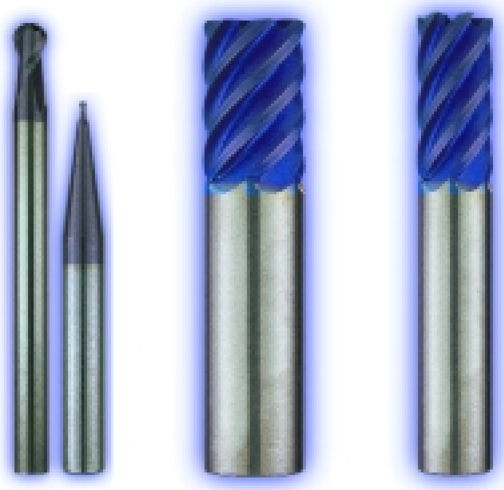
SEPTEMBER 2001 / VOLUME 53 / NUMBER 9

BY MIKE MACARTHUR

High & Hardened



All images: RobbJack Corp.

Figure 1: The three basic cutter designs for machining hard metal (from left to right): ballnose, corner radius (bullnose) and square end.

How to improve productivity when high-speed machining hardened die and mold steels.

While some people feel that the high-speed machining of hardened die and mold steels is a dark art, adhering to some basic principles and guidelines makes it a straightforward, predictable and profitable process. This article discusses the primary aspects of HSM hardened steels, including the effective use of the machine tool, cutting tools, toolholders and programming.

There are three primary machining methods: soft, hard and machining with an electrical discharge machine (EDM). The configuration and hardness of the die or mold steel determine which method, or combination of methods, works best.

Soft machining—machining the part prior to heat-treatment—should be considered when cutting large workpieces or parts requiring deep cuts. After roughing, semifinishing and finishing can be done in the hardened state.

Small parts or parts requiring shallow cuts can be entirely milled in the hardened state. If the part incorporates thin features and requires deep cuts, EDMing may be the only practical option.

Tool Selection

Choosing the proper cutting tool is important when machining hardened steel. There are three basic cutter designs: ballnose, corner radius (bullnose) and square end (Figure 1). Generally, the best choice for hard-metal machining is the ballnose endmill. It's used for roughing and most finishing operations. The tool's large radius dissipates the cutting forces and heat generated while machining hardened steels at high speeds and feeds. Also, the ballnose endmill allows the user to cut closer to the net shape at high speeds and feeds.

If a part incorporates large, flat areas on its floor, a corner-radius tool

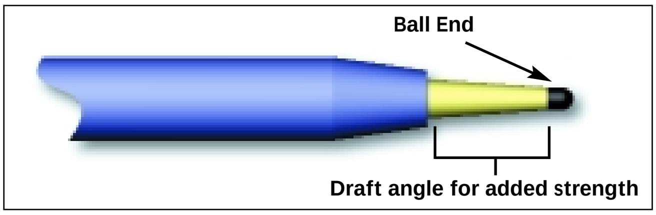


Figure 2: Generally, an endmill should have ½° less draft than the actual part to provide the necessary angular clearance.

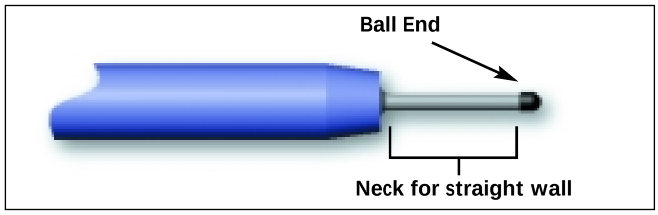


Figure 3: A tool with a neck is recommended for parts with straight walls.

should be used after roughing with a ballnose cutter. The corner-radius tool has a smaller radius than the ballnose and, therefore, does not dissipate the heat and forces as well.

The square-end cutter should be applied where a sharp corner is required, such as at the intersection of a floor and wall. The square-end tool should be used only after a ballnose or corner-radius tool has removed as much material as possible. The square-end tool's sharp corner has a tendency to chip, since it acts as a stress riser—focal point—for heat and cutting forces.

Tool rigidity is important. A small-diameter tool's shank should be much larger than its cutting diameter. This increases cutter stiffness, which helps impart better finishes and extend tool life. Additionally, the tool should not project from its holder any farther than necessary.

When some basic guidelines are followed, high-speed machining of hardened steels is a predictable and profitable process.

The cutting tool should fit the application as closely as possible. Generally, an endmill should have ½° less draft than the actual part to provide the necessary angular clearance, while keeping the tool as strong as possible (Figure 2). For example, if a part has 3° draft, the tool should be modified to have 2½° draft. (This modification can be done easily by the tool's manufacturer.) If the part has straight walls, a tool with a neck can be utilized (Figure 3). A neck strengthens the tool.

Both methods allow a short-length-of-cut tool to cut deeper than its cutting length.

Minimizing Heat

Excessive heat changes the part's surface morphology, which reduces cutting accuracy. However, one way of minimizing heat generation and retention is by controlling the radial step-over of the cutting tool. Radial step-over is the distance between centerlines of successive, parallel cuts (Figure 4).

For roughing operations, the radial step-over should equal 25 to 40 percent of the cutter's diameter. For finishing with a given cusp height on a flat surface, the radial step-over can be calculated with Formula 1 (see box). Cusp height can be found using Formula 2.

Radial step-over determines the length of time each flute spends in the cut, and the length of time it has to cool before re-entering the cut. As such, it determines how much heat accumulates in the tool and part.

When radial step-over is too great, heat builds up in the flute, because

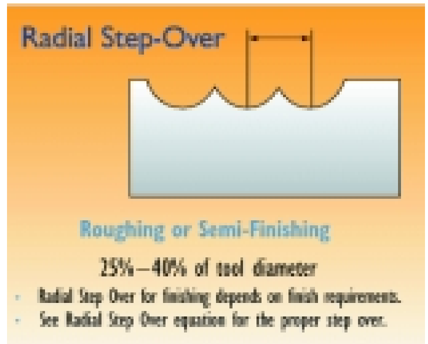


Figure 4: Radial step-over is the distance between centerlines of successive, parallel cuts. A large radial step-over inhibits cooling of the cutter.

there is insufficient time for it to cool before it re-enters the cut. Smaller step-overs provide a continuous cooling action, limiting heat generation and retention.

By limiting heat generation, higher rpms can be achieved without reaching the coating's "fatal" temperature. Once this temperature is reached, the cutting edge rapidly deteriorates, which increases cutting forces and raises the temperature of the tool and part. When the proper process is implemented, there should be no heat buildup in the part.

Additionally, by selecting the proper coating, higher temperatures can be tolerated without compromising the cutting tool. For example, the maximum working temperature for titanium carbonitride (TiCN) is 750° F (400° C), compared to 1,470° F (800° C) for titanium aluminum nitride (TiAlN). Generally, TiAlN is the preferred coating for HSM hardened die and mold steels because it resists heat, enabling the machine to be run at high rpms without damaging the cutting tool.

Proper speeds and feeds also are essential in controlling heat buildup. Large chip loads remove heat with the chip so it does not build up in the tool or part. If the chip load is too light, the

Formula 1: $\text{Radial Step-Over} = \sqrt{4(\text{Cusp Height} \times \text{Tool Diameter}) - 4(\text{Cusp Height}^2)}$
where cusp height is the chordal deviation—finish tolerance

Formula 2: $\text{Cusp Height} = \dfrac{\text{Tool Diameter}}{2} - \sqrt{\dfrac{\text{Tool Diameter}^2 - \text{Radial Step-Over}^2}{4}}$

ROUGHING AND SEMIFINISHING

	RPM		
Diam.	Steels 30–40 HRc	Steels 40–50 HRc	Steels 50–60 HRc
1/32"	20,000-40,000	20,000-40,000	20,000-40,000
1/16"	20,000-40,000	20,000-40,000	20,000-36,000
3/32"	20,000-32,000	20,000-32,000	16,000-24,000
1/8"	15,000-24,000	18,000-24,000	12,000-18,000
3/16"	10,000-16,000	12,000-16,000	8,100-12,000
1/4"	7,600-12,000	9,100-12,000	6,100-9,100
5/16"	6,000-9,700	7,300-9,700	4,800-7,300
3/8"	5,000-8,100	6,100-8,100	4,000-6,100
7/16"	4,300-6,900	5,200-6,900	3,400-5,200
1/2"	3,800-6,100	4,500-6,100	3,000-4,500

FINISHING

	RPM		
Diam.	Steels 30–40 HRc	Steels 40–50 HRc	Steels 50–60 HRc
1/32"	20,000-40,000	20,000-40,000	20,000-40,000
1/16"	20,000-40,000	20,000-40,000	20,000-40,000
3/32"	20,000-40,000	20,000-40,000	20,000-40,000
1/8"	20,000-36,600	20,000-40,000	20,000-30,500
3/16"	20,000-24,000	20,000-32,000	16,000-20,300
1/4"	15,000-18,000	18,000-24,400	12,000-15,000
5/16"	12,000-14,000	14,600-19,000	9,700-12,000
3/8"	10,000-12,000	12,000-16,200	8,100-10,000
7/16"	8,700-10,400	10,000-13,900	6,900-8,700
1/2"	7,600-9,100	9,100-12,200	6,100-7,600

Recommended spindle speeds for ballnose endmills when roughing, semifinishing and finishing. Run at the machine's maximum rpm if the suggested speed is higher than the machine's capabilities.

tool rubs or grinds the part, which leads to heat buildup. Therefore, tool life is extended when producing the largest chip load possible without damaging the tool or part.

Using the largest possible chip load also improves productivity. For example, if the chip load per tooth should be 0.008"—but it's only 0.002"—a part that should take 20 minutes to machine would take 80 minutes. This means the tool would spend four times as much time in the cut as required.

Generally, flood coolant should not be used when HSM hardened metals. Extensive testing by RobbJack Corp. demonstrated that milling metals above 40 HRC without flood coolant increased tool life fivefold, on average. Many methods of coolant delivery were tested: through-the-tool coolant holes, coolant grooves, coolant hoses, and high- and normal-pressure coolant. In every test, tool life was reduced.

The use of coolant leads to a carbide tool suffering thermal shock, which causes a reduction in tool life. However, chips still must be evacuated from the cutting zone to avoid recutting them. Air and mist-application systems are excellent substitutes for flood coolant. The air- or mist-delivery nozzle should be as close to the tool tip as possible.

Replacing Tools

When to replace a tool depends on the user's requirements. Generally, cutter failure can be seen with the naked eye. Look at the tool tip while it's in the cut. A worn tip emits a red glow, which indicates excessive forces and heat (Figure 5). Usually, the glow initially appears in corners or areas where more material is being removed. When the glow is a prominent, continuous red, replace the tool.

The red glow may also indicate problem areas in the part or improper programming. Turning off the light on the machine tool aids in seeing the glow.

Applying tools consistently is also important for making hard-metal machining predictable. Tool tolerance, proper fit in the holders and runout affect performance. The proper tolerances on the tool and toolholder ensure rigidity, accuracy and consistency. To have cutters fit properly in toolholders, including shrink-fit styles, the tolerance on the tool shank should be -0.0001" to -0.0002" of the nominal diameter. This ensures better surface contact between the shank and toolholder.

Although industry-standard tolerances are as high as -0.0005", they result in excessive runout in the holder and an improper fit. In addition, the roundness of the shank should be held to at least ±0.000025".

Runout causes the chip load to increase for one or more flutes, while the other flutes cut too light of a chip—a major problem in hard-metal machining. Shock from runout causes the tool or workpiece to vibrate, leading to chatter and tool chipping. Controlling runout limits the shock introduced to the cutter.

In addition, polished shanks should be avoided. While the tools may be aesthetically pleasing, polished shanks reduce the holder's gripping ability.

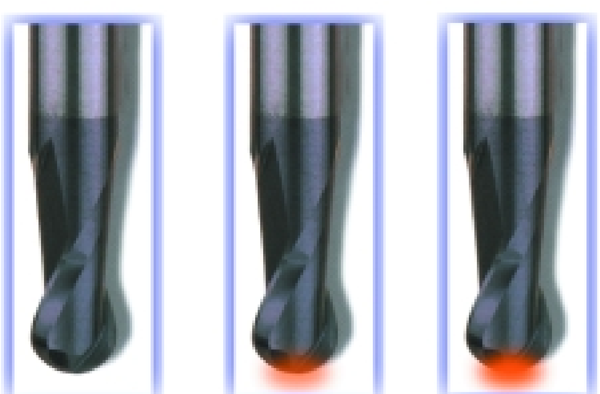


Figure 5: A glowing red tool indicates excessive forces and heat, leading to tool failure.

Machine Tools

The machine tool itself should not be overlooked when HSM hardened steels. Although it is possible to machine hard-metal parts on an older, slower machine, the operation won't be highly productive. If a machine is limited by a low spindle speed, the feed rate suffers proportionally. It is beneficial to have a highly rigid and accurate machine.

The controller of a machine used for high-speed die and mold work must process vast amounts of data. When considering a new machine, give thought to the controller's look-ahead capabilities. To maximize feed rates, the controller must compensate for acceleration, deceleration and spindle growth.

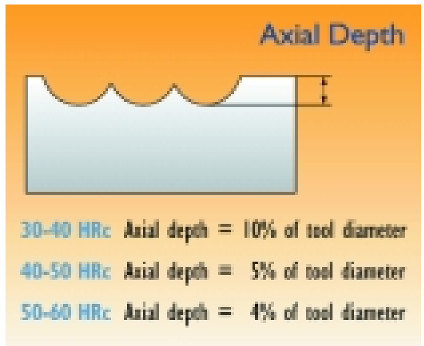
Furthermore, the machine tool should have high throughput capabilities. These capabilities are dependent on the block processing speed, servo response, interpolation rate, resolution of the feedback system, and acceleration and mass.

Programming

Programming determines the way the tool engages the metal and the type of forces that act on the tool. Therefore, programming is a critical element to the success of a HSM operation involving hardened metal.

A cutting tool should be programmed to helically interpolate as it enters a cavity. This minimizes the variations of shock the cutter will experience. Programs should avoid straight plunging (plunging in the Z-axis only). If the programmer cannot set the tool to enter the part from the side or helically, he should program it to ramp in.

The programmer, who also determines the radial step-over and depth of cut, is critical to the HSM process. And, it's important to remember that hard-metal HSM is a process that requires the effective application of the machine tool, cutting tool, toolholder and programming.



Follow established data when programming axial depth.

Each process element must be addressed individually to ensure a predictable, profitable outcome.

About the Author

Mike MacArthur is an applications engineer for RobbJack Corp., Lincoln, Calif. For more information about RobbJack, call (916) 645-6045 or visit www.robbjack.com.