OCTOBER 2006 / VOLUME 58 / NUMBER 10
BY MIKE PRINCIPATO

The $5 million man

My hunch is that if you're reading this, your shop survived the recent lean times and is now having a pretty good year. Maybe even a damn good year, given that demand for manufactured hard goods continues to roll along. The corpses of former competitors may litter your regional market, but you stuck it out through the difficult years and are now prospering from your perseverance. Your shop's throughput and revenue are at record levels—maybe as much as 25 to 100 percent higher than 2 or 3 years ago.

Yep, you're a titan of industry and the envy of your neighbors. Hell, your customers are lucky to have you around. You're ready to go to that fabled "next level" you've been reading about in a zillion self-help business books.

I hate to rain on your parade, Sparky, but you probably aren't the guy to get your shop there—at least not by yourself.

When my shop started taking off in the late '90s, I was smart enough to realize I needed the help of an experienced manager, but not smart enough to get out of his way when he came onboard.

There aren't any statistics on such things, but the Unwritten Encyclopedia of Business Truisms states that it's the rare entrepreneur who makes a successful transition to professional manager. That's easy to understand because the skill set required to launch a shop—the tireless intensity, the ability to think dynamically, to continually douse the fires that flare in every startup, to actually thrive on those very flames—is what makes an entrepreneur.

Professional managers need distinctly different skills to grow a business, and those skills aren't usually born in the hellfire of a startup, but rather are gained through formal education and on-the-job experience.

Give an entrepreneur a stick of gum and a ball of twine and he'll build a product out of them, build a company out of the product and persuade financiers and customers to jump onboard before he's even built a bandwagon.

In contrast, professional managers systematize the procurement of gum and twine, optimize the throughput of the product, grow the company profitably and ensure that the financiers' and customers' ride on the bandwagon is smooth and stress-free.

I call such a professional manager the $5 Million Man. You may have founded your shop or grown it to its current revenue level. Given the size of the average independent U.S. job shop, that would mean annual sales from $1 million to $3 million. The $5 Million Man will get you to $5 million and beyond—a major revenue plateau in the metalcutting business. At $5 million and with typical industry profit levels, a shop owner has enough cash flow to reinvest at a torrid rate, pay down debt fast enough to make a banker worry or maybe even buy a Ferrari.

One question, though. If you need the $5 Million Man—and odds are that you do—are you going to hire him when you find him and, more importantly, are you going to share the power so he can do his job? From experience, I assure you that sharing power is going to be tough—and necessary.

How do I know? Because I'm an entrepreneur, which means my management competence ranges from merely capable to simply awful. When my shop started taking off in the late '90s, I was smart enough to realize I needed the help of an experienced manager, but not smart enough to get out of his way when he came onboard. The result was painful for both of us and for the employees who got caught in the daily crossfire. Tony and I had the classic friction that inevitably results when a sales-oriented, strong-willed owner accustomed to getting his way needs to coexist with his methodical, low-key second-in-command.

Here's the type of conversation we often had:

Tony: "Mike, we need to tell our customers our lead times are longer than we've advertised in the past. We're overbooked and falling behind on deliveries."

Me (melodramatically waving a packet of new inquiries in the air): "So, you want me to tell our biggest customers that they should find another shop? Are you kidding me?"

Tony: "No, that's not what I'm saying. I'm suggesting our estimates should give a more realistic date, given our current backlog."

Me: "Look, we got this far because of my business philosophy, which is as follows: (holding two fingers from each hand in the air to emphasize that the next sound bite is quote worthy and thus of enormous value to the fortunate listener) 'If we sell enough stuff at profitable levels, everything else will take care of itself.'"

Tony (silently): "I can't believe I left GE for this."

Ultimately, Tony left the shop out of frustration.

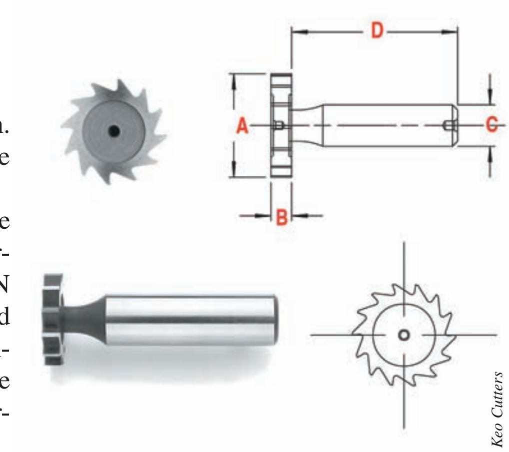
When you find your $5 Million Man, give him the resources, space, authority, goals and time he needs. Then get out of the way and enjoy the ride.

About the Author

Mike Principato is a metalworking industry consultant and former owner of a midsized CNC and EDM shop in Pennsylvania. He can be e-mailed at ctemag1@netzero.net.

'Chiseling' precision slots

BY LAROUX K. GILLESPIE



Shank-mounted keyseat cutters have four basic dimensions: (A) cutting diameter, (B) head width, (C) shank diameter and (D) shank length.

Making a keyway, or keyseat, used to a matter of brute strength. In a 1913 issue of *Machinery* magazine, an "old time fitter" lamented that "… good chipping (with hammer and chisel) one seldom sees nowadays. … To see a man chip a keyway in a pair of shaft couplings, having a 2½"-dia. bore and a length of 5", in an hour and a half to 2 hours is a sight good for the eyes."

Keyways aren't created with hammers and chisels anymore, so machinists need to know their keyseat cutter options. Keyseats are machined into shafts, pulleys, flywheels and gears to lock the shaft and outer hub together so torque can be transmitted.

Three basic keyseat geometries exist: the through-slot, the radiused runout slot and the blind Woodruff keyseat. Broaches produce the first, EDMs and electrochemical machines produce the latter two, and milling cutters and shapers produce all three. Additionally, keyseat cutters are available for use on milling machines and are typically the cheapest way to produce low-volume keyways.

Keyseat cutters will work on any milling machine, but special keyseat cutting machines are also available. In addition to cutting keyways, keyseat cutters produce splines, slots and serrations.

Most keyways measure roughly ¼" wide and use either straight keys or the ½"- to ¾"-dia. half-round Woodruff keys. Keyways on some couplings and clutches, however, are up to 2½" wide, and one of the largest applications cuts 5"-wide × 5'-long keyways in cast steel cylinders weighing 10,000 lbs.

Standard keyseat cutters come in shank- or arbor-mounted designs and have straight or staggered teeth. Cutters over 2" in diameter are mounted on arbors.

Shank-mounted cutters are made of HSS, cobalt-HSS or carbide and have a TiN, TiCN, TiAlN or AlTiN coating. Carbide-tipped versions are also available. Standard arbor-mounted cutters are currently limited to HSS and carbide with a TiN coating.

HSS tools are applied for cutting ferrous and nonferrous materials, and cobalt-HSS tools are used for difficult-to-machine materials. Some carbide-tipped cutters include chipbreakers, but the HSS ones do not.

All keyseat cutters have a similar shape. The ½" round shank typically has a full-length flat or a Weldon setscrew flat to allow setscrews to lock the tool in the toolholder. Niagara Cutter, Amherst, N.Y., produces a straight-tooth Woodruff keyseat cutter having a 5° positive radial rake angle and a 0° axial rake—straight gashed—angle. The toolmaker grinds the sides to have a slight concavity, so the teeth will not drag in the cut. Teeth are typically spaced from 0.13" to 0.29" apart on the circumference.

Unless otherwise requested, keyseat cutters are right-hand cut. Typically, the smallest standard tool is ½", but specials can be ordered to meet specific needs.

Toolmakers mark the diameter and width of shank-mounted Woodruff keyseat cutters with an American Standard number or code in which the last two digits define the cutter diameter in ⅛" increments. The preceding digit or digits defines the cutter width in 1/32" increments. Thus, a cutter coded as 812 would be an 8/32"-wide × 12/8"-dia. tool, or a ¼"-wide × 1½"-dia. tool.

Tolerances are defined in the ANSI B94.19-1997 standard. For the smaller sizes, the typical cutter width tolerance is +0.0000"/-0.0005", and diameter tolerances are from +0.020" to +0.015". The tools have sharp corners unless otherwise specified to produce a square fillet in the bottom of the keyseat.

The diameters for arbor-mounted cutters start at 2⅛" and go to 3½". The standard sizes use a ¾" or 1" arbor hole.

Special thanks to Keo Cutters Inc., Warren, Mich. For more information about the company's keyseat cutters and other cutting tools, call (888) 390-2050, visit www.keocutters.com or ***enter #305 on the Information Services card****.*

About the Author

LaRoux K. Gillespie is a retired manufacturing engineer and quality-assurance manager with a 40-year history of precision deburring. He is the author of 10 books on deburring and almost 200 technical reports and articles on precision machining. He can be e-mailed at laroux1@myvine.com.

Fan appreciation

BY BILL KENNEDY,
CONTRIBUTING EDITOR

Abingdon Tool Ltd. specializes (though not exclusively) in the manufacture of progressive dies, jigs and fixtures for the medical and lock making industries. However, "for fun and for something different," owner Phil Laiacona and the shop staff regularly fabricate classic car parts no longer available from the manufacturers.

One example was a cooling fan for an early-'60s Ferrari V-12 engine. The fan consisted of three tapered and formed steel blades riveted to the arms of a steel center hub. As a sample, Laiacona was given a clockwise-turning Ferrari fan that someone had tried to adapt to the counterclockwise-turning V-12 by heating and bending the fan blades. The unbalanced result was not acceptable.

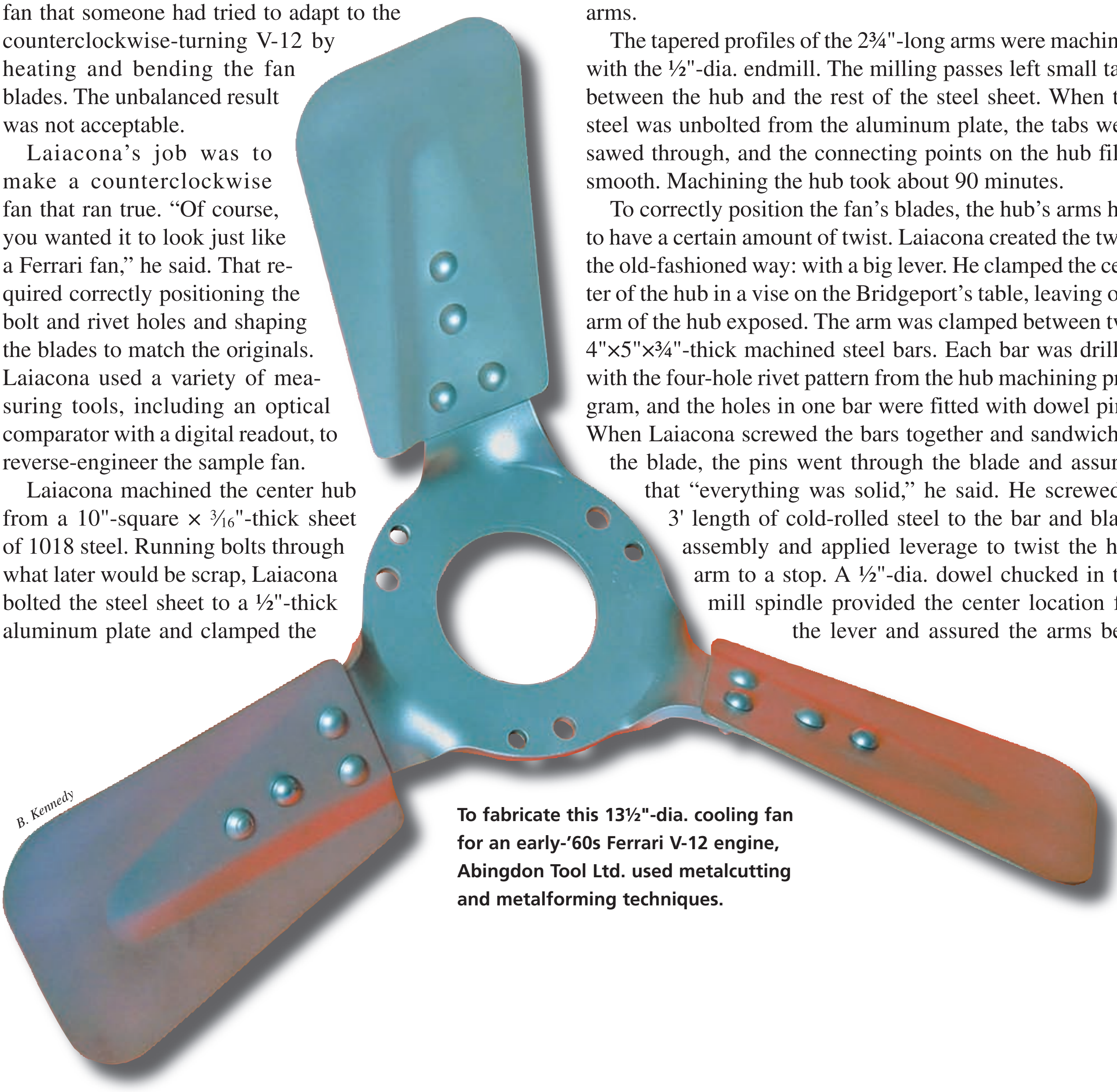
Laiacona's job was to make a counterclockwise fan that ran true. "Of course, you wanted it to look just like a Ferrari fan," he said. That required correctly positioning the bolt and rivet holes and shaping the blades to match the originals. Laiacona used a variety of measuring tools, including an optical comparator with a digital readout, to reverse-engineer the sample fan.

Laiacona machined the center hub from a 10"-square × 3⁄16"-thick sheet of 1018 steel. Running bolts through what later would be scrap, Laiacona bolted the steel sheet to a ½"-thick aluminum plate and clamped the plate to the table of a 3-axis Bridgeport EZ-Trak milling machine. The setup expedited the process because it permitted the cutting tools to machine through the steel into the sacrificial aluminum plate, eliminating the painstaking steps to provide tool clearances.

In the first operation, a ½"-dia., square-bottom HSS endmill run at 800 rpm and an 8-ipm feed circular-interpolated a 1.700"-dia. hole in the hub's center. A plug was inserted into the hole and screwed to the aluminum plate to boost the stability of the setup. Next, around the hub center, Laiacona drilled three ¼"-dia. dowel holes and also made three 8mm×1.25 drilled and tapped holes. Then, he drilled four 3⁄16"-dia. holes for rivets at the locations of the hub's three arms.

The tapered profiles of the 2¾"-long arms were machined with the ½"-dia. endmill. The milling passes left small tabs between the hub and the rest of the steel sheet. When the steel was unbolted from the aluminum plate, the tabs were sawed through, and the connecting points on the hub filed smooth. Machining the hub took about 90 minutes.

To correctly position the fan's blades, the hub's arms had to have a certain amount of twist. Laiacona created the twist the old-fashioned way: with a big lever. He clamped the center of the hub in a vise on the Bridgeport's table, leaving one arm of the hub exposed. The arm was clamped between two 4"×5"×¾"-thick machined steel bars. Each bar was drilled with the four-hole rivet pattern from the hub machining program, and the holes in one bar were fitted with dowel pins. When Laiacona screwed the bars together and sandwiched the blade, the pins went through the blade and assured that "everything was solid," he said. He screwed a 3' length of cold-rolled steel to the bar and blade assembly and applied leverage to twist the hub arm to a stop. A ½"-dia. dowel chucked in the mill spindle provided the center location for the lever and assured the arms bent



To fabricate this 13½"-dia. cooling fan for an early-'60s Ferrari V-12 engine, Abingdon Tool Ltd. used metalcutting and metalforming techniques.

uniformly. After one arm was twisted, the hub was indexed and the bending process repeated for the other two. “We inspected them, and all the angles were identical,” Laiacona said. The actual bending of the three hub arms took about 15 minutes, but making the tooling to do the job took about 4 hours.

Fabricating the fan blades required a mix of metalcutting and metalforming. Laiacona cut the 4½"-long blade shapes from 0.045"-thick, cold-rolled 1018 steel using a Sodick A325 EDM. He stacked the steel stock and “made a dozen at once because we knew we were going to fool around with forming it.” Cutting the blades took about 90 minutes. Back on the Bridgeport mill, he drilled the blades with the same pattern of four 3⁄16"-dia. holes previously made in the arms of the hub.

To match the original blades, the flat stock had to be “formed and ‘kicked’ at a little bit of an angle,” Laiaconca said. To do that, he made a form die from A-2 tool steel. “We got our dimensions and picked up our angles off the sample blades,” he added. “We milled the blade form into the die and made male and female parts guided on a couple of pins.”

The actual bending of the three hub arms took about 15 minutes, but making the tooling to do the job took about 4 hours.

He said the male half of the die was the shape of the blade, while the female was simply a blank hole. Forming a blade took only 15 seconds on the shop’s J&L press. The formed blades required no trimming because the die form was designed to compensate for the amount the blade would pull into the die.

The blades were riveted to the hub arms with 3⁄16" round-headed steel rivets of an appropriate style. The rivets were set with a hand-riveting machine that Laiacona’s father had used at Bridgeport’s Chance Vought aircraft factory in the mid-1940s. Laiacona described it as “a little hand press. It captures the head of the rivet, and you squeeze it, and it produces mechanical force with a couple of levers.”

Laiacona said he mounted the fan on a lathe in his shop and it ran nicely at 3,500 rpm. “I know that after they put it on the car, there were no problems with it,” he added. He made three: two for his customer, Automotive Restorations, Stratford, Conn., and the third was for working out the fabrication processes.

For more information about Abingdon Tool Ltd., Bridgeport, Conn., call (203) 335-8129.

Mastercam X puts the 'to' in 'art to part'

BY BILL FANE

CNC Software Inc. has upgraded its Mastercam program to reduce the "art to part" process to one step.

Tolland, Conn.-based CNC Software updated the program so its Mastercam X looks like Windows-based software and lets users accomplish certain tasks with fewer commands.

But, let's start with a bit of background information. The art-to-part buzzphrase refers to the process of taking a 3-D CAD model of a part and then generating the physical part from it without 2-D working drawings. With today's technology, this actually works extremely well. A couple of years ago, a student of mine took an Autodesk Inventor file and used a Haas VF1 CNC milling machine to produce a functioning propeller for a 9.9-hp Mercury outboard motor, even though he had virtually no machining experience.

The only thing confusing with the art-to-part phrase is that it implies a two-step process, where an intermediate step is required. After modeling the part, the model must be processed to produce the G-code program required to control the CNC machine before the part can be made.

This is where Mastercam X comes in. It is the latest release of a popular, capable postprocessor G-code generator. In Mastercam 9 and earlier versions, however, it's most significant shortcoming was probably that it had been around for a few years BW (Before Windows), so its interface didn't have the look and feel of current Windows applications.

While Mastercam X seems to contain few new features or capabilities, it is definitely worth buying new or upgrading from an earlier version. CNC Software spent the majority of its development effort making the CAM software faster and easier to learn and use. The resulting productivity gains should justify the cost.

When looking at the new user interface, it's apparent that Mastercam X is not simply a bunch of pretty icons that replace the old menu structure. There are also many internal changes linked to the new interface.

For example, if you wanted to import an Inventor solid model into Mastercam 9, it was necessary to go through five menu picks as you worked your way down through the sequence of main menu, to file, to convert, to Autodesk, to Inventor to get to where you could browse for the desired file. In Mastercam X, however, all you need to do is to select the file and then open it. You can directly open the latest release of any of 20 supported file formats, including the major CAD packages as well as the neutral formats, such as ACIS, STEP and VDA.

The improved interface means that one click on an icon in Mastercam X replaces five menu clicks in Mastercam 9.

The improved interface means that one click on an icon in Mastercam X replaces five menu clicks in Mastercam 9.

This same philosophy has been applied throughout the Mastercam X interface. Common functions are grouped together in appropriate toolbars, while the menu sequences and toolbar icons are much more consistent with standard Windows practice.

Also, the new Smart Ribbon Bar improves operator efficiency. It's a toolbar that only appears when appropriate, and then it only displays relevant items. For example, if you are doing 2-D drawing and start the arc command, the ribbon bar appears, displaying everything relevant to drawing an arc. This includes such items as the type of arc and appropriate windows for precision data entry.

Another major improvement in Mastercam X doesn't appear in the menus or toolbars. The code generator has been rewritten so it generates G-code up to eight times faster. On a sample part, Mastercam 9 took more than 10 minutes, whereas Mastercam X took about a minute and a half.

Additionally, Mastercam X includes a new machine definitions functionality. The software ships with a number of generic and common definitions. You can modify them to suit a particular machine or create new ones.

There are two major uses for the machine definitions. First, the definitions won't allow you to specify an operation that isn't possible on the selected machine. The software won't permit selection of a 3-axis machine if a 5-axis machine is needed.

The other use for machine definitions arises when a machinist needs to run the part on a different machine. The software will warn him when the existing program exceeds the number of toolholders, maximum spindle speed, table size and so on of the alternate machine. This makes it easy to move a program to a different machine without nasty surprises.

I do have one minor quibble with the license manager for Mastercam X. Standalone installations use the HASP protection system, which features a hardware "dongle" that plugs into a USB port. You can install a single Mastercam X license on both your office and home computers, as long as you remember to transfer the dongle back and forth. The problem is that the dongle is about the size of a thumbnail, and it does not have a hole in it to allow it to be attached to a key chain. The odds of losing it are quite high; mine went AWOL for 3 days, having gotten tangled up with some papers in my briefcase.

All in all, though, Mastercam X is a worthy upgrade to a trusted and capable program.

About the Author

Bill Fane is a former product engineering manager, a current instructor of mechanical design at the British Columbia Institute of Technology and an active member of the Vancouver AutoCAD Users Society. He can be e-mailed at Bill_Fane@bcit.ca.

BY DR. JEFFREY BADGER

Surface grinding for fast metal removal

Archives MNATP, Collection Meillassoux

Dear Doc,

I grind flat a lot of HSS stock on a standard surface grinder, using a 1"-wide WA60HV wheel. I dress 0.0004" at the beginning and, again, just before the last few passes, taking 10 seconds to dress the wheel, which runs at 1,500 rpm.

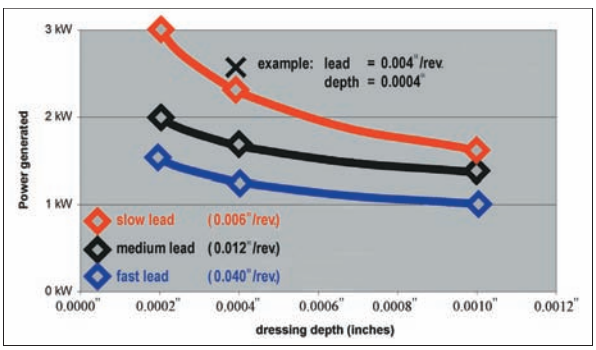
I load up the table and remove about 0.025" in 50 passes with a depth of 0.0005" per pass, adjusting the table speed depending on how it sounds. If I go too fast, I get chatter and grinding burn. I shoot for a surface finish of 40 to 50 R_a.

Grinding takes nearly all morning and is killing productivity. How can I grind these parts faster?

The Doc Replies:

A lot of people have the “grind faster” mentality and want to speed things up. However, with a little strategy, more significant gains can be made. Here’s what I recommend.

1. Switch to a wheel with larger grits, such as a 46-grit wheel. Larger grits mean higher metal-removal rates. They also impart a rougher surface finish, but your finish requirement isn’t that demanding, and you can dress the wheel fine before finish grinding to achieve a finer surface finish.



J. Badger

2. Use a 2"-wide wheel. You can buy a 2"-wide wheel that is recessed in the middle to accommodate an adapter that fits 1"-wide wheels. Now you’ve got double the wheel width. Just make sure you’re within the machine’s safety requirements.

3. Dress and grind aggressively. Dressing a wheel timidly creates a closed, dull wheel. Then, when you increase the mrr, chatter and burn results. The figure on page 32 shows the amount of power generated when surface grinding HSS with different dressing conditions, using a wheel similar to what you’re using. Power generation is proportional to both heat generation, which causes burn, and to forces on the wheel, which cause chatter. Dressing deeper (from 0.0002" to 0.0004" to 0.001") results in less power generation. Also, dressing with a faster lead—the speed at which the diamond traverses the wheel—also generates less power.

When roughing, the goal is to decrease power generation. Once you do this, you can increase the table speed or DOC without increasing the risk of chatter and burn. Your dressing conditions, marked by X in the figure, result in about 2.6kW of grinding power. If you increase the dressing depth to 0.001" and the dressing lead to 0.040 ipr, meaning 1 second to dress the width of the wheel, you’d drop the power generated from 2.6kW to 1.0kW. That means you can increase either the table speed or DOC by 160 percent without any increased risk of chatter and burn.

The formula for dressing lead is: lead in inches/wheel revolutions = (width of wheel in inches × 60)/(time to dress in seconds × wheel rpm).

If you don’t want to mess with equations, just race the diamond across the wheel and open it up. By going so fast, you’ll miss some grits, but that’s what you want. Then, take another pass or two at the same speed without infeeding the diamond to clean the missed grits.

Of course, surface finish will be unacceptable. But you’re not worried about that at the moment—high stock removal is the concern.

4. Dress timidly before finish grinding. In rough grinding, leave 0.001" or so of stock. Then, dress the wheel timidly, as you were doing before. Dress several passes, as you want to get rid of the residual wheel topography left over from the rough grind. Since the diameter of the grits in the wheel is around 0.010" (formula: grit diameter in inches = 0.6/mesh number), you’ll want to dress almost 0.010" to clear away the previous rough dress.

Now, grind the last 0.001" with your usual, less aggressive grinding parameters to impart a fine surface finish.

5. Consider switching to a harder-grade wheel, such as from H to I or H to J. When grinding more aggressively, expect more wheel wear. To compensate, increase the wheel grade—but be careful. Switching from a 60-grit to a 46-grit wheel usually means the wheel acts harder because large grits are harder to dislodge than small grits. (It’s much harder to dig a boulder out of the ground than a pebble.) You may want to first apply the larger-grit wheel and increase the mrr. If all goes well, switch to a harder-grade wheel and increase the mrr again. △

About the Author

Dr. Jeffrey Badger is an independent grinding consultant. His Web site is www.TheGrindingDoc.com. E-mail grinding questions to him at alanr@jwr.com.