APRIL 2003 / VOLUME 55 / NUMBER 4

▶ BY IAN WALLACE & STAN ROBINSON

Machine Your Own

A step-by-step approach to machining collets and jaws for lathe workholders.

It's sometimes necessary to hold workpieces with "soft" collets or chuck jaws. These are workholding devices that have been machined to hold a specific part.

Turning thin-wall parts is an example of when these collets and jaws are used. Because they conform more closely to the workpiece's shape than standard jaws or collets, they grip more of the part's surface area. This, in turn, minimizes the chance of the workholder crushing or distorting the part.

All images: I. Wallace, S. Robinson

A soft collet, set with pins, prior to being bored.

The following machining methods can be applied, for the most part, to either soft jaws or soft collets. (The latter are also known as "emergency" collets.)

First, select the jaws providing the best "fit" for the workpiece and mount the jaws in the chuck. If you select a set of jaws used previously, check to see that they are individually identified with a number, as it is standard practice to number each jaw to correspond with numbers on the chuck. This ensures that each jaw can be returned to the location where it was originally mounted and machined and, in turn, ensures the highest degree of accuracy.

If the jaws have been previously machined but not numbered, it is a relatively simple matter to verify the jaws' original locations by performing a check with a dial indicator. After mounting the jaws in the chuck, run an indicator on the machined surfaces of each jaw to quickly determine if any jaw is out of location.

When satisfied the jaws are in their proper locations, firmly tighten the locking mechanism. It is worthwhile to note that once you have performed this function and established the proper location for each jaw, you should stamp the corresponding numbers from the chuck onto each jaw and save yourself having to perform this exercise again.

It is not necessary to have actual numbers to identify the jaws. With a center punch, simply stamp one, two or three dots, depending on the jaw's location in the chuck. These dots should be placed on the end of the jaw so they are visible when mounted in the chuck.

Do not stamp these identification marks on the jaws while they are mounted in the chuck.

After mounting the jaws, verify that the closing pressure is adequate for gripping the part but not so great as to crush thin-wall sections.

Turning or Boring

If you plan to machine a part that's gripped on the outside diameter, you will need to bore the jaws. If machining a part gripped on the inside diameter, you will need to turn the jaws.

Either way, it will be necessary to close the jaws to machine the hold and location surfaces. To do this, locate a piece of material that fits into the jaws and requires a minimal amount of jaw travel to be held when activated. If you cannot find a properly sized piece, machine one to size. (Note: Most collets come with pins for this purpose.)

Prior to boring the collet, you must determine where to locate the workpiece. Ask yourself questions such as, "Do I need a location face to be turned?" and "Will a location stop at-

tached to the collet suffice?"

Select a suitably sized boring bar to bore the collet or jaws. The bar should be of adequate size to enter the bore diameter and sufficiently rigid to avoid flexing or vibrating. The bar and cutting insert applied should also allow you to machine a slight undercut at the shoulder of the bore so that when the part is inserted into the jaw it sits squarely against the machined face without any chance of interference (Figure 1).

In cases where a lot of metal must be removed to machine the jaws, predrill the jaws close to the finished size. This saves time and eliminates potential tooling problems.

Next, calculate the depth of bore required and locate the boring bar in a convenient tool station. Always keep the tool extension as short as possible to prevent deflection and vibration when boring. This is normally the depth of bore plus a small safety margin to allow the chips to exit the bore while providing maximum coolant access.

Always check that the cutting tool is set exactly on center height. Machine tool builders provide this center-height dimension for cutting tools in the operator's and programming manuals. The center height is a fixed dimension, so once you establish this value don't change it. As the name implies, the cutting tool center height is set relative to the spindle's centerline. This is irrespective of the type of machine, be it a flat- or slant-bed lathe, a turning center or a boring machine. This is the optimum height for most cutters and can be viewed as a line running parallel with the X-axis at a height that takes it through the center of the main spindle.

Many machine tool builders provide a setting block to simplify this task. If you do not have one, determine the center height by manually turning the face of a piece of material and viewing the result. If the part is cut cleanly to the center, your tool height setting is acceptable. If there is an obvious sign that it is not cut cleanly to center, adjust the tool height accordingly and take a second cut. This may have to be repeated a third time, depending on how accurately you adjusted the height after the first cut.

Once the tool is set, bring it into position to take a first cut and note the values in the X and Z axes. The machine may allow the viewing of this position in incremental values. When this is the case, you may also have the option of setting the X- and Z-axis values to zero.

Assuming you have this option, set the positions to zero. This simplifies monitoring the amount of axis motion while machining, relative to the original zero points.

Start the spindle, then check to see that the rotation is in the proper direction for the tooling being utilized and verify the spindle speed setting for the cutting conditions. This includes consideration of material, the cutting tool being applied, tool extension and cutting condition (e.g., an interrupted cut).

In Action

As an example, let's say it's been determined that you should bore to a location face at a depth of 0.500". Proceed with your cut, feeding to a depth of 0.495". Leave the tool in this X-axis position, move the tool so that it's clear of the jaw or collet in the Z-axis and then stop the spindle.

Measure the bore diameter and calculate how far the tool has left to move to achieve the desired diameter. Repeat this procedure, adjusting the X-axis with the appropriate DOCs until you are within 0.005" of the required diameter. You are now ready to take the finish pass.

Set and verify the X-axis position required for taking the final pass, and feed in the Z-axis to the 0.495" depth previously achieved.

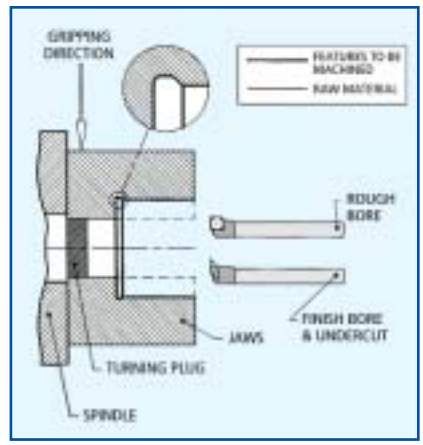
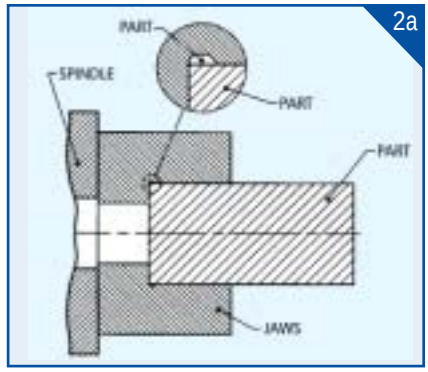


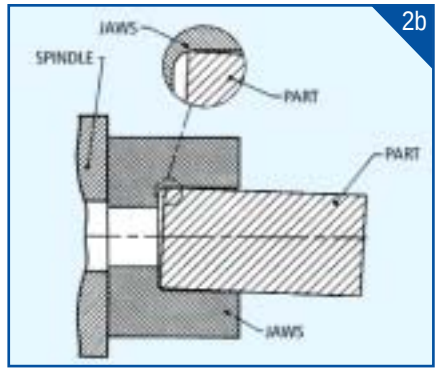
Figure 1: The boring bar chosen to bore a collet or jaws should be of adequate size to enter the bore diameter and rigid enough not to flex.

This time, when you reach the required depth, feed the tool in the X-axis to a diameter "larger" than the bore being produced. This amount should equal, approximately, the nose radius of the boring tool. By doing this you are creating an undercut at the intersection point of the bore and location face at the 0.500" depth. Be sure to note if your adjustment is on the radius or the diameter.

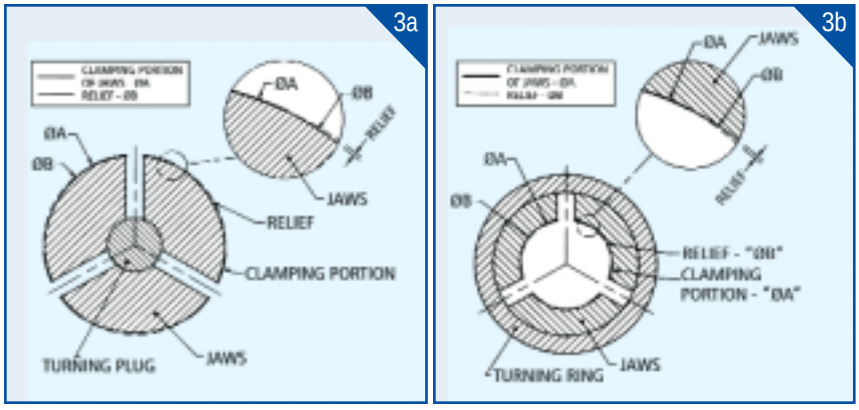
Once you reach this undercut diameter (the desired bore diameter plus 2× the nose radius), again feed the Z-axis slowly to the final depth of 0.500" and stop.

After satisfying both the finish diameter, undercut and bore depth, feed on the X-axis toward the spindle centerline until the tool clears the locating face. Be sure the back of the tool does not hit the opposite side of the turned diameter. Verify by measuring the diameter and





Figures 2a, 2b: When verifying a part has the proper grip and location for the boring process, assure that your undercut allows the part to sit firmly and squarely on the location face and is free of burrs. In Figure 2a, note the full contact on the face and the diameter. In Figure 2b, note how the part rides off the radius bored into the jaws.



Figures 3a, 3b: To grip the OD of a piece of thin-wall tubing, avoid gripping with the crown portion of the jaws. To accomplish this, it will be necessary to machine the finished jaw diameter required to "match" the workpiece's OD.

bore depth of the collet or jaws.

When satisfied, open the jaws or collet and remove the turning ring or pins. The machined area of the jaw must have burrs removed to prevent interference with the workpieces to be machined.

You should then load one of the workpieces to ensure a proper grip and location for the process. Take a close look to verify that your undercut allows the workpiece to sit firmly and squarely on the location face (Figures 2a, 2b).

Whenever possible, run a dial indicator on a part that is known to be accurate to make sure the part being machined is running true and square to the centerline.

As you become more proficient at machining workholding jaws and collets, you will most likely want to create a simple program for this purpose. Until you are fully comfortable with performing this operation, though, perform it manually to learn by "touch."

Final Considerations

When turning the holding diameter of the jaws or collet, it is important to maintain the accuracy of the gripping diameter. If you turn this diameter "oversize," you reduce the available contact area on the part being gripped. This can result in having only a "line" contact between each of the jaws and the part, when there should be full contact around the circumference of each of the jaws and workpiece. Line contact reduces the holding capability and allows the part to move while being machined, possibly distorting the part or breaking the tool.

For specific applications, you can manipulate the turned diameter and contact areas to accommodate special part configurations or materials. For example, to grip the OD of a piece of thin-wall tubing, avoid gripping with the crown portion of the jaws. To accomplish this, it will be necessary to machine the finished jaw diameter required to "match" the workpiece's OD. Once completed, select a turning ring that is slightly smaller than the first and take another finishing pass over the machined section, removing only a very small amount of material. This, in effect, removes the crown of each jaw and leaves the original gripping area on the periphery (Figures 3a, 3b).

CNC lathe machinists must learn different ways to hold and locate workpieces to suit a multitude of part configurations and cutting conditions. This requires experience, and there is only one way to get it. Knowledge, however, requires asking questions and gaining insight from the experience of others.

About the Authors

Ian Wallace is production team leader and Stan Robinson is plant manager at B.C. Instruments, a precision machine shop in Schomberg, Ontario. This article was excerpted from their book, "Mastering C.N.C. Lathe Set-Up: A Guide to Becoming an Effective C.N.C. Lathe Machinist." *For additional information, call or fax (905) 985-4995, or visit www.cnc.bizhosting.com.*