BY MIKE PRINCIPATO

Better business through better altruism

One of my most valued advisers—Rob, a guy who has forgotten more about accounting than I'll ever know and is acutely aware of my rather quixotic management style—gave me a pointed piece of advice several years ago. In response to a presentation I'd just given him and my other advisory board members about the importance of corporate citizenry and community partnerships, Rob replied (in his inimitable, understated prose): "Bullshit. I'd rather hear an explanation of why your net income is falling. No offense, but if this business isn't an income generator for you personally, what the hell's the point?"

He's right. Profits drive all business activity. To steal a line from one of my favorite movies, "The Right Stuff," no bucks, no Buck Rogers.

For entrepreneurs and executives who spend most of their waking hours in the dog-eat-dog world of manufacturing, there are meaningful benefits to participating in charitable works.

But this month's column isn't for Rob, much as I continue to value his unfiltered, hilariously profane and always dead-on financial analysis. The truth is I know I'm a better businessman and, more importantly, a better person because I do occasionally forsake profits in favor of tilting at windmills.

Hold off on the canonization process, however. I'm a long way from being a pure altruist. But that doesn't mean I can't subordinate my capitalistic desires from time to time if I think doing so will make a difference. I don't mean simply writing a check, either. Anyone in the nonprofit world will tell you it's invariably easier to raise money than find volunteers. I mean active participation, from serving meals at a homeless shelter to building houses for Habitat for Humanity, from being a Big Brother to being a foster parent. The needs—and, thus, your opportunities to contribute in a substantial way to those who are less fortunate—are endless.

If you're scratching your head right now, wondering what this has to do with managing your shop, stay with me. For entrepreneurs and executives who spend most of their waking hours in the dog-eat-dog world of manufacturing, there are meaningful benefits to participating in charitable works.

For starters, it offers a different perspective on life's challenges. Moping around over lousy cash flow? Volunteer a few days at your local food bank, and you'll have a whole different view of what it means to really do without. Resentful about the long hours you put in at the shop? Deliver some Meals on Wheels, preferably with the family members who've been feeling neglected while you've been toiling, and you'll meet shut-ins happy to have the 10 minutes of company it takes to receive their lunches. Most of them would trade places with you in a heartbeat. Feeling tired? Next time your kid's team is begging for a coach, volunteer. I guarantee those youngsters will put new life in your step in a way that no health club ever could.

I don't insist that real charity seeks no gain for the giver. That's hypocrisy, frankly; we all enjoy the feeling that comes with helping others, and we, therefore, naturally gravitate toward good works that fit our interests and individual talents. So, volunteer to teach a trade class at your local Boys or Girls Club or community college. Or collaborate with the local Association for Retarded Citizens, an organization that puts highly motivated people with developmental disabilities to work in jobs that are often plagued by high turnover—repetitive assembly and packaging jobs, for example.

Volunteering your time in a hands-on effort for a worthy cause sure beats the hell out of another stiff charity ball or death-by-Power Point charity board meeting. But volunteering shouldn't involve being on a perpetual quest to get your name in the newspaper or win the award *du jour* from the local chamber of commerce. I grew out of that stage years ago and, now, much prefer working toward a life of quiet, substantive, measurable accomplishment.

Maybe you've come to the correct conclusion that outside of your family, the group of people on whom you can have the most positive and enduring impact is your employees. Your active altruism will set an example that—whether you're aware of it or not—will be closely watched, and, if you're lucky, followed by your employees for years to come.

Yep, profits rule. But in the context of other strong management skills, there's a lot to like about a business leader who walks the talk of caring about others both inside and outside the workplace.

So, if you're going to spend any time on extracurricular pursuits, why not focus at least some of that time on altruistic activities that have the potential to improve the life of someone in need, rather than merely your own resume?

About the Author

Mike Principato owns a machine shop in Pennsylvania. He can be e-mailed at ctemag1@netzero.net.

Instruments for gaging bores

A properly functioning bore gage is key to producing bores within tolerance, especially as parts become more complex and tolerances become tighter. Although bore gages come in an almost endless array of styles, there are two basic types: hard, or limit gages, and variable-reading gages.

Hard bore gages are GO/NO-GO plugs—a machined cylinder of a known size, usually made of hardened steel or tungsten carbide, which fits into a bore of a specific size. The GO member is ground to the low-tolerance limit of the bore to be checked. The NO-GO member is ground to the high-tolerance limit. A hole that accepts the GO member, but does not accept the NO-GO member is good.

These basic metrology tools don't have any moving parts. They indicate if a hole is good or bad, but provide no other information.

Variable-reading bore gages incorporate some kind of display, such as a dial or digital indicator, and some physical means to detect variation in bore size. This is most commonly accomplished with mechanical contacts, but other methods, such as air pressure or lasers, are also used.

One of the advantages of variable-reading gaging over hard gaging is the ability to send information to a data-collection system. A variable-reading gage can help an operator reach a certain size hole in efficient stages.

Variable-reading bore gages can be further divided into dedicated and adjustable types. Dedicated bore gages measure variation of one specific bore diameter.

Indicating plug gages are a good example. These are similar to the GO/NO-GO gages in that they have a center sleeve that either fits or doesn't fit into the hole. But they also have two or three movable contacts that provide a reading. Indicating plug gages are the most accurate and repeatable type of bore gage, requiring minimal operator skill, but they have a limited range. Because they remain centralized at all times, indicating plug gages can be rotated in the bore, or slid lengthwise, which is good for finding irregularities such as out-of-roundness.

Adjustable bore gages can be used in a range of hole sizes. They incorporate measuring contacts that are assembled to the approximate size of the bore. A tipping bore gage is an example of an adjustable bore gage.

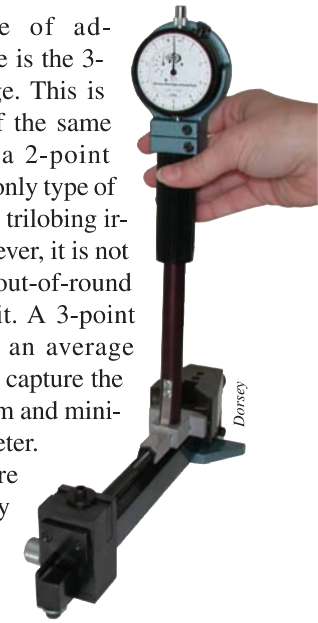
A basic tipping bore gage has two measuring contacts—one sensitive, movable contact that takes the reading and transmits it to the indicator, and an opposing reference contact that is always stationary. The reference contact can be adjusted to any position within the range of the gage. Once adjusted, it is locked in place. The reference contact often has different length contacts available for different size holes.

Most tipping gages have some way to centralize the measuring tool when it is in the bore. This feature ensures that the gage is checking the diameter and not a smaller chord, which is a straight line joining two points on a curve.

When the gage is inserted into the bore, the movable contact is depressed. Keeping the stationary contact hard against one side of the hole, the operator "rocks" the gage, causing the sensitive contact to actuate the indicator as it retracts, then extends.

When the contacts are oriented exactly perpendicular to the bore's axis, the sensitive contact is depressed to its minimum position, indicating the true diameter of the bore. An operator can utilize a tipping bore gage to check out-of-round conditions and other bore irregularities.

Another type of adjustable bore gage is the 3-point contact gage. This is used for many of the same applications as a 2-point gage, but it is the only type of gage that can find trilobing irregularities. However, it is not possible to find out-of-round conditions with it. A 3-point gage only gives an average reading; it cannot capture the absolute maximum and minimum of the diameter.



A DBL series dial bore gage, being used in a DBM-1000 setmaster, from Dorsey Metrology.

Adjustable bore gages are usually set to a master ring. A master ring, or ring gage, is basically a bore of known size. It is typically made out of steel and is chrome-plated.

Whether a bore gage is fixed or variable, it must be maintained properly to work effectively. Limit gages are subject to wear, especially the GO member, so this must be verified periodically to make sure it's to size. A worn GO member allows parts with bores below the low-tolerance limit to be accepted as good.

Variable-reading gages must be checked periodically to make sure that moving parts are working smoothly. Contact points must be checked for wear. Any contact with a flat end or other visible sign of wear must be replaced. The entire gage should be calibrated, or checked for linear accuracy, on a periodic basis.

Special thanks to Dorsey Metrology International, Poughkeepsie, N.Y. For information about the company's dimensional measurement products, visit www.dorseymetrology.com, call (800) 549-4243 or ***enter #430 on the Information Services card.***

The easy way

BY JAMES A. HARVEY

I remember the first time I tried to line up a large plate in a milling machine. I must have been back and forth over the plate 20 times before I got it straight. I remember thinking, "there must be an easier way."

In our never-ending quest to find easier ways of doing things, we can't help but learn a trick or two. Sometimes, we figure it out for ourselves and other times we have to be shown. Either way, finding an easier way to do something is almost always faster and often more accurate.

■ Line up a vise the easy way.

If you do the procedure properly, you can line up a vise with one or two passes of your indicator.

Start by lightly tightening one side of the vise to the mill table. Run an indicator slowly from the snug side of the vise toward the loose side, tapping the vise in as you go. If you get to the end of the vise jaw before it is square, then go back (rapid traverse) to the snug side and repeat the process.

Don't try to tap the vise in by going from the loose side to the snug side. You'll probably get messed up. After the vise is square, tighten both sides.

■ Rough-cut slots by plunging in the Z-axis.

One way to make an accurate slot with a conventional milling machine is to first drill and then bore each end of the slot to finished size with an on-size endmill. Then go back with an undersize endmill and rough the center section by plunging the endmill in the Z-axis as you step-over. This plunging method works well for removing material quickly.

An endmill has a tendency to pull to one side when plunging in this manner, so you may want to bias the endmill to one side. Once the center section has been roughed, you can go back and finish the slot with the on-size endmill.

■ Use double-sided tape to hold thin, sheet-like workpieces.

Double-sided, or double-sticky, tape can hold parts that have a lot of surface area. The tape works well when the pressure of the cut forces the workpiece down onto the tape, such as when fly cutting.

In some cases, you may be able to do side milling, such as pocketing, if you keep cutting pressures light. When pocketing, avoid letting the endmill cut into the tape so the tool doesn't get gummed up.

Don't use coolant with double-sided tape because it causes the tape to release.

■ Construct a 'poor man's' universal angle fixture.

Occasionally, you may need to machine a feature into a part on a compound angle. A variety of ways exist to do compound-angle machining, such as tilting the mill head or mounting the workpiece on a compound sine plate or in a universal angle vise.

One way to easily machine a feature on a compound angle is to use a poor man's universal angle fixture. The fixture is nothing more than a plate mounted on top of a round bar. The fixture can be rotated and tilted in a milling machine vise to provide a range of angles. In other words, the fixture could be used to drill a hole in a part, for example, with a hole at angles relative to the base of the part in both X and Y axis at the same time.

To drill a hole on a compound angle, first machine a flat seat into the workpiece using an endmill that is as large as the diameter of the hole you want to drill. Then center-drill and drill as you would normally.

■ In a milling machine, use an indicator and calculator to find the exact center of a plate.

This is a precise way to find a plate's exact center and the best method when extreme accuracy is needed. A properly functioning digital readout is essential for accurate results.

J. Harvey

One way to easily machine a feature on a compound angle is to use a simple universal angle fixture.

Start by mounting a horizontal indicator in the spindle and positioning it off to one side of the plate. Sweep or rotate the indicator point across the side of the plate so that you get a few thousandths of an inch preload on the point. Set the indicator dial to zero at the point of greatest deflection and then zero the DRO.

Avoid touching the indicator for the remainder of the process. Raise the indicator and move across the table so the spindle goes to the opposite side of the plate. Sweep the indicator across this opposite side while carefully moving the table until the indicator reads zero again.

Make a note of the total table travel shown on the DRO. That number divided by two is the exact center of the plate in that axis. Repeat the process for the other axis to locate the exact center of the plate in two axes.

About the Author

James A. Harvey is a machinist and plastic-injection moldmaker based in Garden Grove, Calif. He can be e-mailed at HarvDog42@aol.com. Machining Tips is adapted from information in his book Machine Shop Trade Secrets: A Guide to Manufacturing Machine Shop Practices*, which is published by Industrial Press Inc., New York. The publisher can be reached by visiting www.industrialpress.com or calling (212) 889-6330.*

Tiny tapping

BY BILL KENNEDY,
CONTRIBUTING EDITOR

At his High Vac Co. shop, Bob Ghamandi machines special and standard components for the semiconductor, bellow, thin-film and high-vacuum industries. One part, designed for a machine that measures coatings on semiconductors, posed both toolmaking and fixturing challenges.

Ghamandi programmed the part manually on the 5020 control of an Okuma MC40VA vertical machining center. The workpiece was a 4½"×3"×2½" piece of 316 stainless steel. To begin, Ghamandi clamped the stock in a Kurt vise and machined the top and periphery to establish basic dimensions. He also created other features via circular interpolation, including four pads at the part corners.

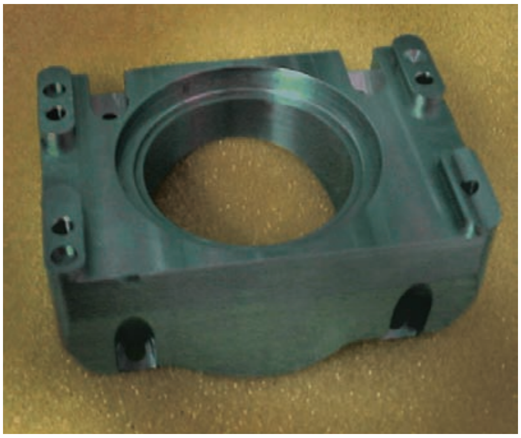
Ghamandi roughed with an OSG ¾"-dia. "hogmill" at a spindle speed of 450 rpm and a feed rate of 2.5 ipm, leaving about 0.005" of excess stock. Then he drilled a hole nearly through the part with a Sandvik Coromant 1½"-dia. indexable-insert drill, run at 625 rpm and 1.5 ipm. He didn't drill through because he planned to complete the hole when machining the part's other side. After creating two steps in the hole with the roughing endmill, he finished the hole and the rest of the part on that side with an OSG 1"-dia. finishing endmill with a 60° helix at 800 rpm and 5 ipm.

Next, Ghamandi used an OSG 45° countersink-angle, carbide-insert tool, run at 2,000 rpm and 2 ipm, to spot and countersink locations for four holes in the pads, which would be tapped to receive ¼-100 screws. He drilled the ¾"-deep screw holes with a 0.234"-dia. cobalt-HSS drill at 950 rpm and 1.5 ipm, leaving about 0.020" of stock in which to cut the threads.

Finding a way to tap the 100-pitch threads was "pretty tricky," Ghamandi said, "because I couldn't find a tap. I called many places and either they didn't make it or they wanted a few thousand dollars for a special."

So, he tried making his own tap. Ghamandi said: "I used a hardened ¼"-dia. dowel pin. I slotted it first and then did some hard turning. It did work, but it galled up pretty quick."

He then came up with the idea to use a Micro 100 0.187"-dia. solid-carbide boring bar in a Kaiser boring head, run with a G84 tapping cycle on the Okuma VMC. "I operated it like a boring bar and tap at the same time," he said. "I adjusted it out to where I barely touched the ID of the hole, calculated the pitch depth of about 0.019" and made my adjustment on the boring head." The thread would be cut in one pass.



High Vac

This stainless steel part for a semiconductor manufacturing machine took 2½ hours to machine.

Ghamandi tested his method in a plain stainless steel block, and used ¼-100 screws sent by the customer to gauge the results. "I didn't run it too fast," he said. "I tried to mimic a tap, about 150 rpm."

Using the G84 cycle required that he stop the VMC and reset it when the boring bar reached the bottom of the hole. That is because—unlike a G76 fine-boring cycle, which at the bottom of the hole repositions the cutting edge for withdrawal—stopping the feed in a G84 cycle prompts the tool to reverse out, which would have ruined the thread and tool.

After tapping the holes on the first side of the part, Ghamandi flipped it over in the vise. He roughed the part to its final height, revealing the through-hole he had started on the first side. He roughed two more steps on that side of the hole, and then finished the hole and the rest of that side using the same endmills and machining parameters as applied earlier.

The next major challenge was machining four angled surfaces with a tolerance of ±0.1° for each. Ghamandi fabricated a fixture that would not require the part to be removed when machining each angle. To make the fixture, he milled a flat on a 5"-dia. × 8"-long piece of steel stock and screwed to it a 1"-thick plate large enough to support the part. He welded a fence around the plate surface that the part could butt against and clamped the fixture in an indexer.

At the time he was making the part, Ghamandi didn't have a programmable indexing head and worked with a Bison indexer that had only whole-degree stops. However, the angles specified were not in whole degrees. One, for example, "was like 29.7°," Ghamandi said.

To enable him to set the indexer to the required angles, he made setup blocks from 2"×3"×½" aluminum stock. Using the Okuma VMC's iMap programming assistant and a Minicut HSS endmill, he machined four different triangles, the hypotenuse angle of each matching one of the angles specified for the part.

To mill each of the angled surfaces, he clamped the part in the fixture with Mitee-Bite clamps and set a triangle on the mill table. He rotated the fixture and part in the indexer while running an indicator across the triangle and part. When the indicator read zero, he tightened the indexer to hold the angle and confirmed it with an Accupro digital protractor/inclinometer. He milled each angled surface with the roughing and finishing endmills applied previously.

The next operation was cutting an O-ring groove around the around the through-hole. From Accugrind, Ghamandi ordered a 0.078"-dia. dovetail cutter that was so small as to be "scary," he said. He machined the groove in two passes. He generated one side of the groove in one pass and then reversed the toolpath and cut the other side, all at 1,400 rpm and a gentle 0.5 ipm.

The final machining step involved clamping the part in a vise perpendicular to the mill table and engraving the customer's part number with a Micro 100 engraving tool run at 1,500 rpm.

Total machining time for each part was about 2½ hours, and the production run was six parts. Ghamandi, who founded High Vac in 2001, said he is proud of this part, especially since his customer told him that, because of its complexity and precision, other shops would not even attempt to machine it.

For more information about High Vac Co., Inglis, Fla., call (352) 447-7033 or visit www.highvac.net.

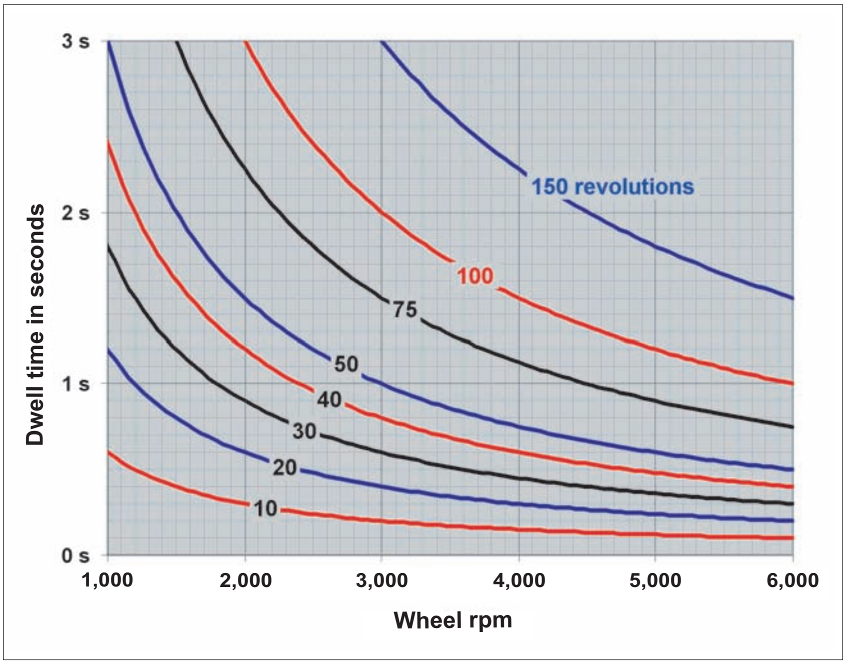
Well-dressed wheel

Dear Doc,

How long should the dwell time be when using a rotary diamond dressing roll, and what happens if I dwell too long?

The Doc replies:

If dressing in the synchronous mode, where the grits in the grinding wheel are moving in the same direction as the diamond grits in the dressing roll at contact, giving a crushing effect, you need a dwell time of 20 to 40 revolutions of the wheel to produce a sharp, open, well-dressed wheel. If dressing in the asynchronous mode, where the grits in the wheel are moving in the opposite direction of the diamond grits in the dressing roll, you need only 10 to 20 revolutions.



Dwelling longer than required dulls the grinding wheel, which increases the amount of heat in the arc of cut.

If you want to close the wheel to produce a fine surface finish on the part, let the wheel rotate more, say, upwards of 150 revolutions in the synchronous mode or 75 in the asynchronous mode. Any more doesn't have any effect. Of course, the number of revolutions also depends on the grit size and the stiffness of the machine, but these figures put you in the ballpark.

How much dwell time does it take for, say, 20 revolutions of the wheel? If the wheel is rotating at 3,000 rpm, you need 0.4 seconds of dwell. If it's running at 6,000 rpm, you need only 0.2 seconds.

Check to see what your dwell time really is. I'll bet it's a lot more than is needed. It's not uncommon to see machines dwelling for 5 seconds or more. At 3,000 rpm, that's 250 revolutions of the wheel. You should not be surprised if you can't get high stock-removal rates when dwelling that long. You're just closing up the wheel—and a closed, dull wheel generates lots of heat. On top of that, you're wasting time.

Dear Doc,

I have a bunch of Normac and Hertlein fluters. I use side coolant nozzles to reduce oxidation burn on the flanks and a main nozzle to cool in the arc of cut, all fed off the same manifold. I find that whenever I open up the side nozzles to reduce oxidation burn, I get more burn in the flute runout. What can I do to have it both ways: less oxidation burn on the flanks and no oxidation burn in the flute runout?

The Doc replies:

The reason your flute-runout burn is getting worse when you open up the side nozzles is that you're getting a pressure drop in the main nozzle.

Remember, the side nozzles don't need to apply high-pressure coolant at a high velocity, but the main nozzle does. The main nozzle needs to have enough pressure and velocity to match the wheel velocity. Opening up the side nozzles deprives the main nozzle of pressure.

What's the solution? Add a separate feed to the main nozzle. Take a 1" hose and connect it from the coolant mains to the main nozzle. That way, pressure to the main nozzle is maintained. I did this last week and it took all of about 30 minutes to hook up. It eliminated oxidation burn in the flute runout, thanks to better cooling. And, thanks to better lubrication, wheel wear was cut by 40 percent. △

About the Author

Dr. Jeffrey Badger is an independent grinding consultant. His Web site is www.TheGrindingDoc.com. You can e-mail him at badgerjeffrey@hotmail.com. Send questions for The Grinding Doc to ctemag1@netzero.net.