OCTOBER 2001 / VOLUME 53 / NUMBER 10

Iscar Metals

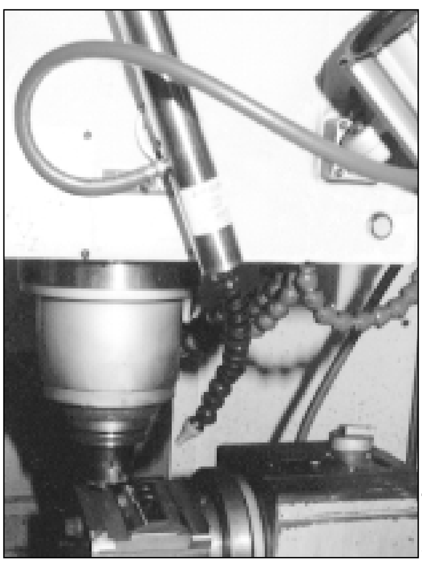
BY MICHAEL DEREN

Deep Impact

Tips for milling deep pockets.

If you haven't milled deep pockets already, sooner or later you will run into a job that requires you to. Companies that cut metal find this operation to be one of the most challenging. Some perform deep-pocket milling successfully while others struggle to do it economically and efficiently.

The consensus is that a pocket deeper than two times the cutter's diameter is "deep." And, according to milling experts, anyone who goes deep will encounter certain difficulties inherent to the operation.



ARTX Ltd.

Figure 1: Vortex-type air guns can clear chips from pockets of nearly any depth.

Brian Davis, milling-product specialist at Sandvik Coromant Co., Fair Lawn, N.J., said these difficulties include recutting chips, tool vibration, improper tool pressure, inadequate machine power and toolholders, and lack of workpiece rigidity. Let's look at each of these individually, as well as other factors that impact deep-pocket milling.

Recutting Chips

When milling a pocket—whether shallow or deep—insufficient chip evacuation creates problems. And the deeper the pocket, the more difficult it is to remove chips. High vertical walls hamper the flushing action, causing chips to settle in the pocket. This leads to their being recut.

One solution is to apply a high-pressure air system, such as a vortex-type of air gun (Figure 1). It clears chips from even the deepest of pockets if you aim one nozzle immediately in front of the cutter and one behind it.

A vortex unit typically costs $250 to $550. It features a magnetic base that makes it easy to position where desired. The unit is simple to connect to a shop's compressed-air system. Eighty to 90 psi is required, and pressure is regulated by adjusting a valve.

When cutting aluminum and other gummy metals, use coolant. Air won't work because it lacks coolant's lubricious characteristics, which are needed to evacuate chips from gummy materials.

Apply the highest coolant-delivery pressure possible to avoid recutting chips. For machine tools that only have flood-coolant capability, aftermarket attachments are available that deliver coolant at high pressures. Called coolant inducers, they can be purchased through local tooling sales reps for about $1,100 per tool. You'll also need a coolant pump. A 200- to 300-psi model should suffice.

If coolant can't drain from the pocket being milled, periodically stop the operation to clear the chips and pooled coolant. Also, apply a tough grade of carbide insert with a strong geometry. This type of insert features reinforced cutting edges and radiused corners. It will wear more slowly than an insert with a weak geometry.

Tool Vibration

Chatter is practically a given when machining deep pockets. There are ways, though, to reduce or eliminate this costly annoyance.

First, apply the largest diameter cutter available. It will be more rigid than a smaller diameter cutter, thereby minimizing tool deflection and the resultant chatter. Remember, the more rigid the tool, the less pronounced the chatter.

When using a close-pitch or extra-close-pitch cutter, the frequency of the inserts engaging the workpiece sometimes coincides with the natural frequency of the machine tool or workpiece. This natural phenomenon can lead to the development of chatter. By adjusting the spindle speed, you can sometimes neutralize these natural frequencies. Try ramping up the spindle speed first. Doing this promotes chip thinning, which usually alleviates the problem. If chatter still occurs, dial back the speed.

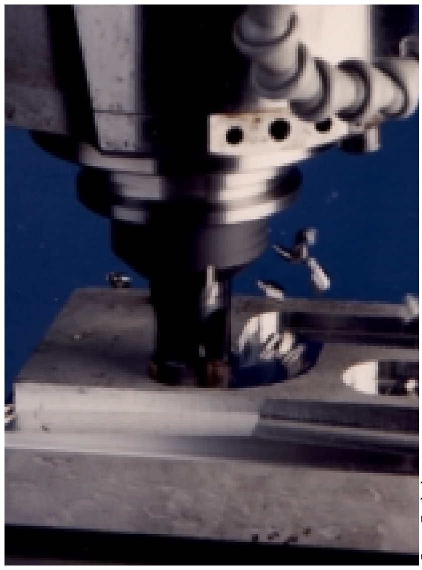
Other options are to use a cutter with differentially spaced inserts or a coarse-pitch cutter, which has fewer flutes than a fine-pitch cutter. Having fewer flutes in the cut at one time lowers the cutting forces, allowing you to more effectively address chatter.

Apply a standard-length cutter, not a long one, when you begin milling a deep pocket. A standard length is usually one to one-and-a-half times the cutter's diameter. Applying the shorter cutter helps reduce or eliminate chatter. It also allows you to achieve a higher metal-removal rate (mrr) in the shallower areas of the pocket.

After milling to the depth the first cutter can handle, switch to one whose length is three to four times its diameter. Decrease the depth of cut when you apply the second tool. The mrr will tail off, but you can use another technique to raise it again.

"By taking extremely light depths of cut, with high spindle speeds and very high feed rates—even in hard materials—you can remove more material in less time," explained Michael Madden, technical specialist at Seco-Carboloy, Warren, Mich. This technique, a cornerstone of high-speed machining (HSM), is widely used in the mold and die industry. It dramatically reduces the radial loading on the tool, which, in turn, reduces vibration.

The advent of high-speed spindles and the higher feed rates attainable on the newer machining centers has brought the technique to the forefront of metalcutting technology. And, it can be performed on older machines—


Seco-Carboloy

A 50-taper machine facilitates the cutting of full-width slots.

those with 7,500- to 10,000-rpm spindles and 200- to 300-ipm feed capabilities. The results won't be as dramatic as if performed on a 20,000- or 50,000-rpm machine with a 1,000- to 2,000-ipm feed rate, but they will be similar.

Tool Pressure

Proper tool pressure is important when milling deep pockets. Today's tools are designed to ensure the correct pressure.

The national training manager at Arlington, Texas-based Iscar Metals Inc., Mike Gadzinski, explained that as pressure is applied to the outside cutting edge of an indexable, positive-rake tool, the insert tends to tilt back and forth in the cutter pocket. "When you're pushing down on that edge, the opposite edge (the noncutting edge of the insert) wants to lift up," he said.

With the newer designs, though, inserts are wedged into a dovetail in the cutter body. They can't move, regardless of the pressure on the outside cutting edge. This type of cutter design permits higher speeds and feeds.

If less than one-half the cutter's diameter engages the workpiece, the chip formed will be thinner than calculated. The greatest pressure and heat generated is when the chip is very thin. In the shear zone, where the cutting edge is lifting and removing material, more heat develops and more burnishing occurs because the cutting edge is taking too small a bite from the part.

Below is formula for determining average chip thickness:

$$Ct = \frac{360 \times FPT \times WOC}{\pi \times D \times Ea} \times \sin k$$

where
Ct = Average chip thickness
FPT = Feed per tooth
WOC = Width of cut
D = Cutter diameter
Ea = Cutter engagement angle
k = Cutting edge angle

By changing some of the variables, such as feed per tooth or width of cut, you can alter chip thickness.

Gadzinski said that in terms of the mrr, it's most efficient to have approximately two-thirds of the cutter engage the workpiece. By taking a larger radial DOC and using inserts with free-cutting geometries, the tool will take a big enough bite and tool pressure will be minimal.

On old mills, operators tend to automatically slow the feed rate to reduce tool pressure. However, with the newer technologies—i.e., CNC machines equipped with indexable tooling—slowing the feed could result in *insufficient* tool pressure. This speeds insert wear, a result of excess rubbing, and causes chatter. The solution is to increase the feed rate to take advantage of the new inserts' geometries, which are designed to be run more aggressively.

Insufficient Power

Before beginning to mill a deep pocket, calculate the power requirement of various cutter sizes at different depths of cut. Knowing the power available at the machine spindle and the power requirement of the cutter will help you determine the optimal cutter size for your machine. The power-consumption formula is as follows:

$$Pc = \frac{DOC \times WOC \times F}{K}$$

where
Pc = Power consumption
DOC = Depth of cut
WOC = Width of cut
F = Feed in inches per minute
K = Material constant

Charts containing specific material constants can be found in technical manuals. Sample constants include 0.65 for 400 HB steel, 1.0 for 150 HB steel and 4.0 for aluminum.

For example, the power requirement to cut 150 HB steel at a 0.250" DOC, 2" WOC and 20-ipm feed would be figured as follows:

$$P_c = \frac{0.250 \times 2.0 \times 20}{1.0} = \frac{10.0}{1.0} = 10 \text{ hp}$$

Coarse-pitch cutters create less drag because they have fewer flutes. Consequently, they require less horsepower to make the cut. And, inserts with free-cutting geometries and coatings that promote the flow of chips allow the tool to bite into the material better.

Toolholding Considerations

With integral-shank tooling, the toolholder, shank and cutting tool are created out of one piece of metal. (If an indexable-insert tool is being used, the only components that aren't integral are the inserts.) Integral systems are the most rigid tooling available.

The next best thing is a dampened toolholder. It consists of a bar filled with a coarse powder or shot, or, sometimes, a liquid. This type of bar is especially useful when deep-pocket milling. Any vibration that develops is channeled into the toolholder, where the dampening material effectively dissipates and absorbs it.

Also, don't hold endmills with conventional endmill holders. They incorporate one or two setscrews that pin the tool to the opposite side of the toolholder's ID. These holders naturally hold the tool off-center by about 0.0002".

A tool off-center by this amount usually doesn't cause a problem when performing a conventional milling operation with a standard-length tool. But with an extended-length tool, runout will be compounded. Chatter and poor tool life will result, and, with a finishing tool, surface quality will be poor.

It's better to use a collet holder when endmilling deep pockets. It compresses radially and holds the cutter in the center of the collet. The cutter isn't offset, so it runs truer and vibration doesn't develop.

Rigidity is also critical when it comes to setting up the workpiece. Without a rigid setup, the best toolholder and the best tool with the best geometry will not perform optimally. Take the time to secure the part with an extra bolt or clamp. By ensuring that the workpiece remains motionless, you will prevent a lot of tools and inserts from breaking.

Machine Type

Another major factor when deep-pocket milling is the type of machine used. Is it a 40- or 50-taper machine? With a 40-taper machine, the mrr will be lower because you won't be able to take as aggressive a radial DOC.

The much stronger 50-taper machine, which provides greater torque and horsepower, will let you apply a larger, more rigid cutter and take advantage of more aggressive radial DOCs. These machines also accommodate fine-pitch cutters, which permit higher feed rates.

The techniques employed on 40-taper and 50-taper machines vary. With 40-taper styles, use tools with shorter gage lengths and apply them at lighter radial and axial DOCs. Use higher spindle speeds and make more passes. When making these partial-width cuts, increase the feed to achieve the desired mrr.

Avoid making full-width slotting cuts. If you have to make such a cut, reduce your feed rate approximately 50 percent.

When taking finishing cuts on a 40-taper machine, Iscar's Gadzinski noted that it's important to consider the radial chip thinning that takes place. "You can often make multiple passes with a solid-carbide tool and complete the operation faster than if you took one long (axial) pass."

For example, say you are using a 1"-dia. endmill. If you need to finish a 3"-deep pocket, run the cutter at a 1" DOC, followed by a 2" DOC, followed by a 3" DOC. The tool will flex less, since you are not trying to machine to the total DOC in one pass, and you will be able to run at a much faster feed rate around the pocket.

A 50-taper spindle accommodates larger cutters. A 40-taper spindle can run endmills with diameters up to about

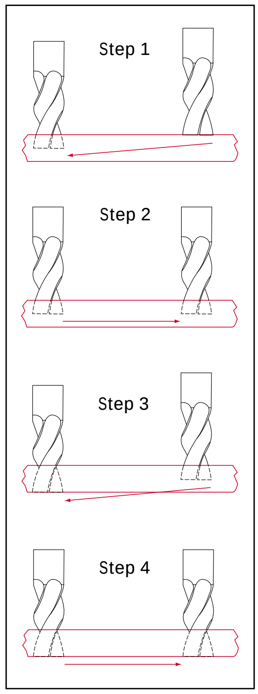


Figure 2: Ramping down involves machining into the workpiece at an angle. Start just above the workpiece and cut gradually into the material. Then slot straight back to the beginning of the cut. Repeat.

1½". A 50-taper machine accepts tools twice that size.

The larger spindle also permits the use of cutters with higher length-to-diameter ratios and the cutting of full-width slots.

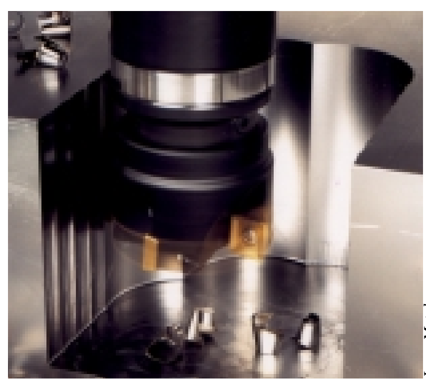
Cutters, Cutter Entry

The type of cutter applied is important, too. Use an indexable cutter to rough out pockets. Not only will you remove more metal faster, but you will save money because inserts cost less than solid-carbide cutters. An indexable cutter also will be more forgiving on older, less rigid machines.

Use solid-carbide cutters for finishing applications. Solid-carbide cutters should only be used on modern, rigid machines with higher-rpm spindles. Solid cutters also require lighter chip loads than indexable cutters.

The three most popular methods for entering a pocket are ramping down (Figure 2), predrilling or counterboring the entry point, and plunge milling (Figure 3).

Ramping down is recommended on a 50-taper machine running a cutter with round inserts. This method requires high horsepower and a rigid spindle to be effective. Again, choose the largest



Iscar Metals

Figure 3: The Z-axis plunge allows deep pockets to be cut quickly.

The following companies contributed to this article:

Iscar Metals Inc.
(817) 258-3200
www.iscar.com

Sandvik Coromant Co.
(800) 726-3845
www.coromant.sandvik.com

Seco-Carboloy
(800) 832-8326
www.carboloy.com

cutter available.

Predrilling an entry hole is another effective method. Simply drill a pilot hole as large, or larger, than the diameter of the cutter. For a blind pocket, use a center-cutting endmill to drill to the depth of the pocket. The disadvantage of the predrill approach is that it involves an additional tool change and an additional operation. However, the extra few seconds this takes can save a lot of wear and tear on cutters.

Plunge milling is also an effective method for cutting a deep pocket. Carboloy's Madden said: "I like plunging because it is extremely fast. All the pressure is going straight through the center of your spindle. There is no side loading. Even with a 40-taper, you can remove a lot of material."

To plunge, rapid the tool to a clearance height above the workpiece, then feed straight down in the Z-axis to the depth of the pocket. Next, rapid up to the clearance height and move in the X- or Y-axis. Feed down in the Z-axis and rapid up to the clearance height again. Continue this until the pocket is cleared out.

The step-over when plunge milling is determined by the cutter type and the tool manufacturer's recommendation. Most producers of indexable tooling now offer plunge-cutting tools.

Trial and Error

Many factors are involved in milling deep pockets. Trying the different techniques and being aware of the potential problems described in this article will help you succeed at machining these difficult pockets. Be aware, too, that different materials also will affect how you machine a pocket and success often comes after some trial and error.

And don't forget the value of your local tool sales engineers. They relish the opportunity to make test cuts at your facility with their tools. They probably have seen many of the problems you may encounter and will be more than willing to offer solutions.

About the Author

Michael Deren is a CNC applications engineer living in Woolwich, Maine, and a regular contributor to CTE.