OCTOBER 2003 / VOLUME 55 / NUMBER 10

The TOOL CHANNEL

BY DAVID FABRY, HORN USA INC.

Fundamentals of grooving operations.

As with any turning application, the keys to good performance when OD grooving are choosing the best tool for the material and applying it at the proper cutting parameters.

Standard turning and grooving tools are made from similar grades of carbide and can be run at similar surface speeds. However, special consideration must be given to chip control when grooving.

It is significantly easier to form a manageable chip with a turning insert, or even a grooving insert when side turning, than to plunge with a grooving tool. This is because the chip is counter-rotating in relation to the workpiece and does not experience the same twisting force as a chip does during axial, or Z-axis, turning.

All images: Horn USA

Grooving encompasses a variety of related machining operations. Many of the same concepts that apply to simple OD grooving can be applied to parting off, side turning and face grooving. Available tools, like those shown, perform one or more operations.

Ideally, a grooving tool produces a chip resembling a watch spring. This indicates the chip is curling back onto itself and will eventually break—either at the end of the grooving cycle or because of the friction between the chip and the toolholder or groove sidewalls—as the chip becomes larger in diameter.

Three major factors impact chip control while grooving: insert geometry, surface speed and radial feed.

Insert Geometry

By applying an insert with a simple radial-ground top rake, the desired watch-spring chips are formed (Figure 1). This type of geometry does not thin the chip, so a finishing pass is usually taken on both sidewalls to produce the

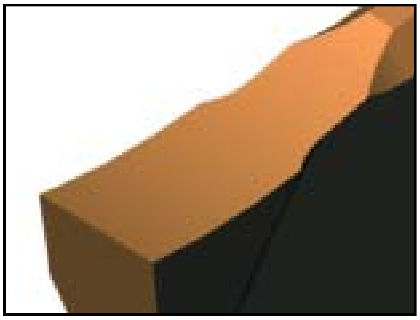

Figure 1: A simple radial-ground top rake geometry.

Figure 2: A radial top rake geometry that provides chip thinning.

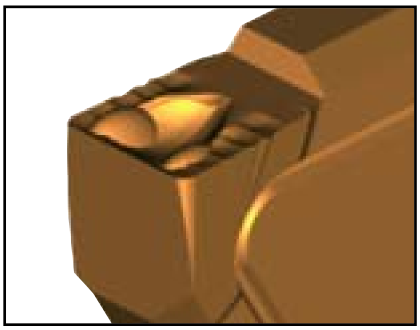

Figure 3: This insert geometry incorporates raised "bumps" that force the chip back on itself.

required surface finish.

For long-chipping materials such as leaded alloys, though, this chipformer does not provide enough resistance to curl the chip. It tends to generate a straight, flat chip that wraps around the tool and workpiece. However, the insert's sharp and positive cutting geometry is ideal for machining aluminum and other soft nonferrous materials.

The insert shown in Figure 2 also incorporates a radial top rake but is designed to thin the chip. This eliminates the need to take finishing passes on the sidewalls. In addition, this type of geometry, being on-center on all three sides of the cutting edge, allows for axial turning in those applications that call for a wide, shallow groove.

Finally, for materials for which chip control is difficult, the geometry shown in Figure 3 provides an aggressive barrier to the curling chip. The raised "bumps" force the chip back onto itself. This insert produces a tightly curled watch-spring chip or breaks the chip.

Surface Speed

To take full advantage of the chip-forming capabilities of modern geometries, the chip must be allowed to flow into the chipformer. This is accomplished either by decreasing the surface speed or increasing the feed. By decreasing the surface speed, the material moves slower across the cutting edge and, therefore, has more time to engage the chipformer.

Normally, a slower speed also increases tool life because the machining operation generates lower temperatures. There is a downside, though. Not only does cycle time suffer, but by having the cutting edge in contact with the chip for a longer time, more heat enters the tool than the chip.

Another drawback to lowering the speed is that the benefits of high-performance coatings might be negated. They often work best under elevated temperatures.

Increase the Feed

Increasing the feed is the preferred method for engaging the chipformer. A heavier feed produces a chip with a thicker cross section. And a thicker chip engages the geometry with more force, making it more likely to break. When keeping the surface speed constant, a faster feed also reduces cycle time.

Like reducing the speed, the benefit of increasing the feed is self-limiting. If the feed is increased beyond a certain point, the surface finish will be unacceptable. Also, a heavier feed requires a solid and stable setup, a strong insert geometry and a rigid toolholder.

For example, the geometry shown in Figure 1 is designed for feeds up to 0.006 ipr. Feeding heavier with this type of geometry will likely break down the cutting edge and will not provide any better chip control.

On the other hand, the geometry shown in Figure 3 can handle feeds up to 0.018 ipr when cutting materials with a medium tensile strength. Normally, feeds this high would not be necessary except for soft, gummy materials. With these materials, increase the feed from

Grooving backgrounder

A groove is a straight-walled recess in a component. Typical applications include cutting grooves for retaining O-rings and for providing thread relief.

Early grooving tools for lathes consisted of a piece of flat HSS with a flat top rake that was held in a block on the machine. The tool was fed into the workpiece, which generated enormous tool pressure and difficult-to-manage swarf.

When producing a groove in a component, the tool shears away the material in a radial fashion (X-axis movement). The formed chip is the same width as the tool and, therefore, the same width as the groove being created. This creates a high level of pressure at the cutting edge, from the friction of the chip on the sidewalls, which tends to result in a poor surface finish on the sidewalls.

The next generation of grooving tools incorporated a chipforming feature. This reduced the width of the chip and also provided a positive cutting condition, decreasing the amount of pressure at the cutting edge. The result was longer tool life, better sidewall surface finish and a more controlled chip.

Today's grooving tool manufacturers offer high-performance geometries that provide better chip control. And, as with turning tools, grooving inserts are being made from better carbide grades and are available with superior coatings.

—D. Fabry

the nominal starting value as needed to control the chip.

Parting Off, Side Turning, Face Grooving

The same concepts that apply to OD grooving can be applied to parting off, side turning and face grooving—with a few exceptions. Let's examine what makes each operation distinctive.

Parting off. Although considered a separate type of machining operation, parting off is nothing more than cutting a groove all the way through the raw material. Parting off is done either through the center of the stock or to a preformed through-hole (as in tubing) to separate the finished component from the remaining raw material.

Two time periods during the parting-off cycle merit special attention. The first is the moment when the component separates from the raw material. If the component is held in a subspindle, parting off is almost identical to OD grooving. If the component is allowed to drop freely, it is wise to reduce the spindle speed. A lower rpm will prevent "bouncing" in the machine and damage to the component. It also lets the operator control the point at which the component separates from the raw stock, so the parting-off tool does not get pinched between the component and stock.

The second time period to consider is when the surface speed begins to decrease after the machine reaches its maximum spindle speed. For example, on a machine running a solid bar that has a maximum spindle speed of 3,000 rpm, 300 sfm would be achievable only until the bar diameter reaches approximately 0.382". When parting to a diameter smaller than 0.382", the surface speed would decrease at the fixed rpm. At around 0.229", the surface speed would be at 180 sfm, or 60 percent of the ideal.

At this point, it would be appropriate to lower the feed rate by 40 to 50 percent, until the tool approached the point of separation (near the component's centerline). Shortly before the moment of separation, the feed should be reduced again due to the surface speed dropping to near zero at the center of the component.

Side turning. Side turning, or groove turning, should only be considered when the depth of the groove is shallow, relative to the width. A good rule of thumb is to side-turn when the width of the groove is greater than twice the depth.

Side turning starts with a radial, or X-axis, plunge into the workpiece, followed by an axial turning move. One of the main benefits of side turning is that the turning portion of the operation significantly increases the chip's ability to break. (With respect to the plunging portion, chip control considerations are the same as those discussed earlier.)

There are two schools of thought on toolholders for side turning. The first advocates the use of a less rigid toolholder. It allows the insert to deflect, providing clearance between the front cutting edge and the diameter that is being turned. This requires a programmed offset to compensate for the "digging-in" of the insert's leading corner. The width of the insert, the workpiece material and the feed determines how the offset is programmed. And, an additional "resetting" move must be programmed before the subsequent axial plunge to square the insert to the workpiece and prevent the cutting forces from damaging the insert.

The second school of thought is to use as rigid a toolholder as possible to maintain the insert square to the workpiece. This allows side turning with true programmed values and toolpaths. The tool plunges radially to depth, turns axially to length and plunges radially to the next pass depth. No offset values or resetting moves are required. With appropriate feeds, the full insert width contacts the component. This eliminates chatter and allows the tool to act similar to a wiper insert, resulting in a superior surface finish.

Face grooving. Face grooving differs from OD grooving in a number of key ways. Because the groove is machined into the face of the component, the material is moving in a different path relative to the cutting edge. The chip no longer flows in a straight line across the cutting edge, but, rather, moves at an angle because of the component rotation (Figure 4).

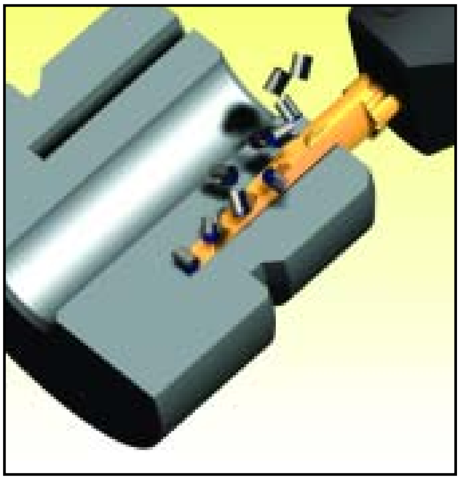

Figure 4: With face grooving, chip control is difficult because the chip is not flowing directly into the insert geometry, but, instead, is moving across it.

This makes chip control even more difficult than when OD grooving, because the chip is moving across the insert geometry instead of flowing directly into it. For this reason, an aggressive geometry, similar to the one in Figure 3, is a good choice for controlling the chip.

Care should be taken not to feed too heavily when face grooving because the chip may be forced directly back into the face of the component, creating back pressure on the cutting edge and eventually causing edge chipping. It is also important to have a geometry that thins the chip so that it is able to exit the groove without adhering to the sidewalls.

In face grooving, as with the other grooving operations discussed, the keys to success are applying the proper tool at the correct cutting parameters. For questions about specific applications, contact a grooving specialist.

About the Author

David Fabry is operations manager for Horn USA Inc., Franklin, Tenn. For more information about the company's carbide grooving tools, call (888) 818-HORN or visit www.hornusa.com.