MARCH 2001
VOLUME 53 / NUMBER 3

Understanding parting-off operations.

BY MIKE GADZINSKI

Parting Know-How

All images supplied by Iscar Metals

PART 2 OF 2

Part 1 of this article appeared in the February issue. The author discussed workpiece material and shape, impact of the machine tool used on a parting-off operation and proper setup of the cutting tool in relationship to the workpiece. In the second installment, he examines carbide insert styles and chipformers, carbide grades and coatings, insert clamping and cutting parameters.—Ed.

The purpose of a molded-style chipformer is twofold. First, it curls and breaks the cut material into manageable chips. Second, and equally important, the chipformer reduces the width of the chip to make it narrower than the groove being cut.

A chip that is not narrowed will score the side walls of the part, leading to an unacceptable surface finish. Furthermore, a flat chip whose width isn't reduced tends to break off in the groove and damage the cutting edge as it continues cutting.

Cut material that's not broken into manageable chips can lead to the creation of long, stringy chips. They can wrap around parts or interfere with subsequent operations. This often leads to downtime while the operator halts production to clear the machine. Figure 1 depicts three types of chips.

The types of carbide insert chipformers used most often for parting off can be divided into two main categories: negative cutting edge chipformers and positive cutting edge chipformers (Figures 2, 2a).

The negative-style tool is applied most often. The insert's small, negative

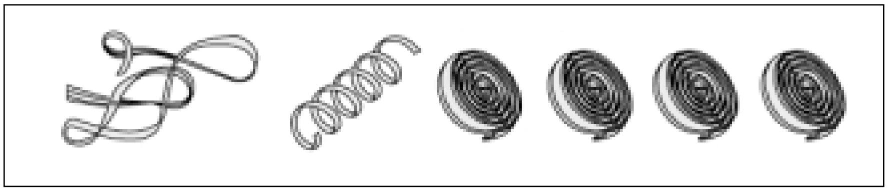


Figure 1: Among the different types of chips are (left to right) long, unformed chips; partially formed "pigtail" chips; and correctly formed "clock-spring" chips.

land at the frontal cutting edge increases edge strength, which permits heavier feed rates and offers protection during interrupted cuts and adverse cutting conditions. The width of this negative land—often called a T-land—is relative to the width of the parting tool. Generally, as the insert becomes wider, the land becomes wider and stronger.

The width of the land has a direct relationship to the feed rate and chip formation. The feed rate must be sufficient to force the material over the land and into the chipformer. A general rule of thumb when setting the feed for negative-rake parting tools is to multiply the width of the insert (in inches) by 0.04. So, for a 0.125"-wide tool, you would multiply 0.125 by 0.04 and arrive at a feed of 0.005 ipr. That would be a good starting point for parting off with a 0.125"-wide insert. However, it is always a good idea to consult the manufacturer's recommendations for individual grades of carbide and materials to be machined.

Despite the more prevalent use of negative parting tools, the positive-geometry style offers a number of advantages. Among them is its ability to narrow the chip at light feeds and generate minimal tool pressure. Excessive pressure can cause workhardening.

Examples of when a positive cutting edge should be used include applications on machine tools with low, fixed feed rates or when the workpiece material requires a low cutting speed. Positive inserts work well on soft materials, such as aluminum, copper and plastic.

Also benefiting from the free-cutting action of a positive tool are operations in which small stub burrs or ring burrs form on the workpiece ID. A negative-rake tool would require a higher feed rate and increase the size of the burr. Small-diameter parts and those with thin side walls are good choices for a positive-rake tool as well. The minimal tool pressure it develops lessens problems associated with obtaining a straight part-off on the sides of these workpieces.

Depending on the workpiece material and its stability, most positive part-off inserts can perform at feed rates as low as 0.001 ipr. Even at this low a feed, they provide excellent chip control and dependable tool life.

The disadvantage of a positive tool is that its cutting edge is much weaker than a negative one. It normally will not withstand the forces generated during interrupted cuts, heavy feed rates or unstable cutting conditions.

Clamp Down

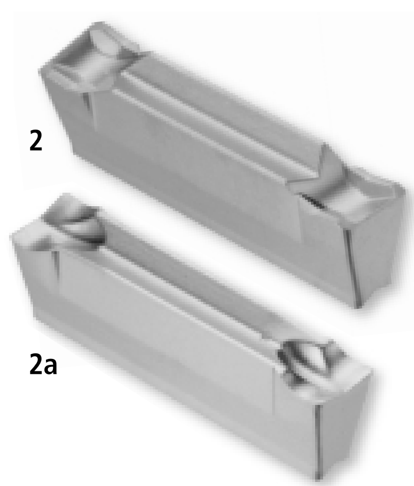
Another consideration when choosing a tool for parting off—or deep grooving—is the insert-clamping method. The majority of inserts produced over the last decade were designed to work with the Self Grip concept, pioneered by Iscar in the early 1970s. This clamping method incorporates no external screws or levers to hold the insert in place. Instead, it relies on the rotation of the part and tool pressure to keep the insert seated in a wedge-style pocket (Figure 3). The inserts designed for this type of holder are usually single-ended, and their geometry permits unlimited depth of cut.

With double-ended inserts—known as "dogbones"—the DOC is limited by the second cutting edge. They traditionally can only cut as deep as the overall length of the insert. Once that depth is reached, the trailing edge will begin to rub inside the groove that the tool is creating. In addition, dogbone inserts usually are secured by a screw-top clamp, which also limits DOC (Figure 4).

Historically, these two factors have tended to offset the benefits of having an extra cutting edge on an insert. But recent advances in carbide pressing technology have solved the problem. Double-ended inserts are being produced that have a "twisted" geometry (Figure 5). The frontal cutting edge is rotated from a parallel plane. When placed in the toolholder, the cutting edge will be square to the workpiece while the trailing edge is twisted several degrees to one side. Therefore, the insert occupies less space, in terms of its width, and will not drag along the side walls (Figure 5a). This design allows the use of a longer insert, one with greater stability in the cut due to its larger clamping area.

The problems associated with mechanical screw clamps holding double-ended inserts have been solved, too. New compression-loaded holders are available that secure the insert without any mechanical devices. The clamping system utilizes a holder with a pocket machined slightly smaller than the overall height of the insert. A cam lever is inserted into the toolholder to slightly lift the top jaw of the pocket. This allows insertion of the carbide tool into the pocket.

When the cam is released, the top jaw acts as a spring and compresses the top of the insert, holding it securely in place. With no screw clamp to interfere with penetration and the twisted geometry eliminating insert drag, there are no limits on DOC with this toolholder/



Figures 2: A small, negative land at the frontal cutting edge allows a negative part-off insert to be operated at heavy feed rates and under adverse cutting conditions. 2a: The positive insert is for low, fixed feed rates or when the workpiece material requires a low cutting speed.

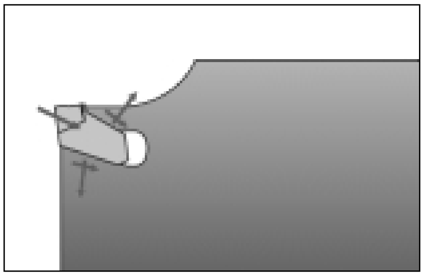


Figure 3: The Self Grip clamping method incorporates no external screws or levers to hold the insert in place. Instead, it relies on the rotation of the part and tool pressure to keep the insert seated in a wedge-style pocket.

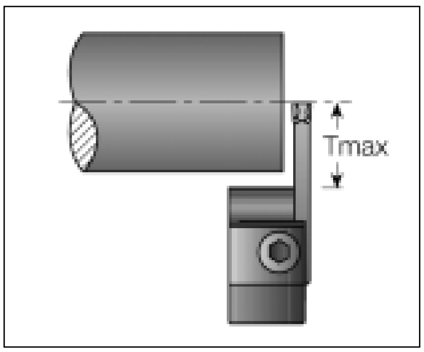


Figure 4: A parting tool with a screw-top clamp is limited in terms of how deep it can cut.

insert combination. The only limiting factors are the practical ones relating to tool width vs. DOC, a subject that will be discussed later in the article.

Carbide Grades, Coatings

Collectively, the material to be machined, workpiece shape, machining conditions and the machine tool drive the selection of a carbide parting-off tool. Usually, a specific grade of carbide and style of insert can be used for a wide range of applications. However, when an application arises that's outside that range, it's best to consult the tool manufacturer or supplier.

The material to be processed is the first consideration when choosing a carbide grade. Certain types of materials, such as high-temperature alloys, cast irons and some stainless steels, will require a specific grade. The best place to start is with the information contained in the tool manufacturer's catalog. Generally, when a supplier's grades work well for other roughing applications they will perform well when parting.

One thing common to all parting operations is the high amount of heat generated. It is very difficult to dissipate cutting-zone heat when making a deep part-off cut or groove. Carbide grades that perform well at elevated temperatures are beneficial. However, avoid selecting a grade based solely on its ability to withstand high temperatures. Most carbides that can withstand high heat are extremely hard and subject to chipping.

A part-off operation places big demands on the insert, particularly when parting to center. A grade that blends hardness and toughness is required.

The workpiece shape and other machining parameters also play important roles in the tool-selection process. For example, workpieces that require interrupted cuts or have uneven IDs will cause great amounts of stress to be placed on the tool. In these cases, carbides with the highest transverse rupture strength and the highest fracture toughness (K1C value) will work best.

Coatings for part-off inserts vary from supplier to supplier. But titanium carbonitride applied by the PVD process practically has become the industry standard for lower cutting speeds and tougher applications.

And when conditions are stable and higher cutting speeds are required, the choice narrows to titanium-aluminum-nitride coatings applied by the PVD process or the newer medium-temperature-CVD process. The reason is because TiAlN acts as a good heat barrier for the carbide substrate and can handle elevated temperatures.

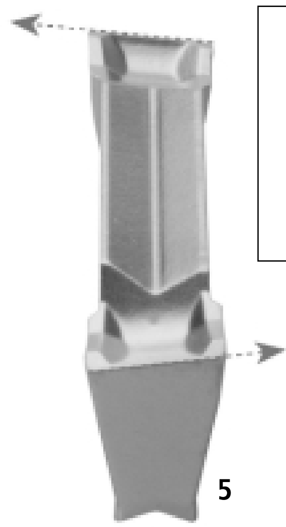
Cutting Parameters

An important rule when parting off is to maintain a constant feed rate and cutting speed throughout most of the cutting cycle. (A CNC machine helps ensure this.) The reason is that a constant feed and speed leads to a constant-temperature cutting zone and consistent chip thickness. Accordingly, tool life and chip control also will be consistent.

Unique problems can arise while parting off. One of them can occur as the tool approaches center on bar stock or small-diameter-hole stock. No matter how fast the spindle turns the stock, eventually a maximum rotational speed is reached. When this maximum is reached, and if the tool is programmed to run at a constant surface footage (sfm), then from that point of the cut forward to center the stock will be rotating at a constant rpm. At a constant rpm, there is a direct, corresponding reduction in cutting speed, until it reaches zero sfm. The grade of carbide and the insert's cutting edge geometry must be able to withstand this change.

Another problem that can arise in this situation is that the portion of stock being parted off can violently separate from the larger piece. It can be thrown at a high velocity, creating a dangerous situation for operators, even with the best of guards in place.

Still another aspect of this type of cycle is the condition of the workpiece. It will always break off prior to the tool completely parting it off. The result will be a large-diameter stub on the piece, or a heavy ring burr on the inside of the hollow diameter. Often, applying an insert that has a frontal lead angle will



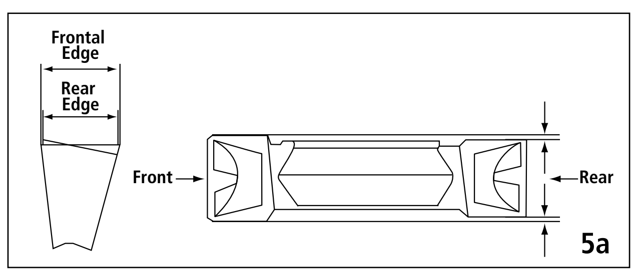


Figure 5: The frontal edge of an insert with a "twisted" geometry is rotated from a parallel plane. Figure 5a: When placed in the toolholder, the cutting edge will be square to the workpiece while the trailing edge is twisted several degrees to one side. Therefore, the insert occupies less space, in terms of its width, and will not drag along the side walls.

help minimize this burr, but the separation will still be violent.

The proper action to minimize the burr and problems with the part dropping free is to slow the feed rate and the rpm as the parting-off point approaches. Another rule of thumb is to reduce the feed rate and rpm by as much as 75 percent.

The slowdown should begin when the tool is at a distance from the separation point that approximately equals the width of the insert. This will help facilitate handling of the part dropping free and reduce the size of the burr left behind.

Obviously, other steps need to be taken with machine tools that don't offer variable control over the cutting speed and feed during the cutting cycle. The first is to choose a cutting speed (in sfm) based on half of the travel to the part-off point. This will give you an average cutting speed going in that is closest to the manufacturer's recommendation.

So, if you plan to part off a solid 4.00"-dia. bar, calculate your cutting speed (in rpm) as if it were a 2.00"-dia. bar. The cutting tool will start off at a faster cutting speed than recommended and slow in actual speed as it approaches center. The effect will be that the *average cutting speed* during the cycle will be what the tool supplier recommends.

A final consideration is the width of the parting tool vs. the DOC. This ratio will vary according to the condition of the machine tool, shape of the part and workpiece material. For example, you would have a much greater depth capability with a given tool if you were parting aluminum rather than a high-temperature alloy.

For most normal carbon and alloy steels, a good starting point for the maximum cut would be a depth-to-width ratio of 8:1. This would hold true for cutting tools approximately 0.125" wide and wider. Therefore, a 0.125"-wide tool would be acceptable for DOCs up to 1.00" (8 × 0.125 = 1.00). This tool would allow you to part to center on a solid material up to 2.00" in diameter.

About the Author

Mike Gadzinski is the manager of technical seminars and training for Iscar Metals Inc., Arlington, Texas.