MAY 2005 / VOLUME 57 / NUMBER 5

➤ BY MARKUS HEUWINKEL, WALTER USA INC., & ALAN RICHTER, EDITOR

Let's Talk RADIAL

Radial chip thinning boosts productivity by permitting a cutting tool to run at a higher feed per tooth when roughing.

All images: Walter USA

When rough milling, the goal is to remove as much material from the workpiece as possible in the shortest amount of time. The metal-removal rate is primarily based on a machine tool's available horsepower, of course, but even on a low-horsepower machine, productivity can be maximized and desired cutting conditions maintained via radial chip thinning.

Radial chip thinning is the effect of taking a radial WOC (a_e) less than 25 percent of the milling tool's diameter. The chip thickness based on the calculated feed per tooth (f_z) will diminish as the radial width decreases, resulting in a lighter actual f_z. This causes the tool to rub the workpiece rather than cut it, so the f_z needs to be increased as the radial depth decreases. The result is a decrease in cycle times and longer tool life.

It's in the Approach

With chip thinning, the most important aspect is the approach angle of the cutting tool. As the approach angle (χ) gets flatter, starting at 90° and progressing toward a lower value, the chip thickness, or h value, decreases.

Based on the different values for the optimal chip thickness when machining a specific group of workpiece materials, the feed per tooth that should be entered into the CNC program can be calculated using the following formula:

$f_z = h \div \sin \chi$.

The h value for a group of materials is presented as a range, with the lower

To attempt to cut chips thicker than recommended for a specific workpiece material, you risk overloading the insert and breaking the cutting edge.

number being the starting-point value. For cutting aluminum and nonferrous alloys on a machining center with up to 50 hp, the h value, or chip thickness, range is from 0.002" to 0.003". For stainless steels, titanium alloys and heat-resistant superalloys, the h value ranges from 0.003" to 0.006", and for steel, cast iron and nodular cast iron, the range is 0.006" to 0.010". To attempt to cut chips thicker than recommended, you risk overloading the insert and breaking the cutting edge.

Whether the approach angle is 90º, 60º, 45º, 30º or flatter, the chip thickness will always be constant (Figure 1). The exception is the round, or button cutter, insert, which, without any top geometry and a slight chamfer, provides the strongest edge.

Unlike inserts with a straight cutting edge, a round insert produces chips that increase in thickness as the DOC increases. Therefore, average chip thickness, the h_m value, describes the thickness of the cut for round inserts based on the insert's radial engagement of the workpiece through the cutter's diameter. The typical h_m value ranges for various workpiece materials are the same as the previously mentioned h value ranges.

Comparing a round insert (RDMT-2006) to a cutting tool with a 90º approach angle (ADMT1606), the volume of chips removed is the same if both are machining at the same DOC and the same feed per tooth. However, if the DOC is half of the round insert's inscribed circle, the round insert produces chips that are 30 percent thinner because the round shape has a longer cutting edge that engages radially with the workpiece (Figure 2). In other words, if the volume of each chip produced by each tool is the same and the length of each chip generated by the round insert is roughly 50 percent longer, the chip coming off the round insert is much thinner.

Now, if the feed remains the same and the DOC is reduced so it equals 25 percent of the button cutter's IC, the thickness of the chips produced by the button cutter is 50 percent less for the same chip volume. To achieve the productivity-increasing benefits of chip thinning, the DOC should be a maximum of 20 to 25 percent of the round insert's IC.

Upping the Feed

Because the chip thins as the DOC becomes shallower, the feed rate needs to be increased to compensate for the light DOC and to achieve a high level of productivity. Whether the result of applying round inserts or milling tools with a low approach angle, chip thinning permits the feed rate to be increased. This is because as the DOC for a round insert becomes more shallow, the approach angle flattens (Figure 3). Therefore, when variables for the average chip thickness and approach angle are entered into the formula for the feed per tooth that's entered into the CNC program ($f_z = h_m \div \sin\chi$), a significantly higher—by up to 100 percent—feed rate is the result.

With chip thinning, the most important aspect is the approach angle of the cutting tool.

To determine the effective approach angle for a button cutter, use the formula:

$\tan\chi = a_p \div (IC_{eff} \div 2)$ χ = effective approach angle.

Note that where the approach angle is 90°, the programmed rate of advance per tooth is the same as the chip thickness. If the approach angle decreases, the chip volume stays the same but the length of the cutting edge that engages the workpiece increases. Therefore, a smaller but longer chip than programmed is generated and a higher programmed feed rate is needed to raise the chip thickness to its intended value when the DOC is less than the round insert's radius.

Although the chip produced by the round insert has the same chip thickness as the one coming off the tool with a 90º approach angle, the button cutter is removing material at a much faster rate.

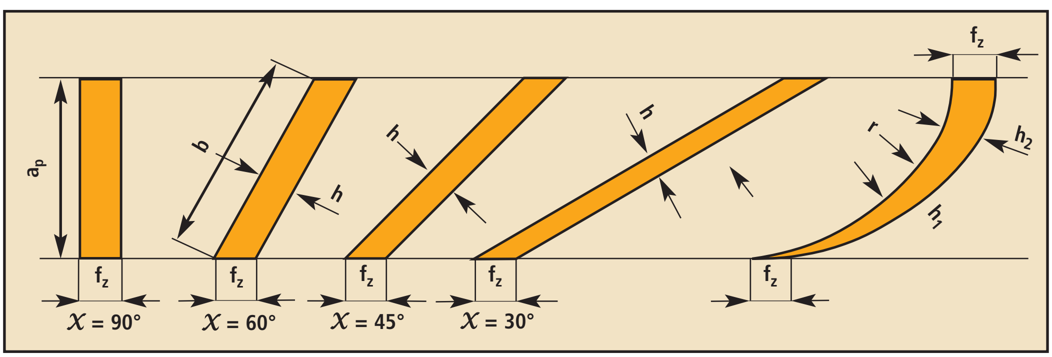

Figure 1: Shapes of the cutting sections for tools with various approach angles, from a 90º insert on the left to a round insert on the right. a_p = DOC; b = chip length; f_z = feed per tooth; h = chip thickness; r = radius; χ = approach angle.

Of course, a certain amount of machine horsepower is required to remove a certain amount of material, and if the feed rate is increased by 100 percent, 100 percent more machine horsepower is needed.

Rough milling with a 90º-approach-angle tool represents the worst-case scenario for advancing productivity, but it can be appropriate for machining a 90º shoulder instead of taking a second cut to remove a radius. Some shops also want to keep their tool inventory down and mill primarily with 90º cutters. Otherwise, such tools are not needed.

However, as a round insert gets smaller, its effectiveness, when applied with a constant DOC, decreases. This is based on the insert's larger approach angle that's entered into the formula for the programmed feed per tooth. In addition, as a round insert's DOC increases, its effectiveness also decreases. Therefore, a button cutter running at a DOC of 50 percent of its IC—the maximum DOC possible—would produce a chip with a thickness exactly the same as its programmed feed per tooth and be as productive as a cutting tool with a 45º approach angle. But if that button cutter has four cutting edges, it would be more cost-effective to machine with a 45º double-sided insert because it has eight cutting edges. (A button cutter has cutting edges on only one side, whereas a double-sided insert with a 45º approach angle has cutting edges on both sides.)

One of the limitations to achieving chip thinning with round inserts is DOC. The largest standard round insert is about 0.800", so the maximum DOC when high-feed milling is 0.200"—maybe 0.250". Although a 45º tool is able to run at a higher DOC, the button cutter can be programmed to rough faster if two passes are needed than if the other tool is taking one.

If the chip thinning technique produces chips that are too thin, rubbing occurs. For example, when applying a heavily chamfered insert with too low a feed per tooth, chips won't form properly, impeding chip flow.

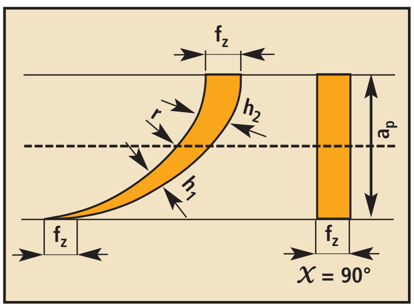

Figure 2: Chip thickness of a round insert (RDMT2006) vs. an insert with a straight cutting edge (ADMT1606). a_p = DOC; f_z = feed per tooth; h = chip thickness; r = radius; χ = approach angle.

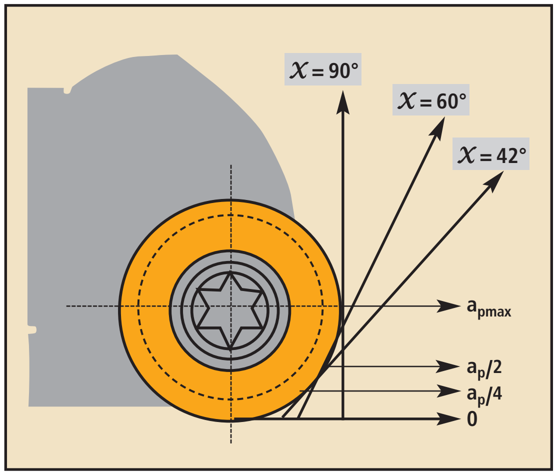

Figure 3: On a copy mill, as the DOC becomes shallower, the approach angle flattens. a_{pmax} = maximum DOC; a_p = DOC; χ = approach angle.

Redirecting Forces

As the approach angle flattens and the chip thins, cutting forces are redirected. For example, when machining with a cutting tool with a 45º approach angle, the axial cutting force is the same as the radial. The radial cutting force causes the tool to deflect and chatter, whereas the axial force goes in the direction of the spindle, making the process less vulnerable to destructive vibration. So a tool with a really flat approach angle primarily generates axial cutting forces.

The radial cutting force causes the tool to deflect and chatter, whereas the axial force goes in the direction of the spindle.

For milling when the goal is to remove the most material in the quickest amount of time, flatten the approach angle and keep the DOC light. The chips will thin and the cutter will fly. △

About the Author

Markus Heuwinkel is manager of product applications for Walter USA Inc., Waukesha, Wis. For more information about the company's cutting tools, call (800) 945-5554, visit www.walter-usa.com or ***enter 300 on the Information Services card.***