FEBRUARY 2002 / VOLUME 54 / NUMBER 2

► BY KIP HANSON

A Creative Turn

Suggestions for turning long, thin parts.

You know the parts: shafts, needles, actuators, pistons and cylinders. They're long, skinny and tough to turn, because they usually must be machined to close tolerances and require fine finishes.

The problem is that part deflection tends to occur as the workpiece length-to-diameter ratio approaches 3:1. Deflection causes unwanted tapering of the workpiece and tool chatter. Worse, the part might deflect enough to climb above centerline, chipping the cutting tool or even shearing off the workpiece.

There's no doubt that long, thin parts are challenging. There's also little doubt that you're probably going to have to keep producing them. This article suggests ways to make the job as stress-free as possible.

Swiss Choice

Let's say somebody in the estimating department accepts an order for 500 roller shafts like the one shown in Figure 1. The shaft is to be made from 8620 steel, measure 3½" in length and ⅜" in diameter, and have a groove cut into it. Some internal machining will be required, too. What would be the best way to tackle such a part?

If you have a Swiss-style lathe, there would be no problems. These types of machines make short work of close-tolerance, fine-finish, long-length-to-diameter work. A Swiss-style machine clearly is the way to go if you turn a lot of these parts.

Part length on a Swiss-style machine is limited only by the length of the bar. But part diameter is restricted to what fits through the spindle—generally, stock less than 1" in diameter.

Also, Swiss-style lathes, especially larger ones, get pretty pricey. They cost up to twice as much as a comparably sized 2-axis lathe. And a lot of shops only handle parts like this occasionally, so they can't justify purchasing such a machine.

If your shop doesn't have a Swiss-type lathe, what do you do? Well, you could always ask the Einstein who accepted the purchase order how the heck you're supposed to make the part! But jobs are a little scarce right now, so you might want to take a deep breath, relax and start thinking with the right side of your brain.

Sometimes the best solution to a difficult machining problem is found by doing a little creative thinking.

Alternative Choices

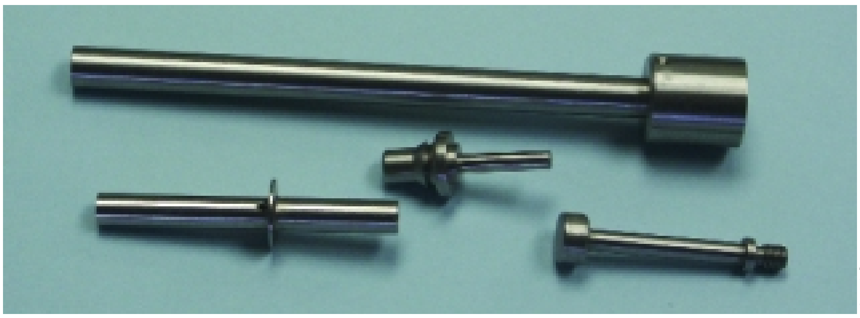
I've tried a number ways to control chatter when machining long, thin parts. The simplest method involved roughing the part first then reaching into the machine to support the workpiece with my finger during the finishing pass. Looking back, that seems pretty foolish. Your shop's safety director definitely will not like this approach.

Another practice likely to prove unpopular with the safety guys is to first notch the end of a long piece of Delrin (a stiff plastic) or wooden round stock, such as a broom handle. Then, while making the finishing pass, position the Delrin on the workpiece, opposite the cutting tool, and apply pressure. This is not the most high-tech solution, either.

However, you won't need to consider either of these suggestions if your shop operates certain types of equipment. Many CNC lathes are equipped with a tailstock, either programmable or manual. If the part can accommodate a live center, you won't have any problems with chatter or tapering.

But not all workpieces can be center-drilled. Many times I've encountered long parts that didn't allow for a center on the print. Sometimes a call to the customer's engineering department is all it takes to get clearance to center-drill a part. Sometimes a call won't work.

So, let's assume that you can't get clearance, or that the part is so small as to make a live center impractical. What



A. Richter

Troublemakers: Parts whose length-to-diameter ratio approaches 3:1 cause machinists all kinds of problems, such as unwanted part tapering and tool chatter.

then? One high-tech solution would be to use a programmable, self-centering work support (Figure 2). It follows the cutting tool, supporting the workpiece while it's being turned (kind of like a notched broomstick).

These devices mount to the machine tool bed and are activated with either an M-code or a macro routine. Several brands are available. But be aware that programmable supports can cost several thousand dollars and require installation by a qualified machine technician.

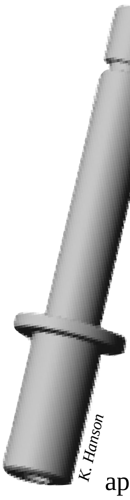

K. Hanson

Figure 1: How would you turn this shaft? It's made from 8620 steel, measures 3½" in length and ⅜" in diameter, features a groove and requires some internal machining.

Despite being great for many applications, programmable work supports do have limitations. One is that unless you have a twin-turret, 4-axis machine, the work support must be mounted in a fixed position, with reference to the workpiece. (Such a machine allows the work support to be mounted to the lower—programmable—turret.) Consequently, this means you can't *follow* the tool with the work support. Also, programmable supports are impractical for parts with diameters smaller than 3⁄16".

A more practical solution for this workpiece would be to wander over to the screw-machine department and borrow a box tool (Figure 3). If you're not familiar with them, now might be a good time to acquaint yourself with these nifty little gadgets.

You say your shop doesn't have any screw machines? No problem. Just ask your local machine tool distributor to send over a screw-machine tooling catalog. Better yet, get on the Internet and search for screw-machine tooling suppliers, such as Boyar-Schultz SMT (www.boyar-schultzsmt.com). You will find a wide variety of tooling that you probably never knew existed, including the box tool.

If you decide to buy a box tool, make sure you pick one that accepts inserts. Some of the older models require that the user grind a tool bit from a piece of ¼"-square HSS and painstakingly eyeball it onto the centerline. Also, buy the largest box tool your machine will accept, as it will offer the greatest capacity and rigidity. Finally, don't forget to order a few extra sets of rollers.

Once you have your shiny new box tool, you'll need to mount it in the turret of your CNC lathe and set it to cut the desired diameter. With the machine spindle off, hand-wheel the box tool up to the face of the part. Then, just barely loosen the nut that holds the box tool's sliding mechanism in place and turn the adjusting screw in or out until the tip of the cutting tool is at the correct (approximate) diameter. (A 6" scale would come in handy in this situation.) Next, tighten the nut, fire up the spindle and take a trial cut. Then measure the part. Repeat the process as necessary.

With any luck, you will be up and running after just a few trial cuts and a turn or two of the adjusting screw.

Once you have the correct size dialed in, it's important to remember that any necessary part-size adjustments cannot be accomplished with a machine offset. Treat your box tool as you would any center-cutting tool, such as a drill or reamer. Once you've set the geometry offset for the centerline of the tool, leave it. And don't use wear offsets to adjust for tool wear. Instead, use the adjusting screw on the box tool.

After setting the box tool, you'll need to position the rollers. These support the workpiece during the operation. The rollers are slightly tapered on one end. Make sure the tapered end faces toward the workpiece. Then slide the rollers into place, snug up the screws holding the rollers in place and lightly tap the rollers until they make gentle contact with the raw stock.

Box tools are limited to a finished-part diameter of about ½" or smaller. For larger diameter parts, a better alternative might be balance turning.

Balance turning can be a little "artsy-craftsy," as they say in Minnesota, where I'm originally from. And while it's difficult to provide any concrete rules for setting up a balance-turning operation, once you understand the basic concept, you may find other applications for this unique way to quickly remove large amounts of material.

There are a couple of ways to balance-turn (Figure 4). The first is possible only if you have a gang-style or chucker-type CNC lathe. Start by mounting a turning tool on the slide of your gang-style CNC lathe. Wheel the tool up to the face of the part so that the tip is sitting about 0.005" to 0.010" away from the face of the workpiece and as close to the finished diameter as possible. This is your finishing tool.

Mount another turning tool upside down and opposite the finishing tool. This is the rougher. Slide the rougher into position so that it is right up against the face of the workpiece, but at a distance from the finish tool that is approximately equal to the finish diameter plus half the total amount of material being removed from the part.

Confused? Let's say we're going to turn our roller shaft from ¾"-dia. stock.

Set the finishing tool at a position to cut a ⅜" diameter—the finished-part size. You only have to get close—say, within 0.050"—because you can use an X-axis offset if you miss by a little bit. Then place the roughing tool in position to cut a diameter of approximately

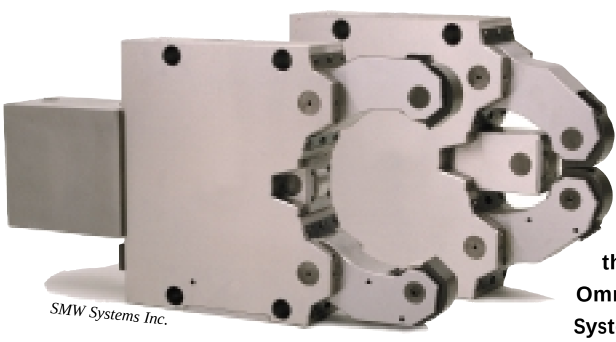

SMW Systems Inc.

Figure 2: A programmable, self-centering work support is a high-tech solution to supporting long, thin parts. Shown is the OmniRest from SMW Systems.

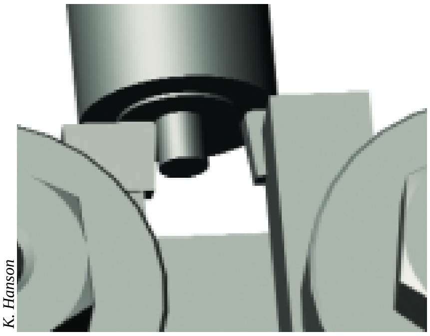
Figure 3: A Boyar-Schultz SMT box tool fitted with a carbide insert.

9/16" [0.375" + (0.375"/2) = 0.5625"]. Set the rougher just slightly (0.005" to 0.010") in front of the finishing tool. Program the machine to take one pass.

The roughing tool will cut just in front of the finishing tool, removing the outer 3/16" of material and leaving a diameter of 9/16". The finishing tool will remove the inner 3/16" of the material directly behind it, achieving finish size in the process.

Under the right circumstances, you can remove a large amount of material in a short time and turn a loser job into a real moneymaker. But this approach won't always be viable. For example, it might not give the required surface finish, because a healthy amount of stock is removed on what is, essentially, a finishing pass.

You can also balance-turn on a turret lathe, but first you'll have to buy a special holder, similar to the box tool. It accepts two opposing cutting tools. You may have to hand-grind your own tool



K. Hanson

Figure 4: Balance turning involves making a single pass with roughing and finishing tools.

bits, though, as I've yet to find a balanced toolholder that accepts inserts.

But, grinding tools is what machinists have to do sometimes, right? Along with a little creative thinking.

About the Author

Kip Hanson is a regular contributor to CUTTING TOOL ENGINEERING. He has spent more than 20 years in the metal-cutting industry and currently works as a manufacturing consultant based in Gold Canyon, Ariz. You can reach him via e-mail (khanson@canyontechnical.com).