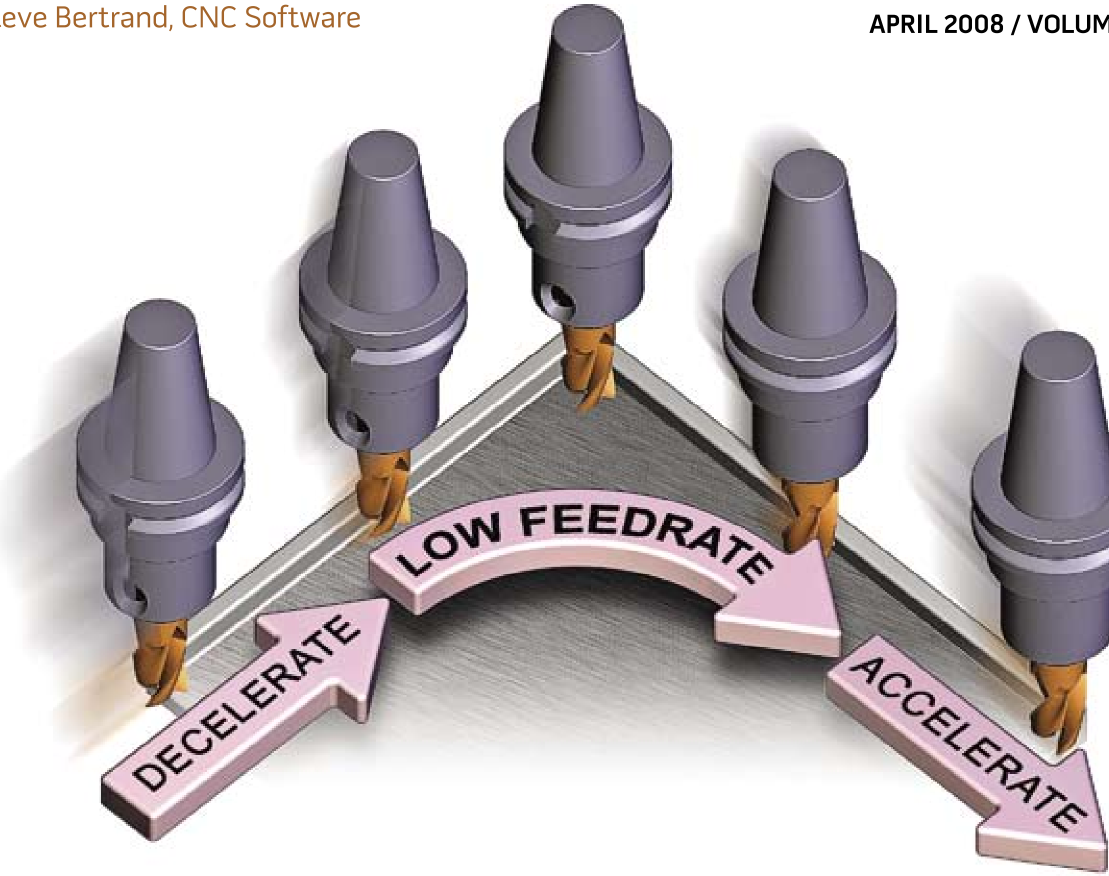
By Steve Bertrand, CNC Software

APRIL 2008 / VOLUME 60 / ISSUE 4



All images: CNC Software

Figure 1: In high-speed CAM strategies, if the tool is approaching a corner, software will slow the tool down. When the tool gets back on a straightaway, it will increase the feed rate up to the maximum allowable speed based on the user-defined feed rate cap.

New Rules of Engagement

New CAM strategies for high-speed machining of hard materials.

Remember the old World War II movies? Combatants met each other head-on with maximum impact. Enemy battalions marched with pavement-whacking goose steps. Whenever a superior dismissed a soldier, the soldier saluted crisply, turned on a dime and departed, making sure he squared his corners at every turn. In today's military, these old-fashioned movements have been exchanged for fast, efficient and surgical actions.

The same can be said for toolpath strategies to machine hard materials. Maximum tool engagement, squared corners and big, frequent retracts and clearance moves—though still prevalent—are gradually giving way to new rules that include minimal WOC and DOC, few or no square corners, high spindle speeds and high feed rates.

These types of cutter movements are typically called high-speed machining (HSM) toolpaths. HSM typically aims to reduce or eliminate sharp corner moves, provide proper look-ahead, optimize acceleration and deceleration (Figure 1), and employs spindle speeds of 12,000 rpm and higher.

This nomenclature is somewhat misleading because these strategies can also be used with a variety of moderately fast spindles (10,000 to 12,000 rpm). Users of this equipment who have adopted high-speed (i.e. high-productivity) machining strategies enjoy a significant advantage over competitors who either have not figured out the benefits of this approach or have been reluctant to depart from traditional machining practices.

Many Advantages

HSM first became popular with shops making parts from hard materials

New Rules of Engagement *(continued)*

(40 HRC to 50 HRC and above). The practice was to use conventional high-engagement tool moves to rough metal in a soft state, heat treat the parts and finish them in near-net shape condition. Conventional finishing of hardened materials frequently left visible scallops, or cusps, that required additional cleanup. If high-precision geometry and finish were required, a sinker EDM was often used from the get-go. This introduced additional operations and additional costs.

The advent of high-speed spindles, better controllers that can rapidly process large amounts of data and sturdier, more rigid machines that are specifically designed for the rigors of high-speed motion made it possible to both rough and finish the part in its hardened state in one setup and eliminate heat treating after roughing. The use of small tools at high speeds and the prevention of over-engagement of the tool (which varies based on toolpath) reduced or eliminated scallops and the need for cleanup. Many shops now use HSM to eliminate EDMing and the need for extensive polishing.

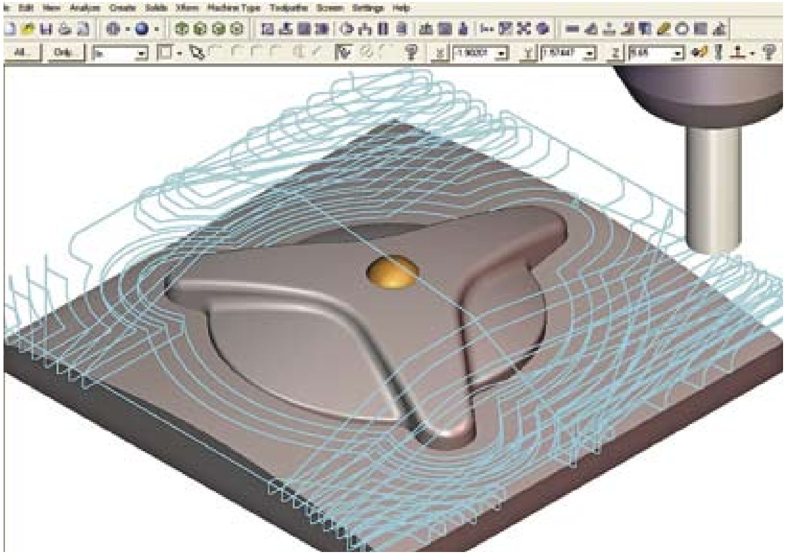


Figure 2: Parts that are shaped like cores (protrusions or bosses) will use a core roughing strategy that starts from the outside of the part and works inward.

As the practice has evolved, HSM has allowed manufacturers to:

- eliminate in-process heat treating,
- reduce or eliminate EDMing,
- produce more precise geometries and surface finishes,
- reduce secondary operations,
- use faster machining cycles due to faster and more precise tool movements with fewer operations,
- improve tool life,
- reduce scrap, particularly in precision parts manufacturing, and
- reduce stress on CNC equipment, preserving the machine's capabilities while lowering maintenance costs.

The last three items have all been achieved largely through reduction or elimination of sharp corners and the reduction of air cuts.

With more CAM-generated toolpaths available for HSM, CAM applications specialists and early adopters of HSM are using it for more than just superhard materials. HSM is also a better way to machine traditional parts.

Fast Moves

While all high-speed CAM strategies must deal with the same theoretical machining considerations, various CNC machines behave differently when a directional change—such as when machining a square corner—is encountered. That is why the "no square corner" rule (reducing or eliminating sharp, angular tool motion) is so important and why every high-speed tool move has a lead in—a radius—that allows for optimally fast, continuous tool movement no matter what type of machine it is used on.

Centripetal acceleration is also a key factor in creating effective HSM toolpaths. If a tool goes too fast around a curve, it will tend to swerve, so algorithms have been devised to keep the tool moving as quickly as possible without going off course.

Other important technical considerations include rigidity of the tool, workholder and workpiece. For optimal high-speed cutting performance, the tool should be as short as possible and the workholder and workpiece rigidly constrained.

Optimal manufacturing performance is a combination of software strategies, tooling and machine capabilities. CAM software vendors have devised HSM strategies to manufacture parts faster with less wear and strain on tooling and equipment. The following are some of the most powerful high-speed toolpaths currently available.

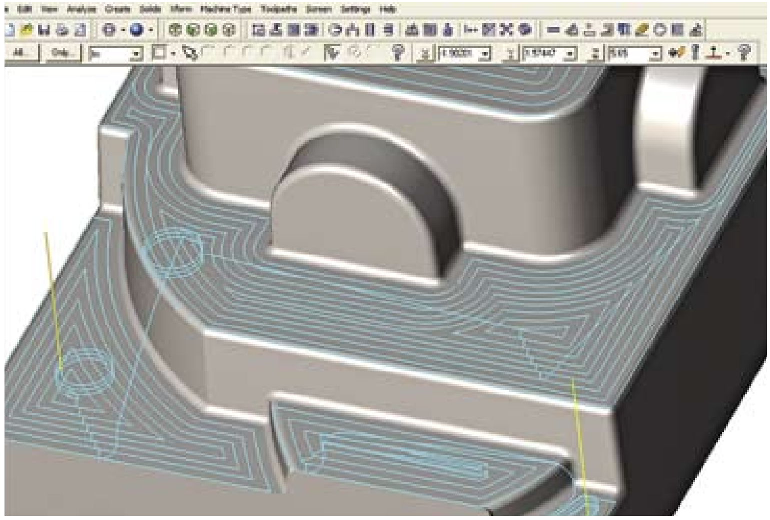


Figure 3: Horizontal milling uses a flat-nose or bull-nose tool to clean flat areas.

Roughing. HSM takes the workpiece down to near-net shape faster than traditional, lower-speed roughing and eliminates the need for one or more semifinish operations. High-speed roughing is typically done with a constant Z strategy (X-axis and Y-axis movements only) as opposed to finishing that may use both constant Z and 3-D motion, depending on the part geometry. Most roughing operations are performed using the following three toolpath strategies, either separately or in combination.

- *Core roughing.* Parts that are shaped like cores (protrusions or bosses) will use a core roughing strategy that starts from the outside of the part and works inward (Figure 2).
- *Area machining.* If there is an internal pocket within the geometry, the area cutting strategy works from the inside out using a helical or ramp entry so the tool doesn't plunge into the part. For parts with both a core and a cavity, it is preferable to use an adaptive machining strategy that will adjust the cut method from outside-to-inside to inside-to-outside, depending on whether it is cutting a boss or a cavity.

> *The use of small tools at high speeds and the prevention of over-engagement of the tool (which varies based on toolpath) reduced or eliminated scallops and the need for cleanup.*

- *Rest roughing.* When geometries are more intricate, a "rest roughing" strategy may be employed. Rest roughing refers to cutting the "rest" of, or remaining, material. This strategy relies on applying a larger tool to remove a majority of the material at high speed and a smaller tool for semifinish passes that remove remaining material the first tool could not reach.

Finishing. There are more toolpaths for finishing than roughing because of the need not only to efficiently produce intricate geometries, but also to eliminate additional semifinish operations on the machine tool and secondary finishing and polishing operations off of it. As with roughing, finishing toolpath nomenclature varies from vendor to vendor. The following are some of the options available to parts manufacturers for high-speed finishing:

- *Constant Z.* An effective strategy for machining steep walls. This capability may be combined with options that allow undercuts or ignore the normal surface direction, greatly simplifying the process.
- *Horizontal area machining.* Uses a flat-nose or bull-nose tool to clean flat areas (Figure 3).
- *Constant scallop.* Maintains a specific, uniform scallop on 3-D-type parts. This is preferable because it provides consistent pressure on the tool.
- *3-D parallel.* A single-direction 3-D parallel cutting motion.

- *Pencil (cleanup).* Constant Z toolpaths tend to leave chunks of material between adjacent surfaces. A pencil toolpath cleans these areas, thereby leaving a smooth radius between the surfaces (Figure 4).
- *Helical cutting.* This technique cuts a 3-D contour in a single, ramping cut, so the processing involves more continuous tool engagement.
- *Trochoidal motion.* This repetitive looping toolpath motion minimizes tool burial. It allows an operator to use the tool's full flute length and is particularly efficient when slotting.
- *Finish blends or morphs.* This is the ability to go from one geometry to the next, such as from a square to a circle, by blending the toolpaths to maintain a constant scallop. This eliminates the need to use another operation to remove excess material left at the transition.

www.ctemag.com

For more information, visit "Articles Archive Index" and enter the keyword "CAM."

For the most part, HSM toolpaths serve obvious purposes and the user can learn them intuitively. The real competitive advantage for manufacturers is learning how to use management tools within the CAM software to automatically apply the most appropriate set of strategies and experimenting to develop settings for families of parts.

Going to the Library

As with other projects, manufacturers learning HSM should spend some time in the library—an electronic repository of tool settings and toolpath strategies that have been fine-tuned and used successfully in their operations. Manufacturers can build libraries of optimal speeds, feeds, DOCs and step-over distances for given tools based on the workpiece material. These tool settings can be associated with specific HSM strategies.

The starting points for a tool library are the conservative values provided by cutting tool vendors. For example, in Mastercam's CAM Applications Laboratory, we determine how far an operator can push a tool using high-speed toolpaths when cutting specific materials by increasing the feeds and speeds and adjusting DOCs until the tool breaks or the finish degrades. Using this information, we establish theoretical HSM limits.

In Mastercam programs, default tool settings for HSM may be as much as 300 percent lower than upper-limit settings determined in the lab, so users can be confident they are starting with safe values. Then, settings can be adjusted to reduce machining time.

Several variables determine how fast a machine can run without sacrificing accuracy, including tool length, workpiece rigidity and machine tool accuracy. Shops typically must experiment with machining two or three parts to

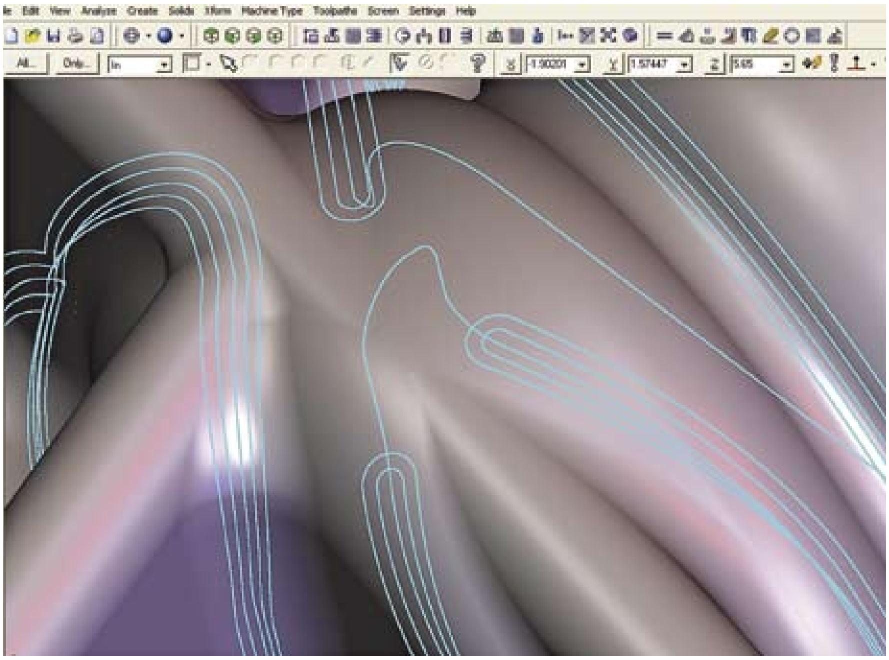


Figure 4: Constant Z toolpaths tend to leave chunks of material between adjacent surfaces. The pencil toolpath will clean these areas, leaving a smooth radius between the surfaces.

determine the best tool and settings for a particular material and type of geometry. When the user is satisfied with the tool's performance, he can save the strategy and use it again for similar parts. This is an effective way of generating efficient toolpaths for part families that are consistent from user to user. If you have a part for which you've proved out all the toolpath specs, you can immediately apply those proven toolpaths to a similar part, saving programming time.

Feed Rate Optimization

If new HSM toolpath strategies allow manufacturers to run CNC machines faster without compromising quality or damaging tools and equipment, how can they make sure they are using all of the toolpaths' capabilities? Most manufacturers manually tweak feed rates within safe, self-imposed limits to protect against part and tool damage. Many of these same users could improve machining output by 10 percent to 25 percent or more if they use a more scientific approach and adjust feed rates based on the volume of material being removed at different points in the machining process. Some intelligent CAM software can do this programming automatically by looking forward along the toolpath and anticipating the amount of tool engagement.

For example, if the tool is approaching a step or a corner, the software will slow the tool feed rate (Figures 1 and 5). When the tool gets back on a straightaway, the software increases the feed rate to the maximum allowable based on a user-defined feed rate cap.

If the end user wants to go all out, he will use this cap on a benchmark procedure that generally takes several hours to perform. This procedure can increase productivity over the life of the machine. However, if the user doesn't have a few hours to spare for benchmarking, he can set the upper feed rate limit to a more conservative, safe value based on the CNC machine builder's published specifications. Even this conservative procedure, which requires no additional CAM programming time, will increase productivity.

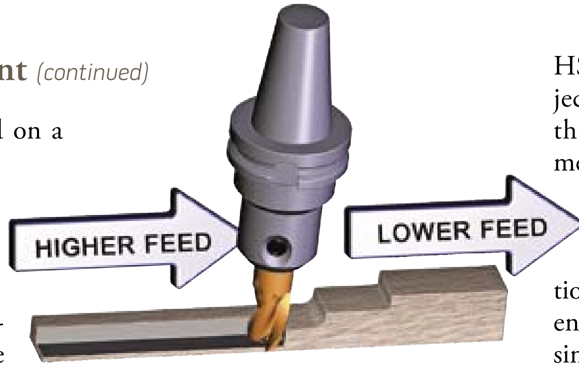


Figure 5: In high-speed CAM strategies, if the tool is approaching a step, software will slow the tool feed rate down. After the step, the CAM software will increase the feed rate to the maximum allowable.

Marching Orders

Almost all manufacturers can benefit from HSM. Many toolpaths have been devised by CAM software developers to help users take advantage of these productive and economical manufacturing technologies. More are on the way.

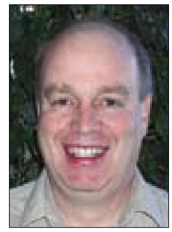
With more manufacturers beginning to use HSM, there are three simple rules to follow:

- Reduce or eliminate sharp corner moves,
- Use smaller tools and run at the higher feed rates recommended by the toolmakers, and
- Optimize feed rates based on material-removal volume.

Even with conventional equipment, manufacturers can shave minutes and possibly hours from each job.

Many CAM software vendors offer HSM toolpath options. While the objectives and even the names given to these movements are similar, implementation can vary. Before selecting a software tool for high-speed CAM programming, discuss the toolpath creation and file management options available with several vendors and engage them in benchmark tests that simulate parts production. A good strategy allows you to advance more rapidly toward the goals of increased manufacturing output, improved part quality and reduced tooling and maintenance costs. CTE

About the Author:



Steve Bertrand is corporate sales manager, CNC Software, Tolland, Conn. Prior to his current position, Bertrand was senior applications engineer.

For more information about the company's Mastercam CAM software, call (800) 228-287 or visit www.mastercam.com/ap.