BY KEITH JENNINGS

A visible shop is an enticing shop

Have you engaged in what seemed like a valuable conversation with someone at a business function and then never heard from him again? Unfortunately, many good manufacturing owners, managers and workers fail to recognize the value of networking and follow-up. Keeping your company and key employees visible in the marketplace presents a valuable image of an active, well-informed shop that's eager to earn a prospect's trust and business.

This can be a time-consuming endeavor that's commonly delayed and excused because "I'm too busy." That's usually a poor excuse because you always should be busy. The point is prospects will have a lot more confidence in your capabilities if they see you at various functions, or when they hear others singing your praises as a knowledgeable industry person, even from those who aren't your customers.

For most shops trying to prove themselves in a competitive marketplace, getting away from that CAD computer and being visible at business events can have a huge impact on your reputation.

For instance, if you're active in your community, other business leaders should gain insight into your shop and enjoy discussing matters with you. Sometimes they may even solicit you for business, and then additional business is often generated by word of mouth.

Staying active is easier said than done, though. Occasionally, I attend chamber of commerce functions and meet entrepreneurs and business owners who present an image of being busy, visible and hungry to earn your business. They seem to be around at every opportunity, helping, talking shop and expanding their networks. Within a few months, however, many of them lose that spark and are no longer active in the community interfacing with other businesspeople and showing that hungry attitude. There are many reasons for this. For one, it's not easy to maintain that image month after month and year after year, conveying that infectious attitude of entrepreneurial desire and business savvy.

In the case of job shops, immersing oneself in the business world by networking and even taking volunteer positions can project a valuable image about yourself and your shop that generates credibility and respect. You're perceived as being an expert, which you probably are. This participation shouldn't be reserved for owners or managers either. In many cases, allowing other employees the same opportunity takes some of the load off owners and managers and increases the number of people spreading your shop's message.

Listening is also important. Many of the other businesspeople you talk with may have valuable information to share about your industry, competitors and your market in general if you're willing to listen.

Now, if you're spreading falsehoods about your company and making claims that you can't or don't keep, it's a given that your word will become meaningless in short order. The proper technique assumes that you're ethical and speaking the truth about your capabilities.

If you're one who looks for any possible reason to stay in the shop, avoiding travel or getting in front of others, then hopefully your history and reputation will carry you for many years to come. Of course, there are few shops that have that luxury. For most shops trying to prove themselves in a competitive marketplace, getting away from that CAD computer and being visible at business events can have a huge impact on your reputation. That interest and desire will make you a player, even if competitors have more capability.

Think about it. When you're seeking a good supplier, what causes you to select one over another? Most likely, you'll select the one that knows its products or services and makes useful recommendations, describes the latest techniques, shows up on time with all the necessary materials and maintains a great Web site. A potential supplier that isn't aggressive, doesn't communicate well, is not informed about new technologies and has a Web site with 4-year-old content will not give you much confidence and probably won't earn your business.

Having numerous contacts and acquaintances spreading the virtues of your company generates business. Most people love to tell others about all the smart people they know, so be one of those smart people they know.

About the Author

Keith Jennings is president of Crow Corp., Tomball, Texas, a family-owned company focusing on machining, laser cutting, metal fabrication and metal stamping. He can be e-mailed at kjennings@jwr.com.

Hardness conversion

BY EDMUND ISAKOV, PH.D.

Hardness is a measure of a material's resistance to surface indentation or abrasion. There is no absolute scale for hardness. To express hardness quantitatively, each type of test has its own scale, which defines hardness.

Indentation hardness obtained through static methods is measured by Brinell, Rockwell, Vickers and Knoop tests. Hardness without indentation is obtained by the dynamic (rebound) method. A typical dynamic method is the Scleroscope hardness test, also known as the Shore method.

Traditionally, selections of machining parameters depend on hardness of workpiece materials, cutting tool materials (HSS, cemented carbides, ceramics, polycrystalline cubic boron nitride and PCD) and the cutting tool geometry.

In the metalcutting industry, the Brinell hardness test performed at a 3,000-kgf load is the most commonly used to define workpiece hardness. The most accurate readings are between 81 HB and 444 HB when hardened steel ball indenters are used. Tungsten-carbide ball indenters are used for hardness numbers between 444 HB and 627 HB. Although the Brinell hardness scale extends up to 745 HB, the values above 627 are beyond the normal range and should be used for information purposes only.

If a workpiece's hardness numbers are available only in the Rockwell B-scale (HRB) or Rockwell C-scale (HRC), those hardness numbers must be converted into the equivalent Brinell hardness (HB) numbers at a 3,000-kgf load.

The most accurate readings of the HRB numbers are between 60 and 100 HRB and those of the HRC numbers are between 20 and 59 HRC in respect to the most accurate readings of HB numbers. Values more than 100 HRB and less than 20 HRC are beyond the normal range and should be used for information purposes only.

The author developed 14 linear regression equations to convert HRB and HRC numbers into HB numbers. There are six equations to convert Rockwell hardness numbers measured on the B-scale. The resulting HB numbers are 99.5 to 100 percent accurate. And there are eight equations to convert Rockwell hardness numbers measured on the C-scale. The resulting HB numbers are 99.7 to 100 percent accurate. These 14 equations are summarized in the table.

As can be seen from the table, the correlation coefficients range from 0.9971 to 0.99996, indicating high accuracy in converting HRB and HRC numbers into HB numbers.

About the Author

Edmund Isakov, Ph.D., is a consultant and writer. He is the author of several books, including "Engineering Formulas for Metalcuttting" (Industrial Press, 2004) and "Advanced Metalcutting Calculators" (Industrial Press, 2005). He can be e-mailed at edmundisakov@bellsouth.net or reached at (561) 369-4063.

Conversion into Brinell hardness (HB) numbers.

Range of hardness numbers	Equations to convert into HB	Correlation coefficient
(60.0 to 69.9) HRB	HB = 2.136 × HRB – 28.345	0.9990
(70.0 to 79.9) HRB	HB = 2.579 × HRB – 59.950	0.9991
(80.0 to 85.9) HRB	HB = 3.303 × HRB – 117.887	0.9997
(86.0 to 89.9) HRB	HB = 4.031 × HRB – 180.006	0.9971
(90.0 to 95.9) HRB	HB = 5.289 × HRB – 293.668	0.9986
(96.0 to 100.0) HRB	HB = 6.736 × HRB – 432.340	0.9996
(20.0 to 25.9) HRC	HB = 5.284 × HRC + 120.491	0.9997
(26.0 to 29.9) HRC	HB = 6.888 × HRC + 78.838	0.9999
(30.0 to 35.9) HRC	HB = 8.195 × HRC + 39.932	0.99996
(36.0 to 39.9) HRC	HB = 8.718 × HRC + 21.825	0.9997
(40.0 to 45.9) HRC	HB = 10.057 × HRC – 31.761	0.9994
(46.0 to 49.9) HRC	HB = 12.642 × HRC – 150.881	0.9989
(50.0 to 55.9) HRC	HB = 15.979 × HRC – 318.988	0.9998
(56.0 to 58.9) HRC	HB = 18.694 × HRC – 469.983	0.9993

Layout and measurement tools

BY FRANK MARLOW, P.E.

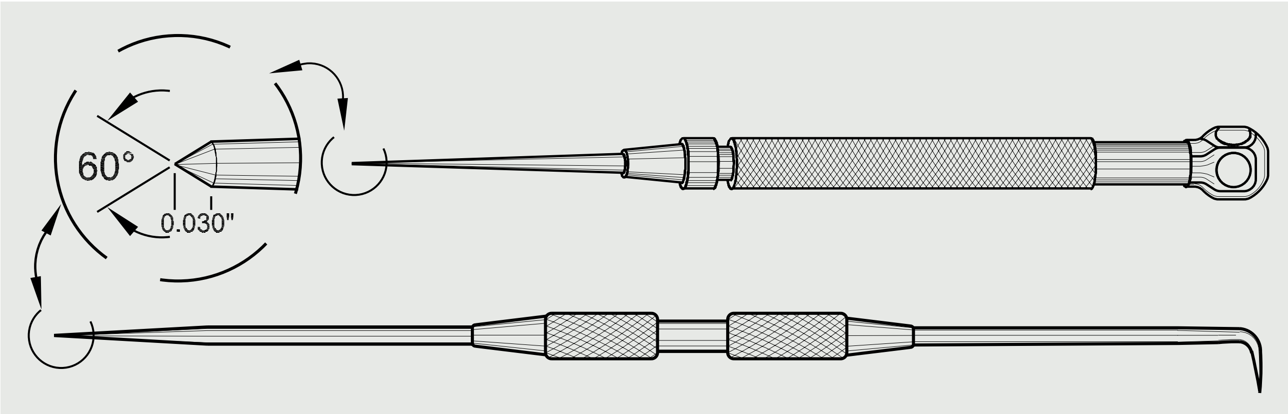
The first in a series of columns drawn from the book "Machine Shop Essentials: Questions and Answers."

Fewer than 25 different tools are required to perform basic machine shop layout and measurement. Many of the hundreds of others tools are merely special-purpose versions of these basic tools that speed, simplify or improve the accuracy of a particular task. Five of these tools are examined here.

■ **Scribers apply scratch marks to work, indicating the position of holes, openings and cut lines.** High-quality scribers have hardened and finely tapered points so they can get close to the rule or straightedge, minimizing errors. The last 0.030" of the scriber should be sharpened to a 60° point by spinning it rapidly in a lathe and tapering it with a flat oilstone. Premium scribers have carbide points. To protect scriber points from becoming blunted, reverse them in their holders or store them inside drilled-out dowels.

■ **Micrometers are the machinists' oldest precision measurement tools.** All are capable of measuring to 0.001" and the high-precision ones to 0.00005". Many machinists prefer to use digital slide or dial calipers because of their direct digital readout and 6" measurement range rather than a typical micrometer with its 1" range. However, machinists revert to micrometers for fine work. Many micrometers are designed for one specific application. High-quality micrometers have lapped carbide measuring faces, a locking mechanism to hold a dimension and a mechanism to provide constant spindle pressure when tightening.

■ **Digital slide calipers are the principal measuring tools in many shops, replacing traditional inside and outside calipers.** Additionally, they make depth measurements and accurately measure distances from an edge for making a line parallel to an edge. They may be set to zero or set to store a reading at any point. Digital slide calipers switch between inches and millimeters at the push of a button. By zeroing the calipers when closed, then opening them to the target dimension and zeroing again, the DRO shows the exact distance remaining to the target dimension. This function

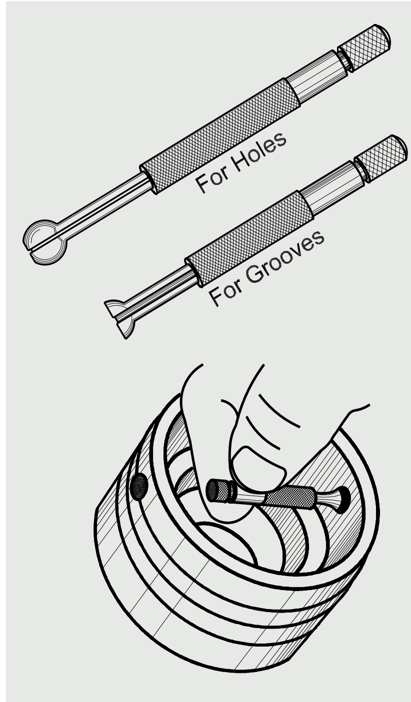


All images: Pamela Tallman

The last 0.030" of the scriber should be sharpened to a 60° point by spinning it rapidly in a lathe and tapering it with a flat oilstone.

speeds work and helps eliminate errors, particularly on lathes and milling machines where a part's dimension is reached by taking a series of cuts. The digital caliper allows a machinist to display the "distance to go," or incremental distance remaining, rather than the absolute dimension. When the absolute dimension is shown, the machinist is forced to calculate how much additional cutting is needed, but with the distance to go, he can just crank in this figure to reach the target dimension, reducing the possibility of error. Most digital slide calipers are made of stainless steel, and the best ones have carbide faces for enhanced wear resistance. They are typically accurate to ±0.001" over 6" and have a resolution of 0.0005".

■ **Hole measuring devices are not always needed because dial or digital calipers often provide the necessary accuracy.** However, for holes smaller than about ½" and when a dimensional tolerance of ±0.002" or better is needed, tools specifically designed for measuring holes are required. There are two relatively inexpensive tools for this purpose: small-hole gages for holes from 0.125" to 0.500" in diameter and telescoping gages for holes from 0.500" to 6.000" in diameter. Both tools capture the diameter and make it available for measurement by a micrometer or caliper. Neither device provides a direct readout, but more expensive devices do and those should be used when making frequent diameter measurements. Accurate measurements using a small-hole gage are obtained by slightly “rocking” the gage in the hole to be measured. This



A small-hole gage is applied to check hole size in a part. Accurate measurements are obtained by slightly “rocking” the gage in the hole to be measured.

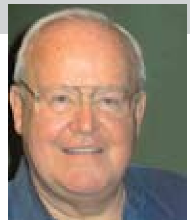
guarantees contact at the true center. The final size is obtained by measuring over the ball with a micrometer.

■ **Radius gages are used to check or lay out concave or convex radii or fillets.** They are made of stainless steel sheet, have smooth edges and are marked with their radius. They are available individually or in sets. Sets are available in fractional and decimal inches, and in metric. Each gage has five radii for use in different situations.

About the Author

Frank Marlow, P.E., has a background in electronic circuit design, industrial power supplies and electrical safety and has worked for Avco Missile Systems, Boeing, Raytheon, DuPont and Emerson Electric. He can be e-mailed at orders@MetalArtsPress.com. Marlow’s column is adapted from information in his book, “Machine Shop Essentials: Questions and Answers,” published by the Metal Arts Press, Huntington Beach, Calif.

BY GEORGE WEIMER



Which way to the universal machine?

The "universal machine tool" has been the shop manager's dream for generations. It does everything: milling, turning, drilling, grinding and other operations needed to make finished parts. In effect, such a machine reduces material handling time between operations to near zero. It also reduces operator time and setup costs.

There are two basic approaches to creating the universal machine—multitasking and manufacturing cells. Which is the better approach? Some say multitasking is a more costly alternative and appropriate only for large, complex parts while manufacturing cells integrated with robots are good for bigger lot sizes and simpler geometries. Others state that quality improves greatly with multitasking.

"Multitasking machines are usually very expensive compared to cell arrangements for the same work," said Jeff Law, product marketing manager for Haas Automation Inc., Oxnard, Calif. He added that there are other aspects of multitasking that could present problems, such as their possibly being tougher to program and having longer cycle times. "For certain classes of parts, however, they can be ideal. Multitasking for crankshafts makes sense."

There are two basic approaches to creating the universal machine—multitasking and manufacturing cells. Which is the better approach?

Still, it's the cost of setup that keeps some shops from purchasing multitasking machines, said Law. He noted that a manufacturing cell with two or three machines and a robot can compete on cost with a multitasking machine.

According to Makino Inc., Auburn Hills, Mich., productivity increases dramatically when a cell is implemented and there are many levels of cellular integration that can be undertaken. Multitasking, according to the machine tool builder, is rarely a replacement for a cell.

Yet multitasking is "the fastest growing segment of the machine tool market," said Chuck Birkle, vice president of marketing for the Cybernetic division of Mazak Corp., Florence, Ky. He said his company recently introduced the Integrex i-150 multitasking center for making small, complex parts. Sensors in the spindle housing analyze temperature, vibration and displacement. The machine "talks" to the operator and can do 5-axis surfacing with little intervention.

When parts are moved between manufacturing cells with two or more machine tools and robots, tolerance buildup can occur, according to Mazak's Birkle. "The less we move the part around, the less trouble we have. In effect, with multitasking there is no tolerance buildup. Multitask machines offer a vast quality improvement," he said.

Birkle, Law and many shop managers agree that for high-volume parts with relatively simple geometries, the cell approach may be the way to go.

"Cellular versus multitasking? Both have value," said Scott Brockie, president of Classic Turning Inc., a machine shop in Jackson, Mich. More complex parts? Take a look at multitasking. But, for high-volume, simple geometries, try the cell approach. Brockie reported that Classic will be creating a hybrid approach to reap the best of both concepts. "We are putting two multitasking machines together in a cell," he said.

"The biggest difficulty in terms of multitasking is finding and training the right operators," Brockie continued. "Multitasking operators are generally younger and more computer literate." In effect, the Game Boy generation is more attuned to the operating requirements of technologies such as multitasking.

"Cellular will often be less costly today," Brockie added, "but the cost is pretty close to multitasking. Over time, there will be an evolution of multitasking machines that will eliminate cells, say in 10 to 15 years."

So, are we on the way to the universal machine, the machine tool that does it all on one base? "Yes," said Brockie. When? Well, compared to the machines of 40 years ago, which this writer can recall learning about with real excitement, today's technology is downright science fiction. But what about a few years out?

Customers want more and more operations in one machine or in cells. Mazak's Birkle predicts machines like that will be exhibited at IMTS '08. "Some day soon, we will be doing heat treating, grinding and hardening as well as standard machining on the same machine tool. We are always trying to raise the intelligence of the spindle, which has more and more sensors," he said.

The industry today is moving towards more artificial intelligence in machine tools. Forty years from now? The only limit is the industry's imagination. We will likely see more operations on a single machine tool as both manufacturing cells and multitasking techniques provide more automation over time. In years to come, that shop manager's dream will become a reality. What do you think?

About the Author

George Weimer, a freelance writer based in Lakewood, Ohio, has an extensive background in the metalworking industry's business press. Contact him by e-mail at gweimer@jwr.com.

Drilling inside the box, thinking outside it

BY BILL KENNEDY, CONTRIBUTING EDITOR

MiniMachine Inc. specializes in the demanding production of tiny, precise parts. One "large" 1.94"-long × 0.50"-dia. part, however, required creative thinking to efficiently machine features that included a relatively deep, off-center axial hole and half a dozen drilled and tapped 0.070"-dia. holes.

The part, a cylindrical housing, was machined from cold-drawn 12'-long × ½"-dia. bars of C-36000 free-cutting brass. To handle production lots of 2,000 to 3,000 pieces, the Bend, Ore., shop used a 7-axis Citizen L20 Swiss-style automatic lathe.

The first operation was facing the bar end with a 35° carbide turning insert from Swiss-tooling specialist Aplitec, supplied by FLP Tooling. Next came drilling of an axial hole located 0.010" off the bar's central axis. With the lathe spindle stopped, a 0.316"-dia. carbide drill mounted in a driven end-working tool station spotted a pilot hole. The drill ran at 3,000 rpm and a feed rate of 10 ipm to make the 0.050"-deep hole.

Achieving the finished hole's 1.75" depth posed a problem because the maximum depth achievable by drills in the lathe's end-working tool positions is 0.900". Machinist Dusty Perry's solution was to chuck a drill in the machine's secondary, or Z2 pickoff, spindle, which normally holds a workpiece instead of a tool. The spindle offers a stroke travel that would accommodate a drill of sufficient length. Perry programmed a 3"-long, 0.368"-dia. HSS U drill to turn at 2,000 rpm, initially feeding at 20 ipm to a depth of 0.200" and then pecking in 0.150" increments to the final depth. Flood coolant was applied.

Mike Rosenboom, MiniMachine co-founder and vice president, pointed out that the secondary spindle collet isn't the best way to securely hold a drill, so part of the drill shank's OD was ground smaller to create a step that would provide a positive stop and prevent the drill from pushing back into the collet.


MiniMachine

Machining features on this 1.94"-long × 0.50"-dia. C-36000 brass housing required creative thinking and innovative tooling application on the part of MiniMachine.

Perry next machined a 0.062"-wide × 0.355"-deep slot across the front of the part via multiple passes with a 0.062"-dia. carbide endmill run at 4,500 rpm. Then the turning tool applied earlier cut a 0.368" OD radius on the part's front edge. Throughout the job, Perry programmed the toolpaths with Partmaker SwissCAM software, as well as hard codes he wrote.

The next feature to be machined was a ⁵⁄₁₆"-wide × 1.12"-long slot on one side of the part. Perry side milled the slot with a ³⁄₁₆"-dia., 2-flute carbide endmill. He chose climb milling instead of conventional milling. Accordingly, he wrote a program that plunged the endmill 0.090" into the part at the end of the slot nearest the guide bushing, while the program simultaneously told the machine to draw the part back into the bushing. "It cuts in a negative direction," Perry said. The endmill cut the slot at 3,800 rpm and a feed of 5 ipm.

Then another ³⁄₁₆"-dia. carbide endmill, this one featuring 0.030" radii on the cutting edges, descended into the slot and endmilled a pocket in the bottom of the part's central bore. That endmill ran at 4,600 rpm and 5 ipm. This time, the endmill started the cut at the front of the part and moved towards the back.

The next step required drilling and tapping six 0.070"-dia. radial holes. Three of the holes were arrayed 120° apart around the part circumference

and 0.255" from the front end of the part, and three more, similarly spaced around the part, were located about 1.5" closer to the guide bushing than the first three.

Making the small holes required more creative thinking. Rosenboom said the part required a large number of operations, and "we didn't have enough stations on the machine, so we had to start coming up with ways to combine operations on one tool station." MiniMachine sought to drill and tap with one tool and combination drill/tap tools are commercially available, but Rosenboom couldn't find any for making such small holes. Instead, he purchased a few OSG 2-64 HSS taper taps and sent them to Almar Tool & Cutter Grinders Co. Inc. with the request that the front end of each tool be ground to act as a drill for a length of three tap diameters.

"We had to use a very stubby drill to drill the hole, and then have the tap to cut the threads right above that," Rosenboom said. Almar ground a drill point and flutes on each tap, as well as a small relief on the tool body before the tap's cutting edges began.

Perry first drilled all six holes, running the tool's drill portion at 3,300 rpm and pecking in 0.050" increments through the part wall. Then he used a carbide ⅛"-dia., 2-flute, 90° chamfer mill to interpolate a chamfer around the mouth of each hole. After that, the tapping portion of the drill/tap tool threaded the holes via a G84 rigid tapping program.

Next came three steps to minimize secondary operations. First, Perry ran the 0.368"-dia. drill back through the axial hole to remove burrs left from side drilling, tapping and slotting operations. Then, to remove any excess material pushed into the tapped holes by the axial drill, he reran the drill/tap with the G84 program. Last, as a final cleanup, he ran the 0.368"-dia. drill through the axial hole one more time.

Next to be machined was a pair of 0.037"-deep × 0.400"-long V-grooves on the part circumference, located 180° apart and positioned 15° from the cross-slot. The same 90° chamfer mill used earlier ran at 3,500 rpm and 2½ ipm to produce the V-grooves.

In the next-to-last operation on the lathe, an Applitec back-turning tool run at 4,000 rpm skimmed the OD for a distance of 1.40" and then plunged in 0.015" to create a 0.485" OD that extended to the part's end. The tool also cut a chamfer at the end of the part closest to the guide bushing and machined a relief area to make space for a cutoff tool to operate.

When cutoff occurred, MiniMachine collected parts in a unique way. The lathe's end-working slide has a location normally occupied by a cylindrical toolholder. In that location, Perry mounted a tube that was about 0.030" larger than the part diameter. The end-working slide was programmed to move the tube to cover the part just before cutoff. When a 0.078"-wide Applitec tool cut off the part, it dropped into the tube. Each finished part pushed the one ahead of it through the tube, which led to the back of the machine and a collection bucket.

The parts required one secondary operation, a 45° internal chamfer on the back end of the axial hole, which was machined with a 90° chamfer tool on a Hardinge Omniturn lathe.

Regarding MiniMachine's continual efforts to find innovative ways to get parts done efficiently, Rosenboom said: "It's about thinking outside of the box. You can have the best tools in the world, and it's only square peg, square hole. A lot of times it takes some other shapes to make things go together."

For more information about MiniMachine Inc., call (800) 854-9806 or visit www.minimachine.com.

OD chatter and 'suck up'

Dear Doc,

I do a lot of OD cylindrical grinding of stainless steel and hardened steel and experience on-again/off-again chatter problems during roughing. When I do, I slow down the table traverse to reduce the material-removal rate. But then the wheel seems to become dull, generating even more chatter. What can I do?

The Doc Replies:

You're on the right track by reducing the mrr. Removing less material per unit of time results in a lower normal force and, in turn, less chatter.

However, keep in mind there are two ways to reduce the mrr in cylindrical grinding: decrease the DOC, taking, for example, 0.001" off the diameter instead of 0.002", or decrease the traverse speed, which you did.

But when you decrease the traverse speed, you're grinding with a smaller fraction of the wheel. The front is becoming dull, and as a result, normal forces increase. The back of the wheel—the cleaning up part—is now much wider, so it's just riding over the part, dulling more and generating larger normal forces.

There's a simple formula for calculating the part of a wheel removing material:

Width of wheel removing stock (inches) = traverse speed (ipm) ÷ workpiece rotation (rpm).

Let's say a 2"-wide wheel traverses the workpiece at 30 ipm and the workpiece rotates at 120 rpm. That means a contact width of 0.25" (30 ipm ÷ 120 rpm), or 0.25" of the wheel is removing stock and 1.75" of the wheel is cleaning up. That's OK for finishing, but you want more of the wheel active during roughing, say 40 percent, or 0.8" of the wheel (0.40 × 2").

If you increase the feed rate to 90 ipm, 0.75" of the wheel is removing stock, or 37.5 percent [(0.75" ÷ 2") × 100]. That's a reasonable number.

But increasing the traverse speed increases the mrr. So a better option is to decrease the DOC. Now you have less material being removed per unit of time (a lower mrr) and more wheel width removing material.

The concept also works on surface grinders with a rotary bed.

Dear Doc,

I grind a lot of stainless parts, usually around 1" in diameter. I find size, stock the loader and let it run. But when I grind production runs, I find the parts are undersized. How is this possible?

The Doc Replies:

It sounds like you're getting what grinding researchers refer to as "spurious part tolerance due to thermal expansion" and what machine operators refer to as "suck up" because the part appears to suck up into the wheel.


Archives MNATP, Collection Meillassoux

During rough grinding, the workpiece gets hot, and hot things expand. So the workpiece diameter is larger than it would be at room temperature. Then you take the finishing pass to achieve final size and surface finish. You then check size, and it looks OK.

However, that finishing pass was taken when the workpiece was hot—and with a larger diameter. It then cools to a smaller size, possibly becoming out of tolerance. To exacerbate things, if the wheel is progressively dulling throughout the process, it is generating more heat and causing more suck up.

If you do the math, using the coefficient of thermal expansion of the workpiece, you'll find that a 1"-dia. steel part that's at 170° F bulk temperature going into the finish pass—as a result of heavy roughing—will expand in the radius around 0.0005". Aluminum will expand a bit more and tungsten carbide a bit less. That means, after cooling, a steel part will be undersized 0.0005".

And that's just considering expansion because of bulk temperature increase. The local temperature increase at the wheel/workpiece interface can be 1,000° F, making matters worse.

What's the solution? You can try to take shrinkage into account, but that's a risky venture.

A better option is to improve the cooling, both bulk and arc-of-cut cooling. During roughing, blast coolant into the arc-of-cut at a speed that matches wheel speed. That will reduce localized temperature increases and hence bulk temperature. Then, after roughing, douse the entire workpiece with coolant, reducing bulk temperature more. Although the higher the better, dousing doesn't have to be done at high velocity, but coolant does need to cover the entire workpiece. Finally, blast again into the arc-of-cut during finishing to keep temperatures down.

If you still have problems, dress the wheel more aggressively to reduce heat, reduce the mrr when roughing and finishing to reduce heat, maintain coolant at a consistent temperature, allow ample time after roughing for the workpiece to cool during dousing and make sure you still have material to remove during finishing even if suck up occurred during roughing. △

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. The Doc will be presenting his 3-day High Intensity Grinding Course Feb. 5-7 in Mundelein, Ill.