SEPTEMBER 2001 / VOLUME 53 / NUMBER 9

ChipBlaster Inc.

▶ BY JIM ROWE

Right Face

The elements required for successful facemilling.

When facemilling, there are some simple and theoretical principles that must be adhered to in order to ensure a successful, trouble-free operation.

And, before beginning a facemilling operation, you should be clear on your expectations of the process. Is it a roughing operation in which surface finish and dimensional accuracy can vary slightly? Or, will the facemill being used produce the final surface finish or create a machined surface that will serve as a datum for machining other workpiece dimensions?

In addition, you need to choose the right tool and cutting parameters for the job.

This article examines some of the key areas that need to be considered to ensure a successful facemilling operation.

Cutter Body Selection

The price of 4"-dia. facemill bodies can exceed $600, making them some of the most expensive metalcutting tools. So, it pays to know what you're getting for your money.

When selecting a facemill, an important consideration is the number of inserts the body can accept. This is the cutter's pitch. For example, a 4" coarse-pitch cutter might have six inserts, while a 4" fine-pitch cutter might have eight. The pitch determines the number of inserts engaged in the cut at one time. Each manufacturer has its own coarse- and fine-pitch facemills, since the size and orientation of a manufacturer's inserts dictates the area available for insert pockets.

While roughing, which involves deep cuts and creates heavy chip loads, the cutting forces can overwhelm the rigidity of the spindle and setup, creating excessive chatter and shortening tool life. Chatter, in turn, can cause carbide inserts to crack. Cracks can quickly destroy an insert and, possibly, lead to catastrophic tool failure.

A coarse-pitch cutter helps reduce the required horsepower and cutting forces that are generated when more inserts are in the cut simultaneously. Lighter-duty spindles, such as R-8 and 30- and 40-taper spindles, can effectively handle a coarse-pitch cutter in a roughing operation.

Coarse-pitch cutter bodies work great for roughing because they have larger chip gullets. This extra gullet space provides more chip clearance, which is needed because of the heavier chip load per tooth. If a chip were to get caught in

J. Rowe

Shown are fine-pitch (left) and coarse-pitch cutters. The pitch determines the number of inserts engaged in the cut at one time.

the gullet due to a lack of clearance, the chip could get recut and cause the insert to fracture or the cutter body and workpiece to rub.

When finishing, lighter depths of cut (from 0.010" to 0.025") and lighter chip loads per tooth (from 0.002" to 0.006") are typically used. Since finishing does not require as much horsepower as roughing, it's possible to apply a fine-pitch cutter body. The extra inserts of the fine-pitch cutter allow higher table feeds, even though the chip load per tooth is light, and the smaller chip gullets are suitable for the amount of metal being removed during finishing.

With larger, more stable spindles, it's possible to rough with fine-pitch cutters. Since the fine-pitch cutter has more inserts in the cut, horsepower and rigidity requirements increase when cutting with heavier DOCs (from 0.050" to 0.200") and heavier chip loads per tooth (from 0.006" to 0.012").

Successful roughing with a fine-pitch facemill requires an adequately sized gullet. Chip evacuation needs to be verified for the DOC and chip load, because if a heavier and thicker chip gets caught in a gullet, it could cause serious problems when it's recut. Obviously, recutting chips should be avoided at all times.

Insert Selection

Base insert selection on the type of facemilling operation to be performed. Sometimes a molded insert is desirable, while at other times you should choose a ground insert.

You can rough successfully with a molded insert. And because it costs less than the ground insert, you'll save money.

Molded inserts don't have the accuracy or as sharp of cutting edges as ground inserts. However, molded inserts are stronger, since their cutting edges are honed. The honed edge prep withstands the shock of higher feeds and the deeper cuts made during roughing. Molded inserts sometimes have chipbreakers to help equalize cutting forces, reducing the amount of friction between the insert and workpiece and the amount of horsepower required to cut metal.

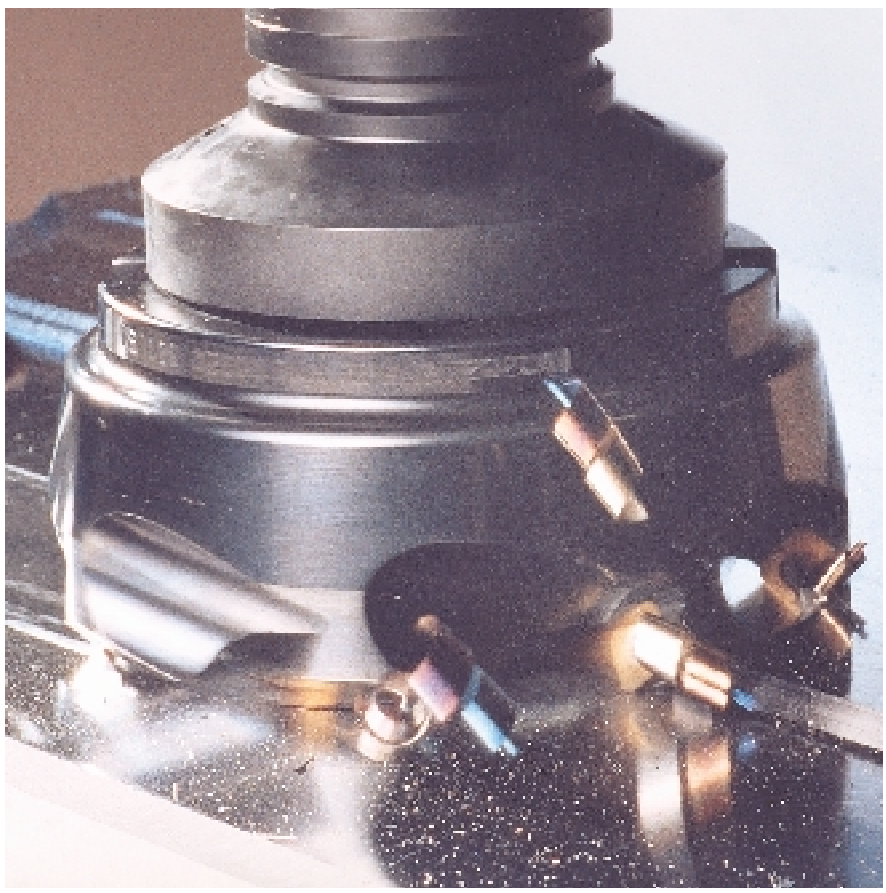
Typically, the molded insert doesn't impart as consistent a surface finish or provide the dimensional accuracy that a ground insert does. Since the molded insert can vary dimensionally, the cutting height of the inserts in the facemill body can vary as well. This is another reason why they cost less to produce and purchase. Molded inserts work great in certain applications, but you get what you pay for.

For finishing, ground inserts are the better choice. Their greater dimensional accuracy results in a more precise alignment of the cutting edges of the inserts in a facemill cutter, which results in a more dimensionally accurate workpiece. The closer these inserts cut on the same plane, the smoother the measurable surface finish will be. (A profilometer can be used to measure surface finish.)

Sharp, ground edges tend to make a "finessing" cut, yielding a smoother surface finish. Inserts without sharp edges tend to smear the surface.

As with molded inserts, ground styles can incorporate chipbreakers. Ground inserts for finishing tend to have very sharp rake angles in their chipbreakers, creating a high-positive cutting edge. These rake angles allow the inserts to cut at lighter feeds and lighter DOCs. Without sharp rake angles, carbide inserts run at light feeds and DOCs and tend to rub the workpiece, which shortens tool life.

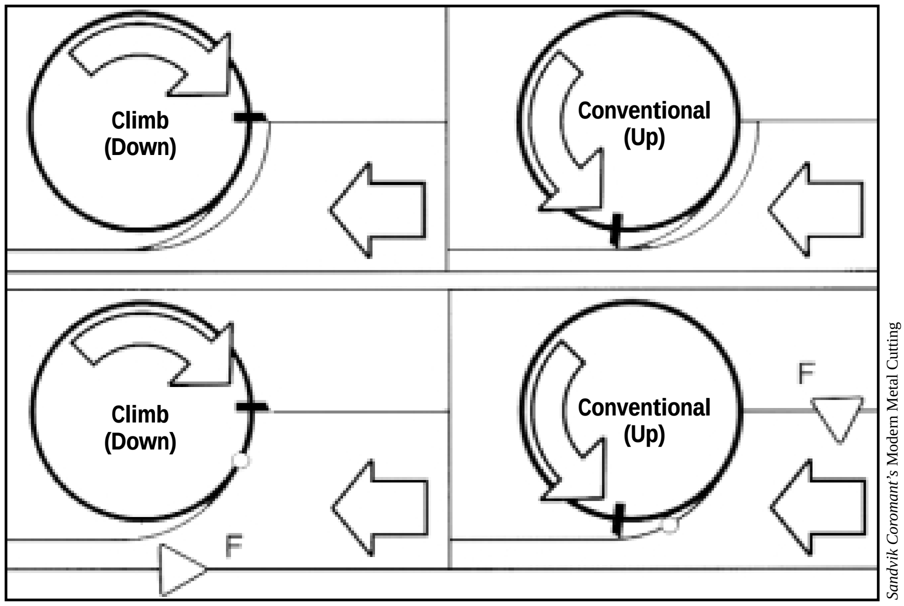
Ground inserts can successfully face-



The goal when cutting dry is to adjust the speed and feed so that the heat transfers to the chip—not the workpiece or cutter.

mill gummy materials, such as stainless steels. The shearing action provided by the sharp edge reduces the friction between the insert and work material, allowing the chip to release from the insert quicker.

As a compromise between molded and ground inserts, you could install molded inserts in the majority of the cutter's pockets and place wiper inserts in the remaining pockets. A wiper cleans up some of the roughing marks and imparts a better finish than using molded inserts exclusively. In addition, wiper inserts can reduce cycle times and costs. Wiper technology is advancing in all machining applications, including turning, grooving, cutting off and drilling.



During climb (down) milling, the workpiece feed direction is the same as that of the cutter rotation at the area of cut. When conventional (up) milling, the workpiece's feed direction is opposite that of the cutter rotation.

Coolant, Coatings

One of the biggest debates surrounding facemilling is whether or not coolant is needed.

When a large-diameter facemill is used, it's difficult to flood the entire cutter and cutting zone with coolant. And since milling is inherently an interrupted operation, the inserts are continuously entering and exiting the cut. If the coolant is directed at the point where the inserts are actually cutting, coolant may not reach the insert when it exits the cut.

This rapid heating and cooling of the insert can cause thermal fracturing. If an insert fractures and falls from the pocket during cutting, the cutter body can suffer serious damage.

Today's tool coatings facilitate dry facemilling by minimizing the chance of thermal fracturing. Certain coatings, such as TiAlN, use the heat generated in the cutting zone to chemically transform themselves so they harden immediately upon entering the cut.

An advantage of running dry is that the operator can see the chip's actual form and color, which provides critical information about what is happening in the cut. Since materials have different chemical compositions, they react differently to heat.

When machining carbon steel, for example, just-formed chips look dark brown when the proper speed is applied. When the heated carbon is exposed to oxygen, a chemical reaction transforms the brown chip to a desirable blue color. If the chips are black, they are too hot and the cutting speed should be reduced.

Stainless steels have low thermal conductivity, so heat doesn't transfer to the chip well. When machining stainless steels at the proper surface footage, it's common to see chips with a light tan tint. If the chip color is dark brown, be careful because you are reaching the top end of the speed range.

Some heat is needed in the cut while machining stainless steel to prevent built-up edge. Chips that cool too quickly—as can happen when coolant is used—will fuse to the insert, which leads to recutting of chips and shorter insert life.

Too high a feed can also cause heat-buildup problems. And, interestingly, too low a feed can cause the tool and workpiece to rub, which also leads to excess heat.

The goal when cutting dry is to adjust the speed and feed so that the heat transfers to the chip, not the workpiece or cutter. So turn the coolant off, adjust the spindle speed and feed, and watch the hot chips fly.

Hot chips are a good sign, because they mean the heat is not affecting the part or tool. Tool life will be satisfactory, since thermal fracture isn't occurring. But remember, when machining a flammable material, such as magnesium or titanium, it's a good idea to apply coolant and keep a fire extinguisher nearby.

Lastly, when cutting dry, it is important to apply a small amount of an anti-seize compound at the screw/cutter body interface. Don't get sloppy with the compound, though, because if it's smeared on the back of the insert the insert might not sit in the cutter pocket properly. Apply according to the tool manufacturer's recommendation.

Climb vs. Conventional Milling

Most facemilling jobs performed on light-duty machines with excessive play in their lead screws or ballscrews dictate conventional (up) milling. However, you should climb, or down, mill whenever possible. You will have more success at facemilling if you do.

There are several reasons for this. With conventional milling, the insert starts out with no effective chip load, causing it to rub before cutting. Rubbing kills a carbide insert and work-hardens the part for the next insert.

When climb milling, line up the cutter to take a radial WOC equal to about two-thirds of the cutter diameter. This ensures that the cutting action begins the instant the insert engages the workpiece, without any rubbing.

Try adjusting this radial WOC and identify which cutter-diameter-to-radial-WOC ratio works best on a particular machine with that cutter. The surface finish will change along with this radial adjustment, because of the “radial swing” the insert is traveling and the cross-hatching that results at a specific feed rate.

Be aware that if you take a radial WOC that is less than half the cutter diameter, the insert will begin to rub because the chip thickness has diminished. At this point, the so-called “radial chip thinning factor” comes into play.

Radial chip thinning is the effect of taking a WOC less than half of the cutter’s diameter. The calculated feed per tooth will diminish as the radial width decreases, resulting in a lighter actual fpt. This causes the carbide to rub the workpiece, reducing tool life. Carbide likes to cut, so increase the fpt as the radial depth decreases.

If you are roughing and the radial WOC is less than half the cutter diameter, increase the feed. Tool life will improve and cycle times will decrease. When finishing, the required surface finish will, obviously, dictate the feed rate.

Measuring Success

The success of a facemilling operation can be measured many ways. One is by determining the cubic inches of metal removed per minute (the metal-removal rate). The mrr also helps determine the horsepower required for cutting, which can help you make the correct decisions before you actually cut anything.

Take the WOC and multiply it by the DOC, and then multiply the resulting number by the feed rate in inches. For example:

$$3'' \text{ WOC} \times 0.150'' \text{ DOC} = 0.450 \text{ sq. in.}$$
$$0.450 \text{ sq. in.} \times 35 \text{ ipm} = 15.75 \text{ ipm}^3$$

The mrr is 15.75 ipm^3—which is a bunch of metal. Whether your machine can handle it depends on the metal’s hardness factor. Aluminum has a hardness factor of about 0.3. If you multiply the mrr of 15.75 by 0.3 you get 4.725, which is the required horsepower to cut that metal. But if you were cutting 4140, which has a 0.7 factor, you would need a machine with more than 11 hp. Most tooling manufacturers can supply these hardness factors.

For successful facemilling, ensure the starting cutting parameters are within the machine’s capabilities before you begin. This reduces the time spent at the machine making adjustments for the specific metal.

About the Author

Jim Rowe is an application salesman for Hi-Tech Tooling & Supply, Fort Wayne, Ind.