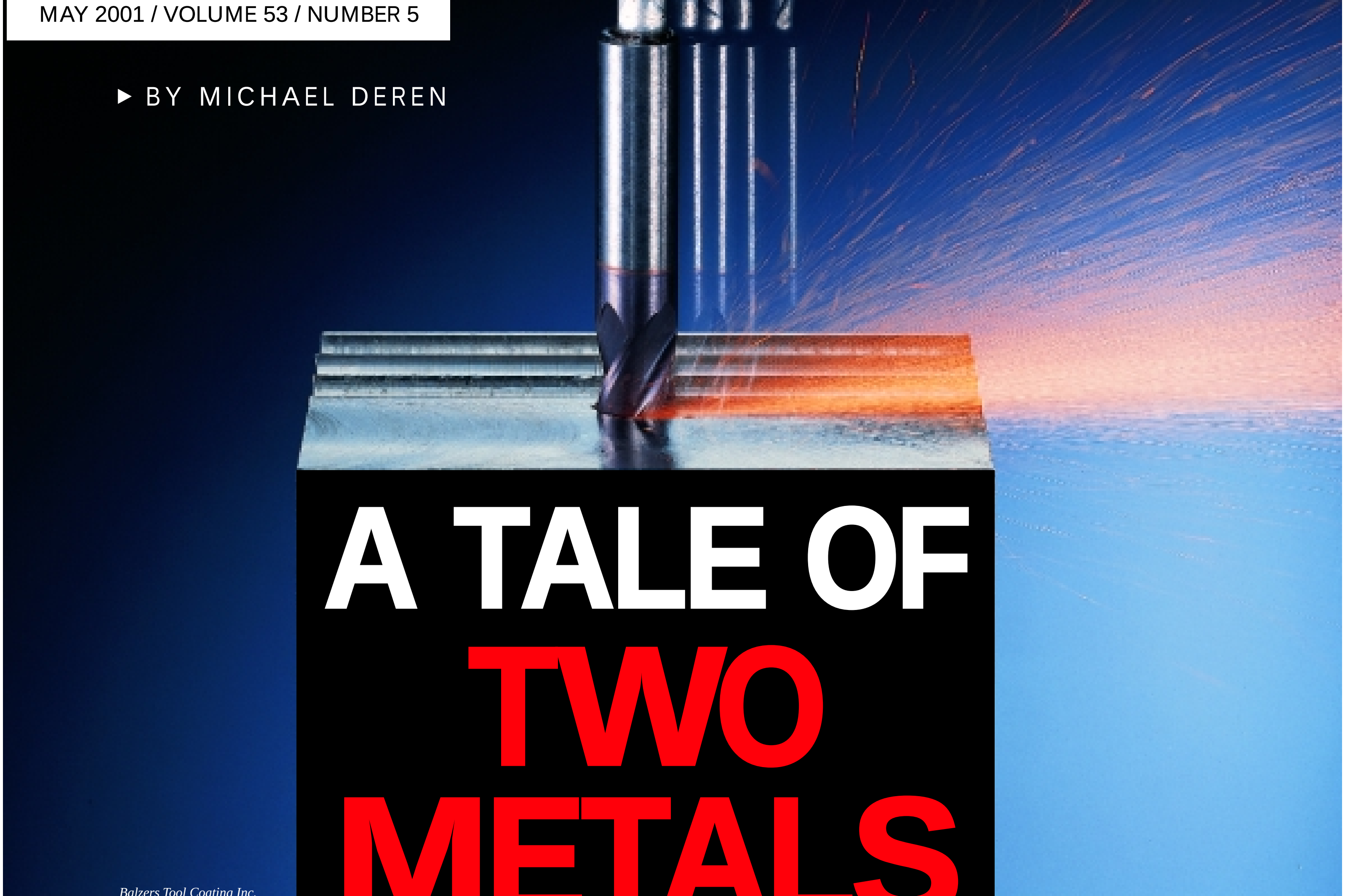
MAY 2001 / VOLUME 53 / NUMBER 5

▶ BY MICHAEL DEREN

A TALE OF TWO METALS



Balzers Tool Coating Inc.

Endmilling stainless steel and aluminum reveals that these two materials are surprisingly similar.

Perhaps no two metals are more dissimilar than stainless steel and aluminum. And yet as different as they are, endmilling them shows that they share significant similarities.

Similarities? No, I haven't been inhaling too much coolant mist.

For instance, both materials have a tendency to bond to the cutting edge. In addition, poor chip formation is a problem with both, and each can suffer from inconsistent surfaces. (Hard, abrasive carbides can be found in stainless steel, and large, silicon particles occur in aluminum).

Two- or 3-flute, high-helix endmills work well in these materials, and, sometimes, the same grades and coatings can be utilized.

There are, of course, differences between stainless and aluminum that become evident during endmilling operations. In the following pages we'll look at characteristics these materials share and ones that are unique to each. Knowledge of these similarities and dissimilarities can save your shop significant time and money.

Shared Characteristics

The critical factor to keep in mind when cutting either aluminum or stainless steel is that each requires a tool with a sharp cutting edge. Therefore, chip control is a primary concern. Some of the chips generated won't break easily, even at high speeds. Both materials are gummy and abrasive, so when cutting them, apply endmills that balance high edge sharpness with edge strength.

When it comes to selecting HSS and solid-carbide endmills, choose a 2- or 3-flute tool with a high helix. Both provide a high chip load per flute and, due to the higher spiral, augment chip evacuation between flutes.

Titanium nitride is an excellent coating for endmills used to machine stainless steel and aluminum. With its high lubricity, TiN facilitates chip flow, reduces friction, and resists galling and built-up edge. BUE is a leading cause of poor surface finishes.

Coated tools require less force to cut material and, therefore, are less likely to break. This permits increased feeds and speeds without excessive heat building up.

By applying a coated endmill instead of an uncoated one, you can bump up your feed and speed by at least 15 percent. In most cases, by the way, you are going to pay less than a 15 percent premium for a coated endmill.

The preferred indexable-insert endmill for stainless and aluminum is the 3-flute design. With it, one flute is in the cut while another one enters.

I have tried 2- and 4-flute indexables on 40-taper machines, but without much luck. Due to the even number of flutes, the tools tend to "hammer" the workpiece.

This occurs with a 2-flute tool because one flute doesn't enter the cut until another exits. With a 4-flute endmill, one flute is in the cut while another enters. But because the flutes are 90° opposed, the hammering effect still occurs. This is most pronounced on 40-taper machines.

It's important to select the proper nose radius when endmilling either material. Use the smallest nose radius you can. An excessively large radius will cause vibration, which translates into chatter.

On the other hand, an excessively small nose radius will break down too quickly.

I apply a 0.020" to 0.031" nose radius for roughing work, which minimizes chatter, and use a 0.008" to 0.016" nose radius for finishing. A finishing tool with that nose radius imparts a 125-µin. (or better) finish, yet can be fed aggressively.

Similar grades and coatings can be used when endmilling aluminum or stainless with an indexable tool. If you look in tool manufacturers' product catalogs, you will find many general-purpose grades. More than enough to be adequate for both materials.

I keep just a handful of material-specific cutters on hand—primarily for aluminum. They are designed for hogging, not slotting or peripheral milling. There is definitely a cost benefit to buying a large quantity of one type of insert rather than stocking smaller quantities of several different types from different vendors.

The last consideration is the direction of cut. Climb milling is the definite choice for machining these materials. In climb milling, the workpiece is fed in the same direction that the cutter rotates. The cutting forces are directed into the most solid components of the setup: the table and frame of the machine.

Endmilling Stainless

When machining stainless steel, select the largest-diameter endmill with the shortest length possible. A larger depth of cut translates into less machining time and takes full advantage of the latest cutter geometries. And don't forget to choke up the tool in the toolholder or collet as much as possible. This adds rigidity to the cutting tool, which minimizes tool deflection and allows you to take a much larger peripheral DOC when side cutting.

When slotting, it's common to make a plunge cut with an endmill (a drill also can be used), then change to a different endmill and slot at a constant depth. Changing endmills eliminates the shock of a single tool having to move down vertically then cut horizontally with a full load on it.

A more complex, but less time-consuming procedure, is ramping (Figure 1). Ramping is the process of machining into the workpiece at an angle. For this operation, start just above the workpiece and cut gradually into the material to, say, a 0.250" DOC. Then slot straight back to the beginning of the cut at a 0.250" DOC. Besides creating less stress on the cutter, this operation also shortens the tool-change time. Because of the decreased stress on the cutter, ramping is considered the better choice for small-diameter tools—those under 1" in diameter.

When making finishing passes, a good rule-of-thumb is the more flutes the better. If you have a 6-flute endmill, then by all means use it, but leave less material to be removed for the finish pass. The flutes of a 6-flute tool are shallow and could get clogged.

When using a 3- or 4-flute cutter, I normally leave 0.010" to 0.020" per side of material for a finishing pass.

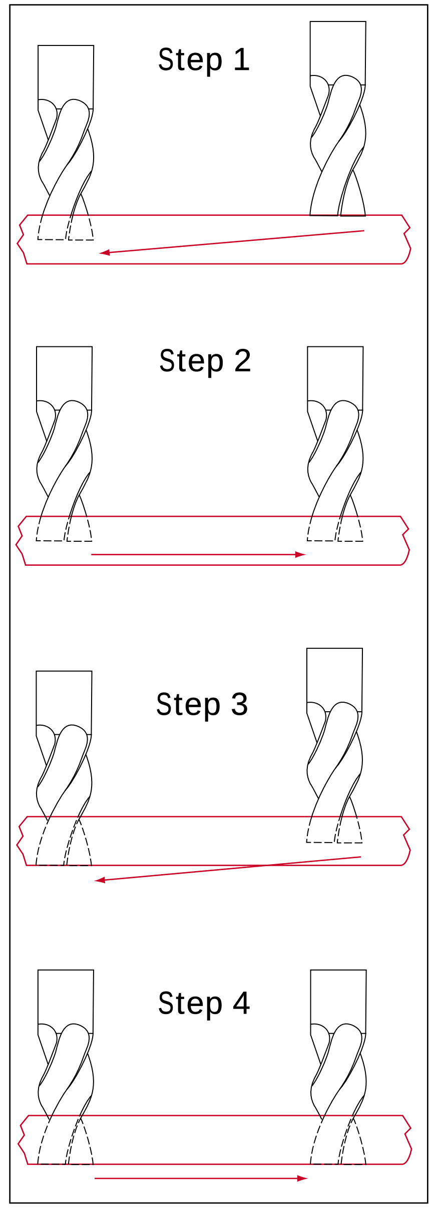


Figure 1: Ramping down involves machining into the workpiece at an angle. Start just above the workpiece and cut gradually into the material. Then slot straight back to the beginning of the cut. Repeat. Ramping puts less stress on the tool than plunging.

When using a tool with six (or more) flutes, you may need to leave only 0.005" or less.

Cobalt-HSS endmills are designed to maintain hardness, withstand high cutting temperatures and resist wear. They are a much better choice for endmilling stainless steel than non-cobalt-HSS tools. However, when possible, choose a coated, solid-carbide cutter.

Titanium aluminum nitride is a multilayer coating designed for a wide range of applications. TiAlN is an excellent choice for carbide tools used to cut stainless steels. It provides high thermal stability with excellent hardness, making it highly abrasion-resistant. An added benefit is its low coefficient of friction, which makes it lubricious.

Increasingly, more people are talking about, or considering, dry machining. Dry cutting is popular in Europe because the process is environmentally safer and minimizes disposal costs.

In my view, though, the jury is still out on dry vs. wet machining. I have made trial and in-process dry-cuts on stainless with indexable inserts, and several U.S. manufacturers have reported good initial results.

The problem I see is that even though little heat enters the part, a lot of it enters the tool. When cutting wet, the insert wears gradually and the operator can hear when it's time to change the tool. That isn't the case when cutting dry. When inserts in dry operations fail, they tend to experience catastrophic failure. This can easily ruin the part.

Endmilling Aluminum

The first thing to consider when endmilling aluminum is tool geometry. Whether made of HSS or solid carbide, the best choice for cutting aluminum is an endmill with a 37° (or higher) helix and a heavy-duty web. Two- and 3-flute endmills are usually applied, with the 3-flute style being the more common.

The 3-flute design provides better chip evacuation and higher feeds and speeds. This translates into more cubic inches of metal removed per unit of horsepower than is possible with a 2-flute endmill.

However, a 2-flute cutter is an excellent way to reduce chatter when machining into corners. This is an area where a 2-flute endmill has an advantage over the 3-flute style. When changing direction, only one flute is in the material, so the chip load remains constant whether you are moving straight ahead or making a 90° turn.

Make sure to apply an endmill made from high-quality HSS or micrograin carbide. And by applying endmills that have center-cutting flutes, you can plunge into the material rather than ramping into it.

Also, use CNC-ground tools. A CNC grinder makes a more consistent tool, resulting in more consistent tool performance. It yields a better chip ramp, too, which enhances chip evacuation and ensures accurate rake angles and a finer surface finish on the critical cutting edges. This reduces friction on the cutting surfaces, making for a cooler tool and workpiece.

Another major factor to consider is the endmill's coating. Two popular coatings for aluminum are TiN and titanium carbon nitride.

The lubricious nature of TiN improves chip flow, minimizes galling and BUE, and reduces friction. TiCN has the same lubricious characteristics as the TiN, but is 30 percent harder. It is ideal for maximizing tool life in long production runs.

With TiN-coated HSS endmills, you can expect three to five times the material-removal rate than would be possible with uncoated ones. And, with TiN-coated, solid-carbide endmills, you should see an increase of up to 15 percent in cutting speed.

Indexable tools designed specifically for aluminum outperform any other type of cutter used for this material. The tools typically have one to three flutes and diameters ranging from ¾" to 1¼". They feature very deep gullets that enhance chip evacuation. Also, the inserts are designed with an extremely high shear and are highly polished, to resist material adhesion and create a slippery flow surface.

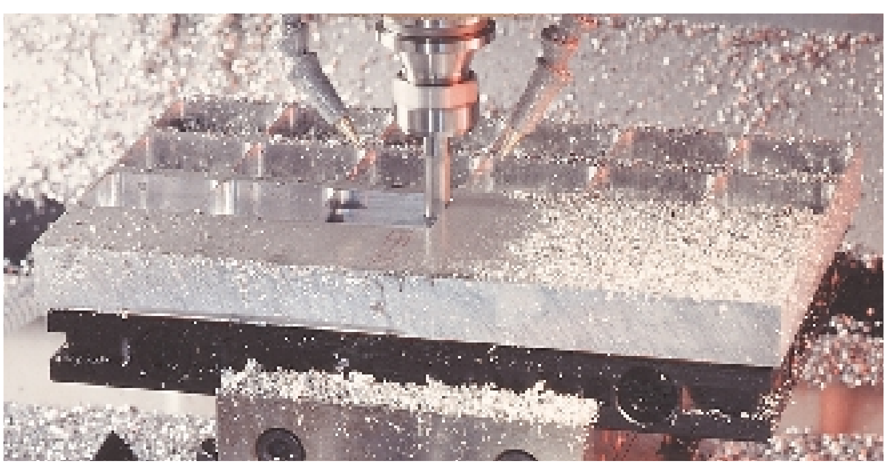

M.A. Ford Mfg. Co. Inc.

Endmills for aluminum should be made from high-quality HSS or carbide.

When machining aluminum, if it doesn't sound like someone is throwing large pieces of gravel at the walls of the machine, you are not running aggressively enough. Crank it up! This applies even to light-duty machines.

If you have a machine with a coolant-through-the-spindle capability, use it whenever possible. At my shop, we have actually doubled our DOC and increased our feed rates by 25 to 50 percent by utilizing through-coolant tools. Furthermore, we consistently mill aluminum at up to 150 ipm on 40-taper machines.

If a pressurized system is unavailable, direct flood coolant at the cutting edges. This helps minimize intermittent cooling, which can wreak havoc on your cutter. Remember, too, that cutters do not like thermal shock. Use as many nozzles as you can to direct coolant to


IBAG North America

Aluminum, like stainless steel, requires a tool with a sharp cutting edge.

the cutting edges. You will not regret it.

Recap

When endmilling aluminum or stainless steel, remember the following:

- apply a tool with the proper coating—TiN, TiCN or TiAlN;
- use a high-helix, 2- or 3-flute tool made of either HSS or solid-carbide;
- when using an indexable insert on stainless, select one with a large-positive-rake angle; for aluminum, select one with a high shear and enough clearance so that it shaves or shears the material away with little effort; and
- lastly, always select the shortest possible tool for the job to ensure maximum tool rigidity.

About the Author

Michael Deren, a CNC applications engineer who lives in Woolwich, Maine, is a regular contributor to CUTTING TOOL ENGINEERING.