JANUARY 2008 / VOLUME 60 / NUMBER 1

➤ BY TRAVIS BIGGS AND RON MALONE, GREENLEAF CORP.

At Quality Industries, a 1.5"-dia. positive-geometry Excelerator endmill from Greenleaf is tooled with three RPGN-43 WG-300 inserts.

B. Kennedy

Not So Hard

New tooling technology and application expertise take the difficulty out of hard milling.

Given enough time and money, it is possible to machine any workpiece material. Most shops, however, don't have the luxury of unlimited lead time and unrestricted resources. Competitive pressures put a premium on productivity and economy, spurring a continuing effort to cut machining time and reduce the number of operations required to complete a part.

Hard milling is a way to achieve those productivity goals. Careful application of metalworking equipment, tooling and strategies can make hard milling less difficult than it first appears.

Hard Jobs

Hard materials (for this discussion 45 HRC and harder) include tool steels, mold steels, chilled and chrome irons, P/M, weld overlays and some nickel- and cobalt-base superalloys. Manufacturers use hard materials to extend wear life and maintain precision in molds, gears, tools and dies, aerospace components and processing equipment.

Unfortunately, the characteristics that make hard-material parts perform so effectively also make the materials more difficult to machine. Bringing a hardened part to size traditionally has been slow and tool consuming.

Facing hard facing

Greenleaf Corp. introduced its WG-300 ceramic composite tool material about 25 years ago. The ceramic matrix is reinforced with silicon-carbide “whiskers” that boost toughness. Initially applied in the relatively constant cutting conditions of turning operations, the composite tools provided a significant increase in productivity when roughing aerospace alloys. Shops are now using WG-300 tools in hard milling operations.

Tom Mahusky, shop supervisor at Cleveland Hard Facing Inc., Cleveland, is familiar with machining hard materials. “We’ve dealt with hard materials for the past 20 years, so we’ve learned a lot of tricks,” he said.

A recent job put that knowledge to use. It involved hard facing and machining hammer components used to forge titanium parts. To enable the four roughly 20"×16"×8" L-6 tool steel hammers to survive the high temperatures and pressures of the forging process, hard facing specifications called for application of a 1/8"-thick layer of low-carbon steel followed by a 7/8"-thick layer of nickel-base Hastelloy and finally a 1/4"-thick layer of Waspaloy, another nickel-base material. Cleveland Hard Facing applied the layers using the metal-inert-gas arc welding method.

B. Kennedy

This pyramid-shaped part is one of four hammers used to forge titanium alloys.

B. Kennedy

Quality Industries vice president Jim Kaplan (from left) and president Jerry Kaplan consult with Greenleaf sales and service engineer Denny Carpenter regarding milling of forged components featuring one layer of low-carbon steel and two layers of high-temperature-alloy welded hard facing.

After the welding was completed, Mahusky rough milled the parts. The hammers had pyramid-shaped tapered faces, so the cutting tool had to machine through all of the weld layers as it descended the sloped sides of the hammers. “Hastelloy and Waspaloy don’t like to be cut,” Mahusky said. The alloys are not unreasonably hard; “only going in the low 30s [HRC] as welded,” he said, but added that when machined, the materials “will workharden just like a 300-series stainless.” This increases the hardness by 10 HRC. Without a large-capacity CNC mill in his shop, Mahusky used a 7 1/2-hp manual vertical mill to rough the welded overlays. Running at 1,400 sfm and a feed rate of 18 to 19 ipm, he took DOCs of up to 0.100" with a 4"-dia. Excelerator milling cutter tooled with four round, negative-geometry WG300 ceramic inserts. “A positive insert is not strong enough to withstand roughing in this case,” he said.

Mahusky worked with Denny Carpenter, a Greenleaf sales and service engineer, to set up the operation. Carpenter made sure the machining parameters were “within the capabilities of the machine, the part and the rigidity of the setup,” Mahusky said. “It depends on the configuration of the part. If it’s big and overhanging, you need as rigid a setup as possible. You have to take all that into account when milling with ceramics.”

Machine stiffness is crucial. “Even if you are taking small finishing cuts, you still need a rigid machine,” he said. “If you are going to take bigger cuts, it’s not just a high-speed spindle you need—you must have rigidity and torque at the spindle.”

Mahusky left 0.020" finishing stock on the hammers and subcontracted finish machining to job shop Quality Industries Inc., also in Cleveland.

Quality Industries finished the hammers on an ACRA FVMC-610 CNC vertical machining center using a 1.5"-dia. positive-geometry Excelerator endmill tooled with three RPGN-43 inserts. The tool ran at 3,800 rpm, 80 ipm and a 0.020" DOC. Vice President Jim Kaplan said the light DOC produced a minimal flow of chips. Greenleaf’s Carpenter added that “compensating for the chip thinning effect is 100 percent crucial” in maximizing tool life in the operation.

Mahusky said he would not consider hard milling without ceramics. “Not today, when you know what you know, and you remember how you did it back when. You’d get carpal tunnel syndrome just changing inserts when it was carbide.” He added that according to a sales representative for the weld materials applied to hammers, some shops don’t attempt to mill the hard welds and grind them instead. “That’s gets pretty expensive because you are getting into wheels, too,” Mahusky said.

—CTE Staff

The traditional approach is to rough and semifinish parts before hardening via heat treatment if possible, then tediously finish them with round tools, inserts, grinding or EDMing. Parts that begin as hard materials or those where hardening would affect final precision, require slow processing right from the start. Long cycle times, multiple operations and part setups and excessive work-in-process boost manufacturing costs.

One way to reduce those costs is milling parts in the hardened state. The moldmaking industry has been a pioneer in hard milling, applying hard yet tough micrograin carbide cutting tools, usually with coatings that increase heat resistance and boost lubricity. Hard milling with carbide tools typically involves small WOCs and DOCs, generating relatively low cutting forces and a small chip. Typical cutting speeds are moderate to slow. They reflect the hardness of the material being milled, falling from perhaps 500 sfm on materials around 45 HRC to 200 sfm or slower on materials harder than 60 HRC.

In the last few decades, the ability of ceramics to retain their hardness at high cutting temperatures has enabled them to replace carbides in many machining applications. For most ceramics, however, increased hardness is accompanied by a decrease in toughness. That tradeoff has limited ceramics' use in interrupted cutting operations, such as milling.

Recently, however, ceramic tools are seeing increasing use in milling applications. Tool development efforts have produced ceramics with increased toughness, such as those composed of alumina-titanium-carbide and silicon-nitride materials, as well as ceramics reinforced with silicon-carbide "whiskers." (See sidebar on page 64.)

As with carbide tools, cutting speeds for ceramics when milling vary by workpiece hardness. Speeds for ceramics are higher across the board than carbide because the heat generated by high speeds facilitates ceramic tools' cutting performance. The heat actually softens the workpiece material ahead

Ceramic tools are seeing increasing use in milling applications. Tool development efforts have produced ceramics with increased toughness, such as those composed of alumina-titanium-carbide and silicon-nitride materials, as well as ceramics reinforced with silicon-carbide 'whiskers.'

of the cutting edge and lowers cutting forces. As a result, a typical cutting speed for ceramic tools machining 45-HRC materials is about 850 sfm, while the speed for materials over 60 HRC is about 550 sfm. Workpiece material softening ahead of the cutting edge permits larger DOCs and WOCs than possible with carbide tools, enabling ceramic tools to offer productive metal-removal rates when milling hard materials.

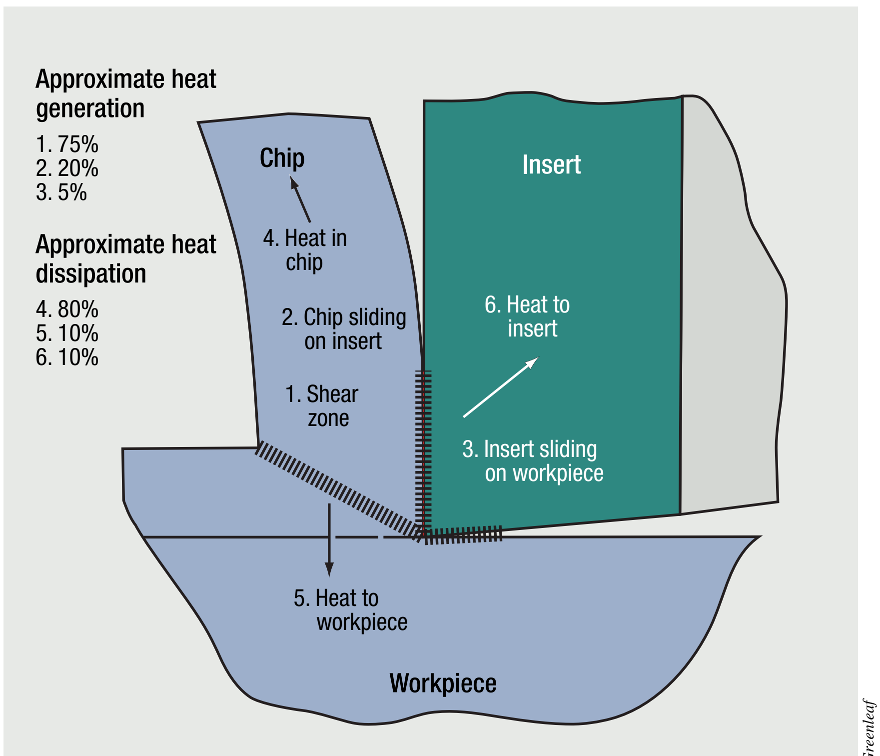
Machining Strategies

Milling with ceramics does, however, require close attention to machining practices and parameters. Rigidity in the machine tool, fixturing and tooling is crucial in any machining operation, and especially so when milling with ceramic tools. Ceramics exhibit maximum strength when cutting forces are applied in compression rather than tension or shear. In addition, abrupt changes in cutting force (caused by vibration or repeated entry into and exit from the cut) will eventually damage the cutting edge.

From a toolholding aspect, rigid systems such as HSK and shrink-fit toolholders, as well as balanced tools, can minimize vibration and runout that shortens tool life.

Certain toolpath programming strategies can increase ceramic tool life in milling. Repeated entry into and exit from the part can damage the cutting edge, so cutter engagement should be as consistent as possible. To maximize engagement of the cutter and the part, manufacturers should use strategies such as ramping, helical interpolation and Z-level milling, in which the cutting tool descends into the workpiece in gradual spirals rather than steps. Within each strategy, care should be taken to limit impact on the tool; for example, ramping should feature a moderate descent rate into the work. In addition to maximizing the time the insert is under compressive force, such programming strategies help maintain the heat at the cut that facilitates the cutting action.

In a milling demonstration involving 52-to 53-HRC P-20 mold steel, reprogramming the toolpath to maximize contact time with the part, with-



When hard milling with ceramics, it is essential to maintain heat in the shear zone between the chip, workpiece and cutting insert while also assuring that excess heat is carried away by the chips. Optimally, the chip should absorb 75 percent of the heat generated in the cutting action.

out changing any other variables, resulted in a 400 percent increase in tool life.

While heat should be maintained in the shear zone between the chip, workpiece and insert, excess heat can be a problem because it can damage or distort the workpiece. It is essential that chips formed in the cutting action carry away excess heat. Optimally, 75 percent of the heat generated should be absorbed by the chip.

Find the Sweet Spot

Retaining sufficient heat in the cut-

Keywords

ceramics:
Cutting tool materials based on aluminum oxide and silicon nitride. Ceramic tools can withstand higher cutting speeds than cemented carbide tools when machining hardened steels, cast irons and high-temperature alloys.

hardness:
Hardness is a measure of the resistance of a material to surface indentation or abrasion. There is no absolute scale for hardness. To express hardness quantitatively, each type of test has its own scale, which defines hardness. Indentation hardness obtained through static methods is measured by Brinell, Rockwell, Vickers and Knoop tests. Hardness without indentation is measured by a dynamic method, known as the Scleroscope test.

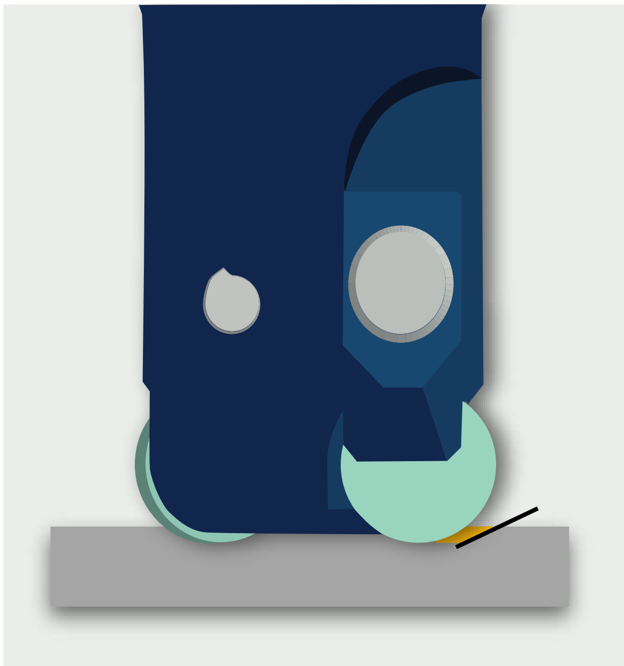
indexable insert:
Replaceable tool that clamps into a tool body, drill, mill or other cutter body designed to accommodate inserts. Most inserts are made of cemented carbide. Often they are coated with a hard material. Other insert materials are ceramic, cermet, polycrystalline cubic boron nitride and polycrystalline diamond. The insert is used until dull, then indexed, or turned, to expose a fresh cutting edge. When the entire insert is dull, it is usually discarded. Some inserts can be resharpened.

—CTE Metalworking Glossary

ting zone while assuring that chips carry away excess heat involves maintaining a “sweet spot” of average chip thickness (average because milling by nature produces chips of varying thicknesses).

A chip that is too thin has insufficient mass to absorb the heat and carry it away. Further, a thin chip may indicate that the cutting edge is rubbing or burnishing the workpiece rather than cutting it. A thin chip also concentrates cutting pressure on the tool’s tip, its weakest point, rather than on the stronger rake face. Such conditions lead to rapid edge wear and result in tool chipping while contributing to work-hardening and to short tool life.

On the other hand, a chip that is too thick carries the bulk of the heat away from the cutting edge, reducing the temperature and generating high tool pressure because of the lack of material softening.



Greenleaf

Controlling chip thinning becomes more complex when applying round inserts. On a round insert, lead angle varies relative to the depth of cut. The shallower the DOC, the greater the effective lead angle, and the thinner the chip produced.

One strategy, programming a cut to produce chip thickness based on axial and radial DOC, does not always achieve the desired result because of chip thinning. Chip thickness is determined by the combined effects of the tool’s axial DOC and radial engagement, as well as its lead angle. For example, a straight-sided insert has a 0° lead angle and produces 100 percent of the programmed chip thickness. A 45° lead angle, on the other hand, spreads the chip over a longer surface and produces only 70 percent of the programmed chip load. To overcome chip thinning and maintain optimal chip thickness, the programmed chip load must be increased using the formula in the next column.

In milling, ceramic tool life can be increased by maximizing engagement of the cutter and the part through machining strategies such as ramping, helical interpolation and Z-level milling.

Controlling chip thinning becomes more complex when applying round inserts. Much ceramic milling is performed with round inserts because round cutting edges are stronger than ones with corners. On a round insert, lead angle varies relative to the DOC. The shallower the DOC, the greater the effective lead angle and the thinner the chip produced.

A formula for calculating average chip thickness when applying a round insert is HM = $\sqrt{(d \div D)} \times F_z$, where HM is the chip thickness, d is the axial DOC, D is the insert diameter, and F_z is the feed rate per tooth. The formula does not take into account every factor in the cutting process, but provides a starting point for determining average chip thickness.

Another factor affecting chip thickness is WOC (radial DOC). If less than half of the cutter diameter is engaged in the cut, the chip thins significantly as the cut moves away from the cutter's center. Increasing the feed rate will help maintain acceptable chip thickness. Typical WOC is 60 to 70 percent of the cutter diameter; maximum insert life will be achieved with a WOC from 40 to 60 percent.

Employing climb milling also can help generate a thicker chip. Climb milling helps develop full chip thickness from the start of the cut as op-

The following companies contributed to this report:

Cleveland Hard Facing Inc.
(216) 883-6414
www.clevelandhardfacing.com

Greenleaf Corp.
(800) 458-1850
www.greenleafglobalsupport.com

Quality Industries Inc.
(216) 961-5566

posed to conventional milling, in which chip thickness is very small at the cut's start and approaches full thickness at the end.

A negative land on an insert's rake face directs forces into the insert's body rather than directly into the relatively weak insert edge, enabling the tool to endure milling's interrupted cutting. Edge hones are applied to the tip of the cutting edge to reduce chipping.

Standard edge preparations for inserts in hard milling applications consist of a narrow land and a light hone. Some applications, such as severe interruptions or workpieces featuring heavy scale, require a larger land. The feed rate per tooth should be larger than the edge hone to ensure that the tool cuts and not rubs the workpiece and that sufficient chip thickness is achieved.

Understanding and controlling the chip thinning phenomenon can boost shop productivity. For example, a 0.020" DOC finish pass with a 0.312"-dia. round insert (RPGN-2.52) at a 0.003-ipt chip load with a 21.4-ipm feed rate produces a low average chip thickness, 0.0007" to 0.0008". That thickness is less than the edge hone typically applied in this application and would probably cause rubbing—which accelerates tool wear—and excessive pressure on the insert's edge. Raising the feed rate to 28.5 ipm makes the average chip thickness greater than that of the cutting edge hone. Elimination of rubbing benefits tool life. In addition, the increased feed rate increases productivity 33 percent.

Use of coolant is usually not recommended because it reduces the heat needed for ceramics to perform most

B. Kennedy

Milling with ceramics can enable a shop to cut machining time and reduce the number of operations required to complete a hard (45 HRC and higher) part.

effectively and reduces tool life. However, recutting chips must be avoided. The solution is high-pressure air, through the tool if possible, to blow the chips from the cutting zone.

High Feed Approach

Cutter body choice for milling with ceramics depends on the operations being performed. In any situation, rigid cutter bodies and secure clamping systems minimize the effect of any runout. For heavy roughing, a cutter with a negative insert presents a strong insert edge to a workpiece. For finishing operations, especially in materials that tend to workharden, high-shear cutters with positive inserts position the insert's cutting edge where it is needed to cut the material rather than push it. High-shear cutters have high positive axial rake geometries.

Cutter choice helps determine the appropriate machine tool for hard milling. Most machine tools can handle basic hard milling operations. However, fully exploiting the capabilities of high-feed milling cutters and advanced ceramics requires substantial horsepower and torque. For example, hard milling with a 2"-dia. cutter at 0.075" DOC in 50-HRC steel would require about 15 hp and 200 ft.-lbs. of torque at 2,165 rpm. Horsepower requirements basically correlate to the cubic inches of metal removed.

Productive hard milling requires a combination of appropriate strategies and equipment as well as expert application of advanced cutting tool materials. With such a combination, a shop can find that hard milling not only is much less difficult, but also increases productivity. △

About the Authors

Travis Biggs is an applications engineer for Greenleaf Corp., which specializes in ceramic cutting tools. He has worked at the company for the past 11 years. Contact him at (800) 458-1850 ext. 2337 or by e-mail at tbiggs@greenleafcorporation.com. Ron Malone is an applications engineer for Greenleaf. He has more than 20 years of experience in the metalcutting industry. Contact him at (800) 458-1850 or by e-mail at rmalone@greenleafcorporation.com.

CTE Contributing Editor Bill Kennedy assisted in writing and editing this article.