APRIL 2004 / VOLUME 56 / NUMBER 4

MILLING THE 'HARD WAY'

BY GARY ZUREK,
MIKRON BOSTOMATIC CORP.

A clear understanding of the factors involved leads to successful hard milling.

One machining advancement that has taken hold over the past few years is hard milling. Typically, mold and die makers perform hard milling to cut P-20, H-13, S-7 and other tool steels.

These materials range in hardness from 45 to 64 HRC and are traditionally electrical discharge machined. But new technologies make hard milling a viable alternative.

Successful hard milling requires several components to come together—the machine tool, toolholders, cutting tools, CAD/CAM system and process "know-how."

Machine Factors

The machine tool is the most significant component. The most fundamental aspect of the machine tool is that it must be designed for hard milling and have the same characteristics found in a high-speed machining center. The machine's base construction and individual components, such as the drive train, spindle and CNC, must be capable of handling the demands of hard milling.

The base construction must be extremely rigid and have a high degree of damping abilities. These characteristics are found in machine tools with bases constructed from polymer concrete. These machines typically have six to 10 times the damping characteristics of machines with cast iron bases. Additionally, polymer concrete has excellent mechanical and thermal characteristics.

The machine tool's drive train should incorporate digital drive technology for optimal acceleration and deceleration. This technology allows the CNC to perform a high degree of contouring accuracy and gives it excellent dynamics capabilities.

One of the most overlooked components is the spindle. The spindle must be able to provide a great deal of flexibility, offering high torque at low spindle speeds and maximum power for a large range of spindle speeds. An ideal spindle's speed ranges from 100 rpm to 20,000 rpm or higher, depending on the application. Hybrid ceramic bearings in the construction of the spindle increase spindle stiffness, accuracy

All images: Mikron

Mikron's HSM 400U 5-axis machine is designed for hard milling, which has similar requirements as high-speed machining.

and temperature stability.

The interface between the spindle and the toolholder should always be HSK. This has proven to provide minimal runout and excellent balancing at high speeds. Furthermore, HSK provides a rigid and balanced tooling setup compared to a taper interface.

The performance of the CNC is also critical. A control with the maximum block-processing rate ensures that the data it receives is handled quickly and efficiently. This data should reside on a hard drive located on the CNC. Incorporating a numerical algorithm to calculate the velocity profile in the control assists in smoothing machine motion.

All servo systems on a CNC machine exhibit a characteristic called "servo lag." Servo lag is the actual amount that the machine position trails the position commanded by the control. In hard milling, any motion that is not continuous with the programmed toolpath creates excessive stresses, which leads to premature failure. Therefore, it is essential that the CNC have the capability to handle and control servo lag.

Hold On

Another important aspect is the interface between the toolholder and the cutting tool. The cutting tool can be held by several methods, including collet chuck, hydraulic expansion, power shrinking and heat shrinking. The selected method is determined by the requirements of the machining operation (see Table 1).

Collet chucks are, by far, the most flexible. In addition, they are easy to handle, provide excellent shock-absorbing characteristics and offer a range of clamping diameters. They are ideal for aggressive roughing and semifinishing of hardened materials.

Hydraulic-expansion toolholders provide ease of use, high clamping forces and minimal runout. However, they can be expensive and bulky to use. Hydraulic, as well as power-shrink, toolholders are excellent for roughing and semifinishing. Shrink-fit toolholders provide excellent capabilities for finishing hardened cavities and cores with a high degree of accuracy and fine surface finishes.

VG = very good G = good S = satisfactory P = poor	Runout	Radial stiffness	Handling	Flexibility	Transferable moment	Application: Roughing	Application: Semifinishing	Application: Finishing	Cutting tool extension	Speed limitation	Cost (including equipment)
Collet type	S	S	G	VG	S	VG	VG	G	S	G	VG
Hydraulic expansion	G	S	VG	VG	G	VG	VG	VG	G	P	S
Power shrinking			G	S	G				VG	VG	P
Heat shrinking	G	G	S	S	VG	S	VG	VG	VG	VG	P

Table 1: Comparison of toolholding systems.

Cut It

One of the main contributors to successful hard milling is the cutting tool. Many companies tend to skimp when it comes to cutting tools. A toolmaker that specializes in cutting tools for hard milling or offers a well-defined product line for hard milling is the best source.

For roughing hardened materials, endmills with four or more flutes are recommended. These provide small chip loads while having the capability to cut at higher feed rates. Additionally, toroid endmills are recommended for roughing because the sharp edges of conventional endmills do not provide sufficient resistance against vibration and thermal stress when cutting hardened materials.

The cutting tools should be short with short flute lengths and have a helix angle of approximately 30°. A 30° helix has proven to be optimal for chip flow and dispersal of heat.

The carbide substrate should also be considered. Only carbide tools with fine or ultrafine grain sizes, about 0.5μm to 0.6μm, should be used. These tools provide increased edge strength and reduce built-up edge.

For milling larger hardened cavities and cores, cutting tools with inserts should be considered. Carbide inserts are less expensive than solid-carbide endmills, and by indexing the insert, tool life can be extended. However, these tools are typically not designed for high spindle speeds and runout can be significant. There is also a significant safety risk if improperly handled.

Hard milling puts a great amount of stress on the cutting tool from high heat and abrasive wear. To help overcome

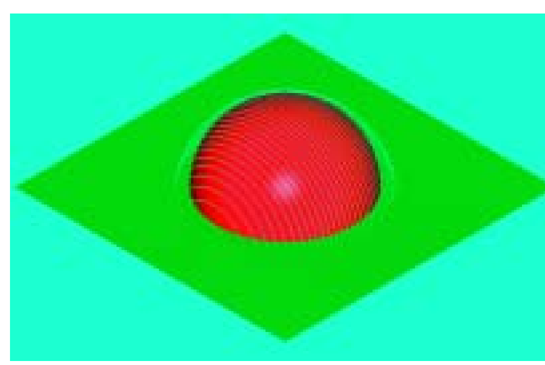


Figure 1: Single parallel finishing toolpath strategy.

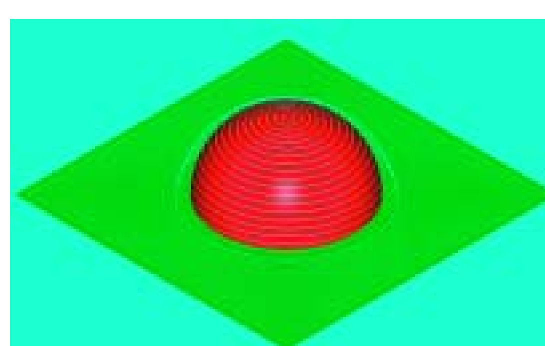


Figure 2: Two-part parallel finishing toolpath strategies.

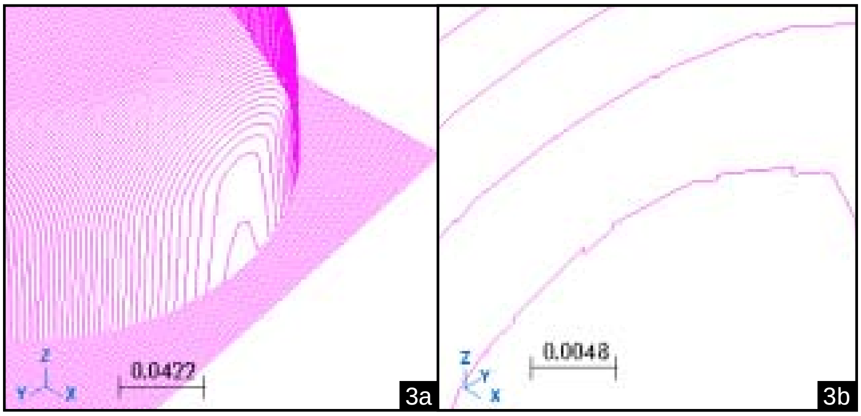
these stresses, coated cutting tools must be specified. Coatings offer a protective layer on the tool, substantially increasing tool life.

The most common coatings are titanium nitride, titanium carbonitride, titanium aluminum nitride and titanium aluminum carbonitride.

Coating selection should be made based on individual properties. Titanium-based coatings, such as TiCN and TiAlN, are the most common for hard milling. The wear resistance, or its hardness, is the most important property of TiCN, while TiAlN resists heat and oxidation better. The toolmaker may further enhance its coatings by offering unique multilayer blends.

Flood coolant is not commonly used in hard milling. Hard milling often generates a tremendous amount of heat, which is transferred into the chips and causes the coolant to vaporize as it hits the hot chips. The use of coolant can also create thermal instability with the cutting tool.

To help displace chips during cutting, compressed air is used. Additionally, a combination of oil and mist is often selected. Oil helps reduce friction, thereby increasing tool life and improving surface finish. When using oil and mist, an extraction unit should be integrated into the machine tool to help remove the oil from the air.



Though the toolpath shown in 3a looks normal, a closer view (3b) shows unnecessary directional changes.

CAD/CAM Analysis

The CAD/CAM system is another important component. CAD/CAM systems have greatly advanced over the years, and now provide a variety of advanced features and capabilities. However, not all systems are created equal and there are still many that do not have the capabilities to create toolpaths for hard milling.

Although no CAD/CAM system is designed exclusively for hard milling, many of the systems that offer HSMing capabilities have the same strategies for hard milling because the two are related. When hard milling, strategies that keep the cutting tool in motion should be used. This ensures the tool is continuously cutting with a constant chip load, which is one of the more desirable conditions to maintain when hard milling.

Before toolpaths can be applied, a complete analysis of the part must be performed. Not all parts are suitable for hard milling. The specific areas to be machined should be clearly identified, determining the smallest internal radius and largest working depth. A tool with a 4:1 length-to-diameter ratio commonly does not pose any problems.

Problems arise when the ratio grows. When ratios are excessive, hard milling experience plays an important role in determining how successful one is. Hard milling with cutting tool diameters as small as 0.005" is possible as long as care is taken to maintain a constant chip load and machine at minimal DOCs. DOCs from 0.0002" to 0.0005" are common with such small tools.

As mentioned, it is important to keep the cutting tool in motion and avoid dramatic changes in direction when hard milling. Therefore, multiple toolpath strategies may be required to complete the part, depending on its complexity.

The process of recognizing and separating the key areas of the part and applying different toolpath strategies is commonly called "modular toolpath programming." MTP is generally used to maintain high cutting speeds. Similarly, MTP can help keep the tool in motion while avoiding dramatic changes in direction.

In its simplest form, Figure 1 illustrates a single parallel finishing toolpath strategy. Although simple, this is not the ideal method for machining this part in its hardened state.

If individual part features are recognized and separated, two different strategies can be applied to this part, as shown in Figure 2. In this simple example, a spiral morph toolpath on the green surface, combined with a true spiral from top to bottom on the red surface, provides a suitable machining method.

Toolpath quality is commonly overlooked in a CAM system. Figure 3a represents what appears to be a normal-looking toolpath. But upon closer inspection, it is revealed that there are many unnecessary changes in the direction, as shown in Figure 3b.

Control manufacturers have incorporated elaborate acceleration and deceleration servo-tuning algorithms and complex servo-lag algorithms (look-ahead features) into their controls to enhance motion control. These look-ahead or motion-control features control feed rates by analyzing directional changes within the NC code. The greater the directional change, the more the control has to slow down to maintain the programmed toolpath. In hard milling, these abrupt changes in toolpath direction, as shown in Figure 3b, create dwells and slow downs, which can affect tool life and surface finish. Therefore, toolpath quality should be an important feature of a CAM system.

With hard milling, programming errors have severe consequences if not caught in time. Cutting tools can easily be broken. Toolholders, fixtures and even the machine tool can be damaged, costing hundreds or thousands of dollars. Personal safety can also be at risk.

To ensure programming errors are caught before they happen, the NC code should be thoroughly reviewed. Most CAD/CAM systems incorporate some type of toolpath verification or simulation. Unfortunately, many of them view only the intermediate file rather than the posted NC code or the toolpath file where errors can occur. Therefore, the posted NC code, and not just the intermediate file, should be reviewed for errors.

If a CAD/CAM system does not have the tools to verify or simulate the NC code directly, there are numerous software packages on the market that can. These products cost a few hundred dollars to several thousand dollars, but they can save major problems by eliminating potential crashes and safety issues at the machine tool.

Know It All

Finally, proper know-how is vital to successful hard milling. All of the necessary components are of no use without knowledge of the principle processing procedures. Successful hard milling is based on specific know-how, advanced knowledge HSMing, proper choice of cutting tools and clamping systems, and using a HSM-capable CAD/CAM system.

A clear understanding of all the components provides better awareness of what is needed to be successful at hard milling.

About the Author

Gary Zurek is applications engineering manager at Mikron Bostomatic Corp., Holliston, Mass. For more information about the company's machine tools call (508) 474-1100.