JANUARY 2009 / VOLUME 61 / ISSUE 1

STAYING SHARP
manager's desk

By Keith Jennings

Making the best of a slowdown

It's a new year and if the pundits are correct, many of us are in the midst of a downturn, if not a recession. If this is your situation, the stress factor is probably high. Even under these circumstances, however, there are ways to effectively use the extra time to add value to your business.

While most everyone prefers a full and busy workload, that's not always the reality. Identifying some valuable activities and making time to complete them can have a positive impact when your business rebounds. Under the present conditions, one of the best uses of time is employee training.

"Training" has broad meaning and can include education in plant safety, forklift operations, cardiopulmonary resuscitation skills, quality and inspection methods, equipment functions and many others. During a busy period, most of these functions are usually low priorities and may even be forgotten, but a slow cycle can be an easier time to focus on improving employee skills through training.

> *Under the present conditions, one of the best uses of time is employee training.*

Because many small shops aren't in a position to pay for expensive training programs, consider using more experienced employees to facilitate. Or perhaps a cost-effective program through a trade organization or a local college would be worthwhile. Your payroll company may have training resources, or if you use an outside company for safety supplies and site inspections, it may have a good program with materials readily available.

At our shop, we've recently focused on equipment training as one area of need, including cross-training some employees on different machines. Increasing operators' knowledge of your machines and their many idiosyncrasies helps not only the operators, but even related functions like job estimating, where operators and programmers can assist in the estimation process to ensure quotes are accurate and realistic.

A slowdown may also be a good time to implement a more organized procedure for quote reviews, ISO adherence or shop data collection. Many of these tasks require reevaluation from time to time, which is never convenient. The effort can reap rewards later, though.

Software training can also be important. With manufacturing technology becoming more essential, exploiting it can provide a significant return on investment. Even though sitting in front of a training computer all day sounds dull, a slowdown could be the best time to get it done. This could include training with a focus on better utilization of CAD packages, manufacturing ERP applications or even Microsoft Office. We all need this on occasion. Can your employees read and understand their productivity reports? Can employees effectively use e-mail distribution lists and prepare invoices, purchase orders and quality documents? Many probably can, and some probably have no clue. These tools can make shop management easier, especially when employees are trained to maintain them.

How about management and supervisory training? After our company participated in this kind of training a few years ago, our key employees handled situations more professionally and defused potential problems.

While employee training is one of the most important activities that can be accomplished during a slowdown, it isn't the only one. Machine maintenance could be a priority. Or maybe some plant cleanup and rearranging is in order.

However you determine the appropriate task to tackle, make use of the time and put your shop in a better position to compete more effectively when the economy turns around. And it will.

Happy 2009! **CTE**

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.*

Building bridges

By Bill Kennedy, Contributing Editor

Quality Industries Inc. has always been quick to react to new trends by reinventing itself, according to Jim Kaplan, company vice president. With his brother and company President Jerry Kaplan, Jim runs the job shop their grandfather founded in Cleveland in 1946. The shop has continually evolved in response to changes in its customers and marketplace. Over the years, it has produced parts for a range of familiar industries, as well as "a lot of oddball stuff," Jim said.

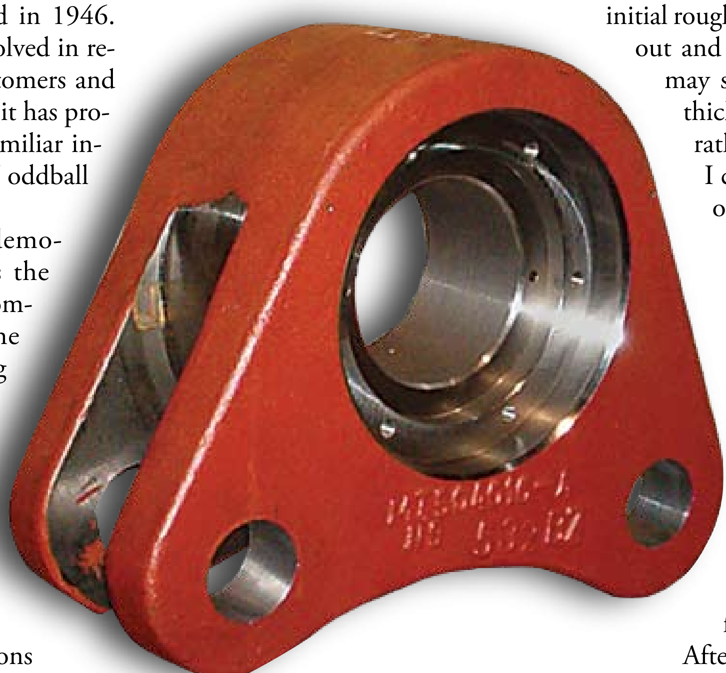
Currently, a maker of demolition equipment contracts the machine shop to make a component called a bridge. The part looks like a giant bushing with two pairs of protruding flanges, or "ears." In use, two bridges connect demolition shears, or claws, to the boom on a piece of heavy equipment.

Quality Industries makes the bridge in three sizes, with basic finished dimensions from 12"×17"×7" to 15"×20"×9" and weights from 140 to 260 lbs. The parts are machined from near-shape castings of heat-treated 4340 steel (32 to 37 HRC). Part runs typically consist of 10 pieces, or five matching pairs.

To create concentric bores and counterbores in the part, initial operations take place on an Acra CST 50200 CNC lathe. The casting's weight and semi-triangular shape make it a challenge to chuck and spin it. The solution is bolt-on counterweights. For a midsize bridge, for example, the counterweight is about 60 lbs., an amount determined by experimentation. The weight balances the part and enables it to spin smoothly.

After indicating a casting to put it on center in the lathe, the shop faces the part's front with a Tungaloy CNMG 643 insert run at 200 sfm, a 0.100" WOC and about a 0.025-ipr feed rate. Jim Kaplan said he runs the insert "somewhat slow" to lessen the tool-damaging effects of voids and hard spots in castings.

Next, to create the part's central bore, Quality Industries uses a quick-change boring system that was designed and fabricated in-house. The tooling consists of a 2½"-dia. bar with bolt-on heads to hold standard ¾"-square-shank turning tools. "I have three or four different heads with tools in them," Kaplan said. "That way, when changing tools, I don't have to change the bar itself."



Quality Industries

Quality Industries fabricated custom tooling to efficiently machine this approximately 200-lb. heavy equipment component made of steel.

The first boring operation opens a 5.499"- to 5.500"-dia. hole about 7" deep through the part's center, using a TMX coated TPG 322 insert running at about 125 sfm and a 0.01-ipr feed. Then another boring head with a WMNG trigon insert enlarges a 9.040"- to 9.045"-dia., 1⅜"-deep counterbore at the front of the part.

Another head and tool change permits the shop to reach in and create a counterbore at the back of the part. The tool is moved into the part, and then the tool back bores towards the front, enlarging the 5.500"-dia. hole to 7.410" to 7.415". Part specifications require a distance of 4.375" (±0.005") between the front and back counterbores. Therefore, after the initial roughing passes, the tool is backed out and measurements are taken. "I may still have 0.010" left on the thickness," Kaplan said, "but I'd rather have the stock than not. I can go back in and remove it on a finish pass."

After ID boring is completed, the part stays on the lathe, which turns the OD between the ears to a diameter of 11.498" to 11.500". The shop fabricated another custom toolholder for this operation. The holder is a steel extension with pockets to hold square-shank MWLNL and MWLNR toolholders for accepting trigon inserts.

After the shoulder and part OD closest to the front counterbore is turned with one tool, Kaplan loads the extension with the opposite hand tool and finishes the rest of the OD, blending the cuts together. "I flip it back and forth, removing about ⅜" of material per side," he said.

Casting variations sometimes require additional facing inside of the ears to achieve the 3.25" width required between them. Kaplan said the surface finish requirement is generally 63 μin. R_a.

Lathe operations for the midsize bridge consume about 8 hours per part, after which it is moved to a Johnford VMC 1124 CNC vertical mill. There, the first set of operations involves opening a 9.48"-dia. (±0.010"), 1.25"-deep counterbore on what was the part's back side when it was clamped in the lathe. To remove material quickly, the shop rough bores the ID with a Sandvik Coromant 2.0"-dia. shell mill tooled with four ½"-

dia. button inserts, helically interpolating in 1/16" steps at 600 rpm and 15 ipm.

Then a 2"-dia., 5-flute, square-shoulder inserted shell mill removes scallops in the part wall left by the roughing cutter, leaving 0.010" to 0.015" stock for finishing. The mill's square shoulder also forms a 90° corner in the counterbore's bottom and establishes its depth.

After a 2-flute, 1.250"-dia. indexable K-tool chamfering tool puts a 3/16"×45° chamfer on the counterbore's outside edge, a solid-carbide endmill finishes the bore. Kaplan said he typically applies a Niagara or an OSG cutter, in whatever diameter between ⅝" and 1" is handy at the time.

Next, jobber-length HSS drills make 10 holes at the bottom of the 9.5"-dia. counterbore. Eight of the holes are tapped to ½-13, and two are reamed to 0.376" in diameter.

Describing drilling and boring holes in the part's protruding ears, Kaplan said, "That's where we make up some time." Previously, the 2⅝"-finished-dia. holes in the ears on one side of the part were roughed with a drill/mill tool and bored to a rough size. Then, after the heavy part was turned and indicated back into place, the drill/mill and boring operations were repeated on the ears on the other side. At that point the holes were finish-bored inline. The process required "a lot of handling and a lot of time," Kaplan said. "It probably took us a couple of hours, per hole, to get it roughed in." Completing all the holes on one part took a day and a half to 2 days.

Kaplan said the local Sandvik Coromant representative suggested using a high-feed plunge mill to create the holes instead. "It has the reach to get through from one side and eliminate all the handling," Kaplan said. "Now we do it in one setup. We use a 1½"-dia. plunge mill, running around 1,150 rpm. It helically interpolates, feeding down 0.040" per pass at about 88 ipm. Each hole in an ear takes about 5 minutes."

After enlarging the holes, the shop bores them to a ±0.002" tolerance with a Criterion or Command boring head. For one part, operations on the VMC consume a full day or a day and a half.

Final part operations consist of drilling cross-holes and grease holes on a Scharrman horizontal boring mill. CTE

For more information about Quality Industries Inc., call (216) 961-5566.

HSM: What's right for you?

By George Weimer

When machining, one goal has always been to increase the spindle speed as much as possible because making parts faster increases profits. But is there a machining speed limit? Not theoretically, but in practice speed is limited by the need to control chatter and tool wear and the spindle speed the workpiece material can tolerate. Also, a higher speed is not always a good idea. A better idea might be to consider the optimal speed for a given operation.

There are primary issues to consider when high-speed machining, said Dr. Tony L. Schmitz of the University of Florida's Machine Tool Research Center. "It is important to consider the process dynamics," he said. "There are particular spindle speeds that enable a higher axial depth without producing chatter. These speeds are directly related to the system's natural frequency. That corresponds to the most flexible vibration mode."

This relationship between the ideal spindle speed and axial depth is demonstrated in stability lobe diagrams. The diagrams plot regions of stability and regions of chatter as functions of spindle speed and DOC. Stable regions are called lobes, and they may be increasingly pronounced at higher speeds.

Interestingly, academic research and industrial trial-and-error efforts have shown that some chatter, particularly "regenerative chatter," where a cutting tool leaves a distorting pattern that causes vibration, can be overcome by increasing the spindle speed. Traditionally, the solution was to reduce the speed and increase the feed. With HSM, that approach can worsen the problem.

Of course, spindle bearings are crucial to any machine tool's operation—especially at higher speeds. There has been some work on hydrostatic bearing technology where a pressurized fluid supports the spindle shaft. However, most bearing and machine tool manufacturers achieve gains by improving contact-type rolling element bearings.

Spindle manufacturer IBAG Switzerland AG, for example, offers various bearing technologies, ranging from hybrid ceramic angular contact ball bearings to air bearings and active magnetic and fluid (oil and water) bearing systems. IBAG North America, North Haven, Conn., produces IBAG spindles in the U.S.

HSM remains a moving target. "An

increase from 10,000 to 20,000 rpm could be considered going to high speed," said Erich Vilgertshofer, general manager, HPT Drive Systems Inc., Newport Beach, Calif. "Our highest speed spindles are up to 100,000 rpm with life-lubricated bearings and 140,000 rpm with oil-air lubrication."

Vilgertshofer added that advances in bearing technology allow greater speeds, loads and precision. "The maximum speed is essentially determined by the OD of the spindle and the bearings," he said. "Physics still limits the maximum; the bigger the slower. There is a trend to increase speeds to achieve finer finishes and higher feed rates. The limits are usually related to the maximum speeds that a material will tolerate or, possibly, by the existing motor spindle's maximum output power and torque at a given range."

Vilgertshofer warned that expertise in tool selection for any given process is "vital" but often not sufficiently covered or ignored altogether.

NSK Americas, Ann Arbor, Mich., offers what it calls "the world's fastest oil-lubricated spindle at 3.8 million DmN (pitch diameter times speed), running at 50,000 rpm." A machine tool demonstrated at the JIMTOF trade show in Japan incorporated a 60mm bore-bearing set based on a "super lean oil lubrication design," said Bimal Nathwani, the company's project manager, machine tool/linear motion.

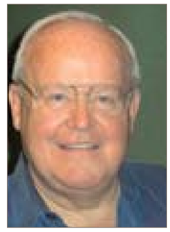
According to Nathwani, bearing design improvements that enhance lubrication performance should keep speeds going up and costs coming down. However, the limitations as to what most bearings can achieve in terms of spindle speeds are "matters of friction and other forces that apply on the rolling elements," he said.

Perhaps it's not high speed itself that should be the goal, but rather the most productive combination of speeds and feeds.

Make sure your machine tool supplier is current on the latest developments and trends, as well as the research being conducted at universities and organizations like the National Institute for Standards and Technology.

More importantly, end users must recognize that a machine tool is only a part—albeit a crucial part—of the total operation. The fastest spindle in the world will not live up to its potential if the material handling or another element of the machining system isn't properly coordinated. **CTE**

About the Author:



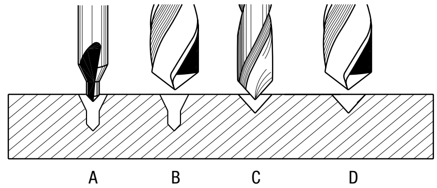
George Weimer, a freelance writer based in Lakewood, Ohio, has an extensive background in the metalworking industry's business press. Contact him by e-mail at gweimer@jwr.com.

Holemaking operations

By Frank Marlow, P.E.

Drilling holes is the most common of all machining processes, and the tool most commonly applied to make them is the twist drill. Twist drills remove metal from holes and reduce the metal to chips in a fast, simple and economical process. Nearly 75 percent of the metal removed by machining is drilled. Drilling is often the starting point for other operations, such as counterboring, spotfacing, countersinking, reaming and boring.

■ Center drilling uses a short, rigid drill to make a cone-shaped starting hole for twist drills or lathe centers. Using a center drill ensures more accurate hole placement on the spindle axis than starting a twist drill on a flat work surface or center punch mark. Because twist drills flex and wobble when starting a hole, there is no certainty they will begin drilling on the spindle axis. There are two center drill designs: the NC spotting and centering drill and the combination drill and countersink. Both work well for starting holes, but only the combination drill makes properly shaped holes for lathe centers. Spotting drills are available with point angles of 60°, 82°, 90°, 118° and 120°. Combination drills, which have 60° point angles, drill a matching hole for lathe centers. They also drill an additional pocket for lubricant so the center does not burn. When using a spotting drill for a subsequent carbide twist drill, the spotting drill should always have a flatter angle than the carbide drill point so the twist drill's chisel edge makes contact with the work first—not the drill edges. For example, apply a 120° spotting drill for a 118° carbide twist drill.

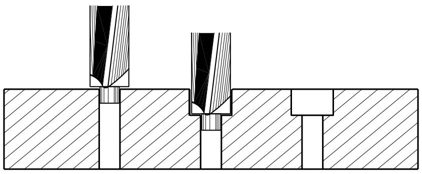


Credit for all images: Pamela Tallman

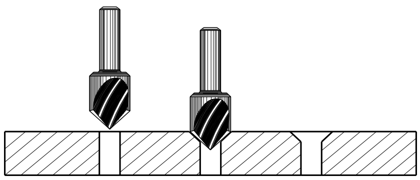
Making starter holes with a combination drill (A), a NC spotting drill (C) and a twist drill (B and D).

■ Counterboring produces a larger, square-bottomed hole in the upper portion of an existing hole and provides space and seating for a bolt or cap screw head below a workpiece's surface. The cylindrical guide, or integral pilot, on the end of the counterbore ensures the enlarged diameter is concentric with the original hole.

■ Spotfacing mills a flat area around an existing hole to make a flat seating surface for a bolt or washer and is usually necessary on castings and parts with sloping surfaces. Failure to spotface an uneven workpiece puts excessive stress on the bolt. On flat work, use a spotfacing bit or counterbore in a drill press, but on a sloping surface, because of the interrupted cut, use an endmill in a milling machine. Spotfacing cutters differ from counterbores because they cut larger diameter holes and sustain greater cutting



When counterboring, a counterbore's pilot end enters an existing hole (left) and aligns the cutter, and increases the diameter of the existing hole (center).



Countersinking uses a cone-shaped cutting tool to chamfer or bevel the edges of an existing hole.

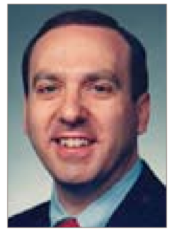
forces, but they work the same way. Large spotfacing cutters have separate centers, allowing the same cutter to work in different hole diameters. On sloped surfaces, spotfacing is performed before drilling. This provides a flat surface on which to start the drill and makes it easier to follow-in straight. Remember to use a counterbore or milling cutter slightly larger than the washer, cap screw or bolt head diameter specified on the print.

■ Countersinking uses a cone-shaped cutting tool to chamfer or bevel the edges of an existing hole, usually so flathead screws can be seated below the workpiece surface. Countersinks also deburr holes. Countersinking can be done with a hand-held drill or in a drill press, lathe or milling machine.

■ Reaming enlarges an existing hole to a precise diameter and improves wall surface finish. It can be performed equally well in a drill press, lathe or mill.

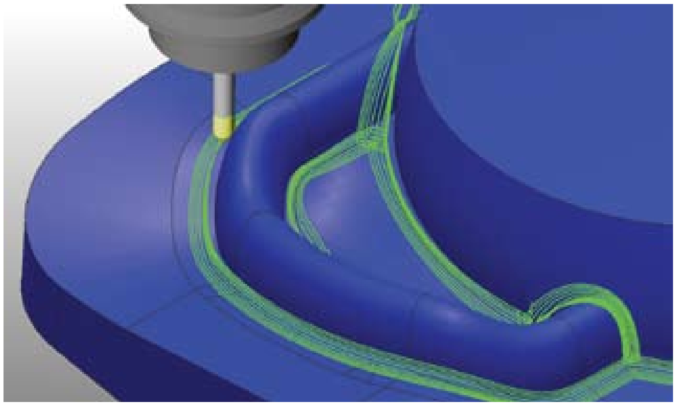
■ Boring enlarges the diameter of an existing starter hole with a single-point lathe tool. It can be performed in a drill press, but is more often done in a lathe or mill. Boring offers precise diameter control and perpendicularity. It typically produces smoother walls than drilling. **CTE**

About the Author: *Frank Marlow, P.E., has a background in electronic circuit design, industrial power supplies and electrical safety. 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.*



SURFCAM Velocity 4 for HSM

By Sashko Kurciski, Surfware Inc.



Automated rest machining is one of the new high-speed machining toolpaths in SURFCAM Velocity 4.

Surfware

SURFCAM Velocity 4 is a new CAD/CAM software release from Surfware Inc. Additions have been made to Surfware's 3-axis and 4- and 5-axis products, based on requests from the company's machinist customer base. Following is an overview of some of those additions in the new release.

With increasing demands for shorter cycle times and improved part quality, high-speed machining is a critical component for the success of many machine shops. The new HSM finishing strategies in SURFCAM Velocity 4 feature smooth and fluid toolpaths that accommodate maximum cutting speeds and shorten the machining cycle.

The rest machining functionality automatically calculates the areas where material remains after the previous operation and generates a toolpath in these areas. There is no need to manually create boundaries. The rest machining differentiates between near horizontal and near vertical corners, and it uses the best-suited cutting strategy to machine them.

The 3-D offset finishing feature provides a fine surface finish regardless of part shape. A constant scallop height is ensured by passes that are at a constant distance from each other along a surface.

To also impart an excellent finish, the software uses offset pencil milling passes to machine a part's corners. It is possible to make single or multiple offset pencil milling passes at a constant distance from each other.

The enhanced steep/shallow feature machines a part's steep and shallow regions as one operation by using the optimal machining strategy for each region. The feature can also enable machining only the steep or only the shallow regions. The adaptive step-down feature automatically modifies the toolpath to ensure this operation leaves a uniform amount of workpiece material.

Smooth and tangential linking moves between the passes enable maximum speeds and feeds. In earlier versions, the retracts consisted of orthogonal rapid moves to the rapid plane. In SURFCAM Velocity 4, these moves are optimized for the part topography, thereby shortening the machining cycle.

Processing speed, powered by multiple processors, is up to 80 percent faster than the previous SURFCAM version, which provides significant time savings.

With SURFCAM's simultaneous 4- and 5-axis machining, full visualization and verification, machinists gain control over multiple-axis machining, ensuring safe tool motion in challenging applications. The multiple-axis module provides support for standard tools with collision checking, gouge avoidance and toolpath containment.

More gouge avoidance strategies have been implemented in the latest version. Users can specify four different cascading tool gouge strategies, which can be applied to a single operation.

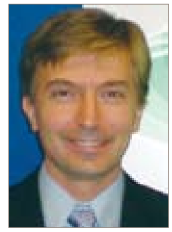
Additional tool control has been added to swarf cutting to allow the tool to be tilted away from the surface to better control the cut. Better surface edge controls have been added to handle geometric inconsistencies found in imported geometries. Axial adjustments to allow tool shifting in respect to the calculated cut point give the programmer added control when needed.

Full surface edge and gap handling has been added, and seven different lead-in and lead-out options can be applied to control tool motion during any event in the cutting process.

New retract types, including plane, cylinder and sphere, allow machinists to tailor the retract motion to be closer to the part type and orientation.

In SURFCAM Velocity 4, an enhanced, intuitive user interface was a key focus. In addition, the help documentation provides up-to-date and useful knowledge that is easily accessible.

The product is available with TrueMill, Surfware's toolpath technology that manages tool engagement to increase productivity and extend tool life. CTE



About the Author:
Sashko Kurciski is product manager for Surfware Inc., Camarillo, Calif. For more information about the company's CAD/CAM software, call (818) 991-1960 or visit www.surfware.com.

STAYING SHARP
machinist's corner

By Michael Deren

How far have we come with quality?

My first manufacturing job was back in 1973. It was at a dark and dingy shop that produced fittings on screw machines and turret lathes out of different materials, such as brass and copper, but I couldn't even tell you what industry the parts were for.

The tolerances on most dimensions were about ±0.015". The parts were checked with GO/NO-GO gages for the threads and calipers for the other dimensions. The parts were only spot checked during production and then shipped.

Several years later, I worked as an inspector for another company. A huge stepping stone at that facility was its use of NC mills and lathes. There, I had access to outside and inside micrometers, profilometers to check surface finishes, pi-tapes for checking diameters, an optical comparator and various other inspection equipment. The shop routinely achieved tolerances of ±0.005" with an occasional ±0.002". Previously, that shop only had a final inspection program, but I implemented in-process inspections, which saved untold machining hours and significantly reduced scrap.

Eight years later, I was an applications engineer for a builder of multiple-axis turning centers that could machine to tenths and incorporated in-process inspection capabilities using touch probes. The primary customers were automotive and marine engine builders. At that time, we could also offer customers automated offline probe systems that checked dimensions and automatically updated cutter compensation on the turning center for whatever tool was wearing. There was even a broken tool detector.

Fast forward a few years and I'm employed at a job shop that purchased a large coordinate measuring machine to verify dies being machined. We thought the CMM would help improve part quality and in a way it did. But what it really showed was how close the parts were to being out of tolerance. Not just new parts, but previously made parts we thought were good. Don't get me wrong, the parts weren't bad—they just weren't as good as we thought. Having the CMM forced us to make better parts.

Now I work for a shop that produces aircraft engine and power generation components. We have several large CMMs, a couple of portable CMMs and a variety of other inspection devices to support our quest for quality. Many of the machines have built-in touch probes to verify part quality. Not only do we have in-process and final inspection personnel, but also quality engineers who evaluate and document data for customers. We have to provide 100 percent inspection for many part dimensions, and some parts are so particular they require a cleanliness test.

As my experience shows, improvements in inspection equipment and techniques have evolved dramatically over the years.

As my experience shows, improvements in inspection equipment and techniques have evolved dramatically over the years.

However, we still have the human factor to contend with. Despite our best efforts, human error can still creep into any process. With fully automated machines, which are great for production runs, you can virtually eliminate bad parts. Once a machine is programmed and tooled correctly, it can run unattended and repeatedly make good parts.

When I was an inspector, there was a machinist, as good as he was, who occasionally made bad parts. He always had a good part on his bench ready for inspection, but when he made bad parts, he would bury them in the pile of completed parts, hoping to get them past the inspector. Fortunately, I caught them most of the time.

I don't know if it's a result of an individual's poor work ethic, lack of training or what, but until that problem can be corrected, quality will never be a given. It will always be something to strive for. CTE

About the Author: *Mike Deren is a manufacturing engineer/project manager and a regular CTE contributor. He can be e-mailed at mderen1@roadrunner.com.*