JUNE 2001 / VOLUME 53 / NUMBER 6

BY RANDY CLOUD

Bore On

From the spindle to the cutting edge, advice on adjustable boring heads, boring bars and inserts.

"The inserts I bore with are causing me all sorts of problems with breakage, chatter and part quality. What can I do?"

Customers often ask me this question during technical support calls about insert-style boring bars. The most misunderstood boring bar item is the common screw-down carbide insert, especially one with a chipbreaker geometry.

End users seem to rely on the axiom, "If a little bit is good, then a whole lot has got to be great!" They believe that if a chip-control insert is effective for OD hogging, then it *must* be good for boring.

While chip-control inserts work well for OD turning and facing, the chipbreaker tends to "get in the way" when boring holes, especially holes less than 1.0" in diameter. In most cases, the machinist will be able to—and should—drill the hole to be bored within 0.030" of its final size. This leaves a depth of cut of about 0.015" per side, just right for a finishing pass with a boring bar.

But because most chip-control inserts are designed for a minimum DOC exceeding 0.020", the chip-control insert is not only ineffective but it can cut the wrong size hole, alter the roundness and impart a poor surface finish.

A typical chip-control insert has the chipbreaker geometry pressed into the "green" carbide cutter before it's sintered, or baked, in a vacuum chamber. (Some geometries are ground in after sintering.) Pressed-in chipbreakers result in a slightly rounded cutting edge. At a light DOC, this rounded edge causes the insert to push away from the workpiece being bored and then dive in again, resulting in chatter, an uneven surface, and an out-of-round and/or tapered hole.

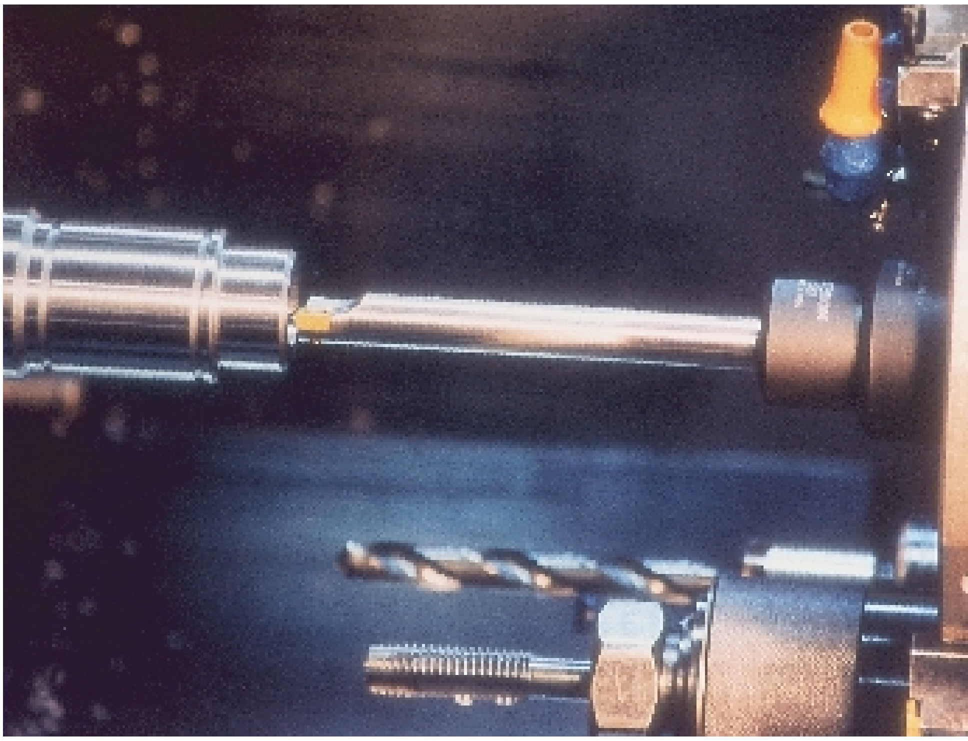
The cure for these problems is a "flat-top" insert. These inserts are ground on all sides and have very sharp cutting edges. This style of insert cuts freer, imparts a smoother finish, eliminates chatter and produces a rounder, truer bored hole.

Considering Clearance

Besides chip control, other insert considerations are clearance angle, support under the insert and top rake angle. The clearance angle becomes increasingly critical as the diameter of the hole decreases. When boring holes under 0.500" in diameter, side clearance of 11° is paramount.

But when boring holes under 0.250" in diameter, the tool reaches a "critical mass." Inserts need more than 11° of clearance, clamp screws become nearly microscopic, insert support becomes a wish and positive top rake turns into a dream.

All images: Criterion Machine Works



A heavy-metal boring bar absorbs vibration and reduces chatter, which is critical when the tool-length-to-diameter ratio exceeds 4.5:1.

A boring-system designer wants end users to bore with inserts and clamp screws that are readily available, lest his product be viewed as insupportable. As most carbide-insert producers tend to concentrate on cutters for milling and OD turning, there is a dearth of high-positive-clearance inserts on the market, especially small inscribed-circle (IC), non-chipbreaker, flat-top inserts.

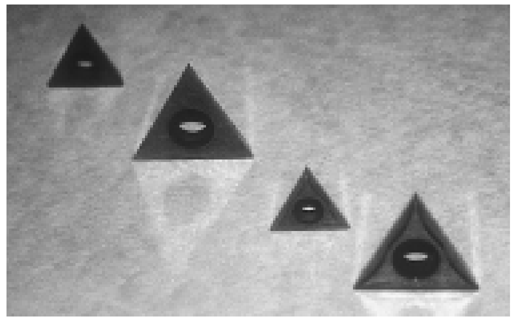
To use common ANSI- or ISO-standard inserts and clamp screws in a boring bar involves a trade off. For example, the positive top rake angle is sacrificed for bore clearance, because the insert is rotated into a negative rake angle. While rotating an insert into a negative inclination achieves more bore clearance, gives more support to the insert and more material for the clamp screw, it results in a high cutting force, inducing a negative top rake angle. This is just what a small, relatively weak boring bar does not need.

Countering this problem requires an insert with a high-positive-rake gash across its top face. But creating this feature on an insert with an IC of 0.156" or smaller is extremely difficult.

Small-diameter holes need the insert's cutting edge to be from "on center" to as little as 0.005" above the center. Setting a boring tool below center causes the neck of a steel tool to flex "down" as it enters the cut (a solid-carbide neck remains rigid or breaks). As the steel bar flexes downward, the cut becomes deeper and the downward force increases, which causes the tool to flex even more. Conversely, when an above-center boring tool flexes downward, the cutting edge "comes out" of the cut slightly, thereby reducing the downward pressure until the cutting force is balanced against the flex.

These problems have driven some boring-system manufacturers to design their own inserts and screws. Manufacturers of boring tools try to design inserts as "mainstream" as possible, but certain changes are needed to produce cutting tools that work. The manufacturing tolerances on such inserts are tighter than the industry standard, and I have found that for screw-down inserts, IC size and concentricity to the screw countersink is extremely critical.

Carbide grade is another important factor when selecting an insert for boring. The basic grades are C-1 through C-8, with the most popular being the C-2 and C-6 grades. C-2 is harder and more brittle, whereas C-6 is softer and less brittle, since it exhibits a higher cross-rupture strength. Traditionally, C-2 has been a general-purpose grade for machining cast iron and nonferrous materials, while tools made of C-6 have been utilized for machining steels or when there is an interrupted cut.



Micrograin-carbide inserts resist breaking and chipping in interrupted cuts.

Micrograin carbide is also available. It has a higher cross-rupture strength than traditional C-6 carbide. Micrograin-carbide inserts can handle interrupted cuts that would chip or break other grades of carbide.

Coatings that increase the surface hardness and/or lubricity are another consideration. While there is a plethora of coatings available, the most popular and recognizable coating is titanium nitride—the gold-colored coating. Insert manufacturers can, and do, tailor coatings to suit a specific operation and/or workpiece material. See your carbide supplier's catalog for specific examples.

Controlling Chatter

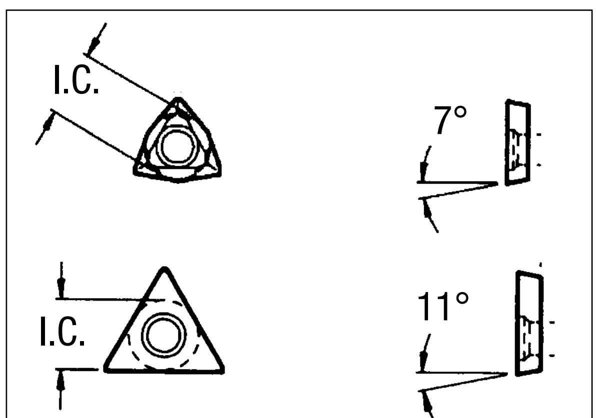
A critical aspect of boring that end users also need to pay attention to is overhang. Overhang is the portion of the boring bar that's not supported by the adjustable boring head. The boring bar should not exceed a tool-length-to-diameter ratio of 4.5:1 unless the bar is made of carbide, heavy metal (a high-tungsten alloy that's machinable) or is mechanically dampened (a dampened boring bar has a mechanism installed inside the bar that increases dynamic stiffness). Even with an overhang ratio of less than 4.5:1, chatter can be a problem.

The primary cause of chatter is an insert radius that's too large. At any overhang ratio greater than about 4:1, an insert radius of 1/64" or smaller is required. Applying an insert with a larger radius *will* lead to chatter. Here again, the chip-control insert rears its ugly head, increasing the likelihood of chatter. As the insert radius is increased, the cutting pressure is increased. This causes the tool to flex, which results in chatter.

If the need is for a fairly deep bore and boring is performed with an insert radius of more than 1/64", then a boring bar made of a dense material or a mechanically dampened bar is required. But even these bars only go so far in solving this problem.

Feeds and speeds also have to be balanced to achieve good bore finish and size. Often, though, speed-and-feed calculators prove inadequate. Some combinations of boring bar, insert and overhang ratio require "creative" speeds and feeds. Trial and error is the name of the game here.

Furthermore, always choose a bar with the largest minimum bore diameter (the size of the bore produced when the bar is spinning around its own centerline) that will fit in the hole to be bored. The bar's minimum bore diameter can actually be larger than the pre-



When boring holes under 0.500" in diameter, a side clearance of 11° is paramount. An insert with a side clearance of 7° is typically applied when facing or turning.

bored hole. For example, if you have a 0.720"-dia. hole and need to bore it out to 0.760", a 0.750"-dia. minimum bore bar would be the most desirable.

At the other end of the spectrum is the shallow bore, which usually has less than a 2:1 bore-to-depth ratio. A stubby bar is perfect for shallow boring. It accepts the same insert and clamp screw for a given minimum bore diameter as its longer brother, but it will be very stiff. Nearly any insert radius will work, without causing chatter. Since this style of bar is so stiff, hogging can easily be performed.

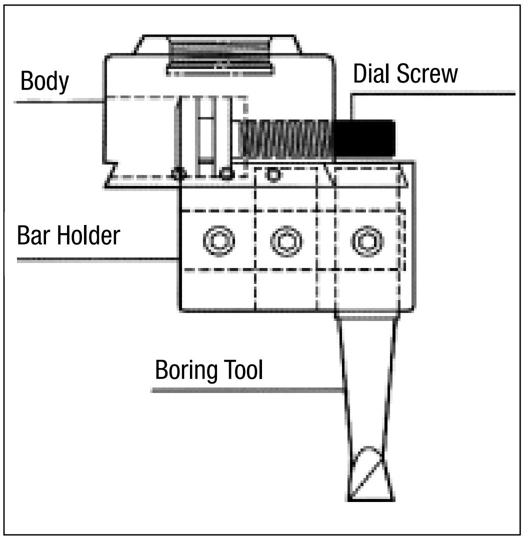
Head Adjustment

Proper adjustment of the boring head is also very important to maximize boring efficiency. The vast majority of boring heads have three basic parts: a bar holder, the body and the dial screw.

The boring bar should have a setscrew flat milled on its shank or a square shank with rounded corners to fit the ground bar-holder holes. When assembled correctly, the flat on the bar mates with the setscrew in the bar holder to set the cutting edge on centerline. If no flat exists on the bar, mill one on. If you don't, the setscrew will displace shank material, making the bar difficult to remove from its holder.

The bar holder fits into the body via a precision-machined and ground dovetail. The body has three setscrews that affect the fit and locking of the dovetail. The two outside setscrews are set at the factory and are called "gib-adjust" screws. They apply pressure to the dovetail to prevent it from moving out of position when the center "lock" screw is loosened to adjust the bore size.

However, don't panic if you have to adjust these factory-set outside screws. Just loosen all three screws, place a hex wrench in the dial screw and, while turning the dial screw, first snug one and then the other outside setscrew to achieve a little drag on the dial screw. There are no other rules; it's all a matter of "feel."



The basic boring head is made up of three components: the body, bar holder and graduated micrometer screw (dial screw). The body is usually threaded to permit its acceptance into most machine tools. The bar holder contains a dovetail mated to a dovetail slot running the full width of the body. And the dial screw moves the bar holder along with it as the dial screw is advanced or retracted in controlled adjustments of 0.0005", or less.

The third part is the dial screw. All boring heads should be direct-reading, which means that when you adjust the dial screw by one line, the cutting edge is advanced 0.0005" radially (0.001" on the diameter).

When boring with an adjustable (offsetting) boring head, the nature of the machine tool will cause an out-of-balance condition. As the adjustable head is offset, the bar-holder mass moves farther off center, causing an increasingly greater out-of-balance condition. The more offset, the more out of balance you will be.

An offset boring head and close-tolerance boring are the enemies of high-speed boring. As speed increases, the centrifugal force causes the boring bar to "whip," which can result in an out-of-round and/or tapered hole.

To reduce out-of-balance and large-offset problems, try to use a boring bar that is as close to the finish size as possible. This will prevent large offsets and allow higher speeds, since the head will be more balanced and the boring bar will be stronger.

The final consideration for effective boring is the taper shank that fits into the machine. I specifically use the term "taper shank" here because—of the three ways to mount a boring head on a machine tool—a taper shank is the most rigid. This practice is acceptable with all CNC-style shanks (CAT V, BT-style, etc.) and conventional-style taper shanks, such as Bridgeport R-8 or Morse Taper (i.e., MT No. 3).

The second best way is to use your largest endmill holder and a straight shank. The smallest straight shank you should ever use is one that is at least 25 percent of the diameter of the boring head. Of course, the larger the shank the better.

The least-effective method is to use a straight shank in a collet holder. The problem here is if you are using a single-angle collet and the shank is anything other than the size the collet was ground to, then the collet locates on a single ring of contact. This allows the shank to "chuckle" in the collet, resulting in chatter and an out-of-round hole.

If, however, you have a double-angle collet (clamps at the front and back) and the straight shank is ground to nominal size, satisfactory clamping can be achieved. Be aware, though, the more "chunks" you have between the spindle and the cutting edge, the greater the chance of failure.

About the Author

Randy Cloud is a manufacturing engineer at Criterion Machine Works, Costa Mesa, Calif. For additional information about Criterion Machine Works, call (949) 631-5444, visit the company's Web site at www.criterionmachineworks.com.