motion axes, all crammed into a relatively small space, with several things moving at once. The chance for collisions between machine components, tools, and parts is very high."

Fueling this trend has been the steadily falling cost of such machines, Hasenjaeger notes. "Even small and midsized shops that previously would not have considered buying a multitasking machine now have to learn how to set up and program these complex machines," he says, "so accurate NC program verification and machine simulation becomes a mandatory tool. Skipping the verification step creates costly production delays where the machine tool waits and NC program errors are corrected."

More customers are also cutting exotic materials, such as complex composite structures, he adds. "Due to the extremely high cost of each part prior to machining, customers machining these composite structures benefit greatly from a defect-free NC program. Machining is usually the last operation on a composite part. There is a long sequence of costly processes to get the part ready for machining including: material trimming

or precutting; molding, forming or layup; autoclave baking or curing; multiple inspections; and multiple part transfers. By the time machining occurs, the time and money invested in the part is significant. Mistakes are not an option, and rework or repair is usually not allowed. All this cost is often not the most critical factor—the lead time to get to the machining stage is usually so long that any mistakes completely ruin the delivery of the entire project or product, not just the one part."

In February, CGTech released Vericut 7.1, adding some 400 improvements to user interaction, simulation, cutting tool creation, and CAD/CAM integration. "For several years, we've been programming and simulating automated fiber-placement CNC machines, and we've also started programming and simulating autodrilling and fastening machines. Understanding these other types of automated machinery helps us understand and assist with the customer's complete process. Whether cutting composites with a milling cutter, a waterjet, an ultrasonic knife, or other process, Vericut can reduce the chance of a very expensive mistake late in the production sequence." **ME**

Want More Information?

CGTech Corp.
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

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

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

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

Open Mind Technologies USA Inc.
Ph: 888-516-1232
Web site: www.openmind-tech.com

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