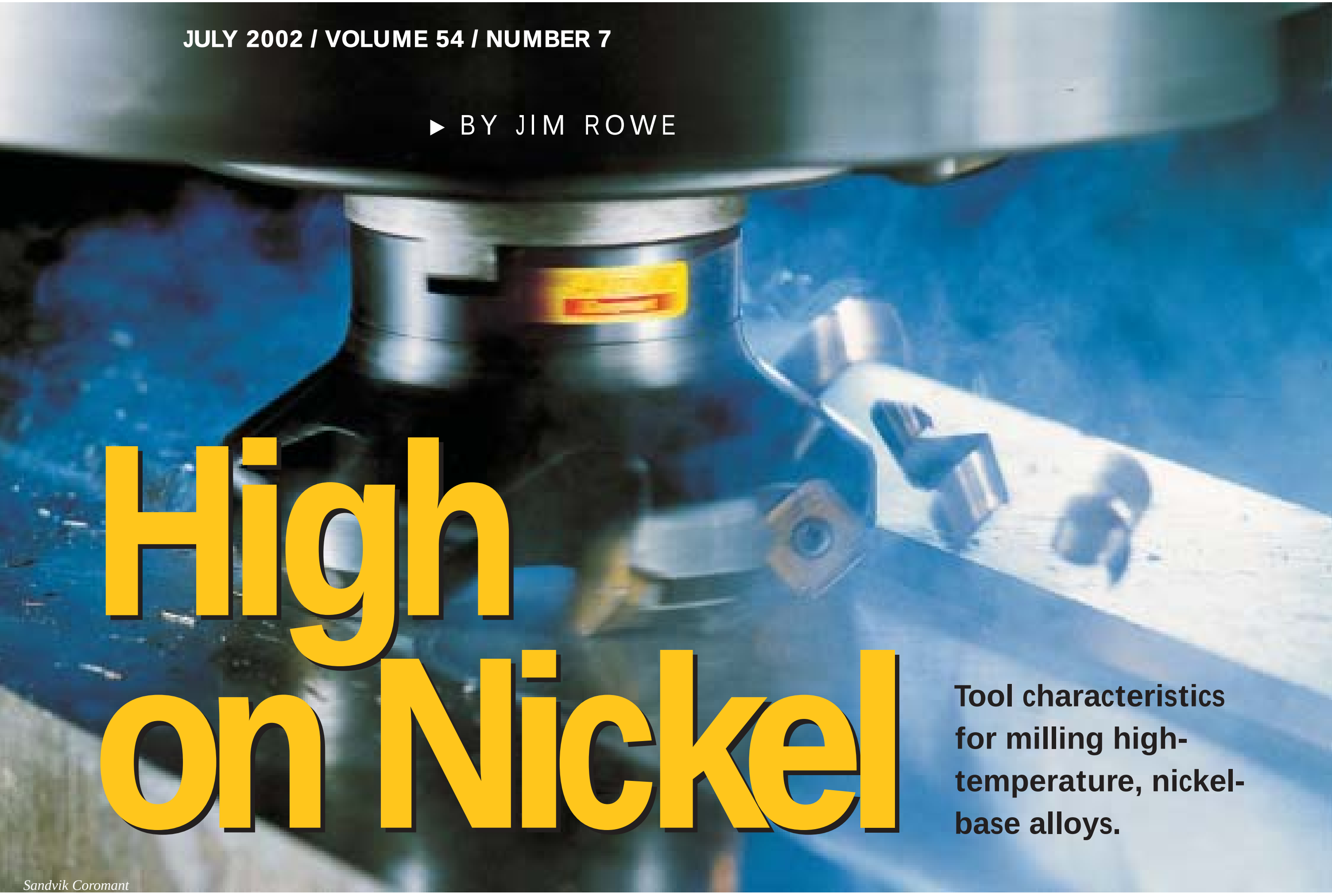
JULY 2002 / VOLUME 54 / NUMBER 7

▸ BY JIM ROWE



High on Nickel

Tool characteristics for milling high-temperature, nickel-base alloys.

Sandvik Coromant

The mere mention of alloys such as Hastelloy, Waspaloy, Inconel and Kovar instills horror in machinists, but it needn't. For when it comes to these and other nickel-base alloys, knowledge is power. Nickel alloys are being milled increasingly to make critical components for the aerospace, medical and chemical-processing industries. These materials offer great strength and corrosion-resistance and are able to withstand extremely high temperatures.

However, the same metallurgical properties that provide these benefits also make milling the alloys difficult.

When it comes to milling metal, familiarity with the material's specific chemical composition is beneficial. But when milling difficult-to-machine nickel-base alloys, that knowledge becomes critical. Knowing the percentages of toughness-enhancing nickel and hardness-adding chromium—the two main ingredients in this group of alloys—and the balance of other ingredients can help you predict tool wear.

Other elements found in the material's mixture might be silicon, manganese, molybdenum, tantalum and tungsten, among others. Notably, tantalum and tungsten are also two of the main ingredients used to make carbide cutting tools. Since these ingredients are effective for cutting metal, their presence in the workpiece guarantees a difficult milling operation. It's almost like applying one carbide tool to cut another.

Tooling or Process Failures?

It's important to understand why the tools that cut other materials fail quicker when milling nickel alloys—even though some consider these "failures" normal tool wear.

Yet, everyone agrees that tooling costs are higher when cutting these alloys. So what should you expect? Many shops consider a tool cost five to 10 times the amount spent on tools to mill "typical" steels as acceptable.

There will be no arguments if you say that heat is the most important thing you need to control when milling nickel alloys, because too much heat destroys even the best carbide cutter. Knowing the correct amount of heat generated for the type of tool (high-speed steel, carbide or ceramic) being applied is very important.

However, excessive heat is not the only problem encountered when milling nickel alloys. Many tool failures associated with these materials are actually not the tool's fault. Faulty fixturing and toolholders can also shorten tool life. The lack of rigidity when holding a workpiece allows it to move while being cut. This movement can cause "mechanical" fracturing of the carbide substrate. Sometimes this fracturing appears as small cracks running along the cutting edge. Other times there will be flakes or large chips removed from the carbide. These are signs of an unstable cutting condition.

Chipping can also occur when the carbide is too hard or the chip loads are excessive. You could apply an HSS tool to reduce the chipping, but HSS will not tolerate heat as well as the harder carbide tool, so there's a trade-off. Be sure you are looking at all of the causes of chipping and not just one.

In addition, taking some up-front time to strengthen the workholder will pay big dividends in the long run. With a more rigid workholder, not only is tool life extended, but finer surface finishes and tighter tolerances can be achieved.

Similarly, when selecting the toolholder, the wrong choice can shorten tool life. Let's say you were to place a ⅛"-dia. endmill in an endmill holder instead of a collet. The runout associated with the endmill holder, which is pushed off-center by the setscrew because of the clearance at the tool shank, will produce a greater chip load per tooth for one of the endmill's flutes. This leads to an uneven cutting action, which is undesirable when cutting any material but especially true when milling nickel alloys.

By using toolholders that provide improved concentricity, such as hydraulic or shrink-fit holders, a more even cutting action will reduce tool wear and allow more balanced cutting. This imparts better surface finishes as well. And, when selecting a holder, choose the shortest one available.

These characteristics for tool and workpiece holders will help when milling any material, but you need as many advantages as you can get when milling nickel alloys.

Know the Tool

The manufacturer of the cutting tool, regardless of its design or what it's made of, should provide some starting-point recommendations for the cutting speed and feed per tooth. If you do not have this information, call the toolmaker's technical assistance group.

The manufacturer should also know the cutter's capabilities when performing full-width slotting, peripheral cutting, plunging or ramping. The clearances on many standard cutters won't allow you to perform many of these processes. For instance, if a cutter does not have a generous amount of secondary clearance, the possible ramp angle will be reduced.

Obviously, if you try to go beyond the tool's capabilities, there will be failures. The same holds true when slotting. If the tool does not evacuate chips from the bottom of the slot quickly, the slot will pack with chips and the tool will fail. Any way you look at it, tool life won't be extended when milling high-temperature alloys.

Kennemetal Inc.

Compared to carbide, ceramic cutters can mill high-temperature alloys at significantly higher speeds.

But if you think slowing the feed will extend tool life, most times you are going the wrong way with your troubleshooting techniques. Typically, when making the initial cut, you'll find that these materials are fairly hard. If you tend to feed slowly (0.001" to 0.002" per tooth for indexable inserts), the tool's cutting edge rubs the workpiece. The result is rapid—if not immediate—failure.

Rubbing can cause the workpiece to workharden. To avoid this, one should make sure the tool enters the initial cutting path with a generous chipload (0.006" to 0.008" per tooth).

The depth of cut is determined by many factors, such as tool design, insert height, fixture rigidity, overall tool length and available horsepower, but the geometry of the cutting edge plays a huge part. A positive cutting clearance, or rake angle, of 5° to 11° works best when cutting these somewhat gummy materials.

Helix angles play a big role as well. Endmills should have a helix angle between 35° and 50°. The sharper "slicing" of the material these tools provide creates chips that, hopefully, carry the heat away from the tool. Review your tool arsenal and try a few with varying helix angles to see how they perform.

Selecting the proper cutting speed is, of course, also very important when milling nickel-base alloys. This cutting parameter controls the initial amount of heat generated in the cutting zone. Recommended speed ranges from as low as 40 to 50 sfm for HSS tools to 75 to 120 sfm for carbide cutters to 600 to 800 sfm or higher for ceramics. Increasing the feed and DOC can also generate additional heat because of the corresponding increase of force and contact area between the tool and workpiece.

To handle the stress and cutting forces found when milling nickel alloys, select a tool with a tough submicron-carbide substrate. Combined with heat-resistant TiAlN coatings, this carbide grade can be very successful. However, TiCN-coated tools work well at lower speeds.

Even with the best substrates and coatings, if you do not apply the tool carefully, it will fail. Let's say a part needs a slot with a depth of 0.150" and you plan on cutting it in three passes. Usually, a CAM system will indicate that consistent DOCs need to be taken during this process. The problem here is the consistent depth will eventually create a notch in the coating, because the workpiece repeatedly contacts the same area of the tool. Once the notch breaks through the coating, it will then damage the substrate, causing tool failure. Therefore, vary the DOC slighty (0.020" to 0.030") over these areas to preserve tool life.

Hot or Cold

Due to the extreme amount of heat generated while milling these alloys, the common reaction is to flood the cutting zone with coolant. While this is practical for small-diameter tools, larger-diameter tools, such as facemills, cannot be entirely flooded while in the cut. In this situation, turn off the coolant and mill dry.

When the cutter cannot be covered with coolant, the inserts move in and out of the heat, eventually leading to thermal fracturing of the carbide. You will notice small cracks at first, running perpendicular to the cutting edge, before total loss of the insert occurs.

In some instances, even smaller tools can mill effectively without coolant. If nothing else works, run dry and see if tool life improves.

Since medical and aerospace components are often made of nickel alloys, certification papers usually accompany the material. The "certs," as they are known, detail the chemical structure of that particular material. This is where you can find out what you will actually be milling. Note the material's composition and how it cuts under the parameters you selected.

As previously noted, the two main elements of this group of metals are nickel and chromium. When the metal producer adjusts the percentage of each, the metal's resistance to certain corrosive elements and its strength and hardness characteristics change, as does its machinability.

Cutting tools can easily be designed to cut either tough or hard materials, but the problem to overcome when milling nickel-base alloys is getting a tool to successfully machine a workpiece that is both tough and hard.

You may have your own unprintable names for these alloys, but by understanding their composition and properly applying the appropriate tool, you can successfully mill metals such as Carp 20, Rene 41 and Haynes 242 without dread.

About the Author

Jim Rowe is a cutting tool application salesman for a Midwest distributor.