CNC CONCEPTS, INC.

The Optional Stop

Spring 2003 Issue 55

Tips For Fanuc Control Users From CNC Concepts, Inc.

44 Little Cahill Road Cary, IL 60013 Ph: (847) 639-8847 FAX: (847) 639-8857

Are you taking full advantage of turning center offsets?

Current model Fanuc-controlled CNC turning centers actually have three types of compensation (offsets), geometry offsets, work shift offset, and wear offsets. Geometry offsets are used to assign program zero. The work shift offset can be used to shift the program zero point for all tools. Wear offsets are commonly used for sizing and to allow the operator to hold size on workpieces as tools wear.

Unfortunately, many companies don't take full advantage of what is possible with these three important turning center features. The full implications of how they can be applied aren't related in most basic CNC turning center courses. For this reason, many turning center using companies confuse or misapply the usage of these offsets. In this lengthy article, we're going to fully examine the use of turning center offsets, including how program zero can be efficiently assigned and how work can be minimized from setup to setup and when tools are replaced during a production run.

For the most part, this article assumes you are not using a tool touch-off probe to help with program zero assignments. While many of the points we make do apply nicely to tool touch-off probe users, the largest potential gains from applying techniques we show will be for companies that don't use them.

Where is program zero?

In the X axis, program zero is always placed in the center of the workpiece (also spindle center). But for the Z axis, the programmer can choose a program zero point position that matches how dimensions are specified on the drawing. This lets them use print dimensions as programmed coordinates. Since most turned workpieces are dimensioned from the end of the workpiece being machined, most turning center programmers make the program zero point the extreme end of the workpiece in the Z axis.

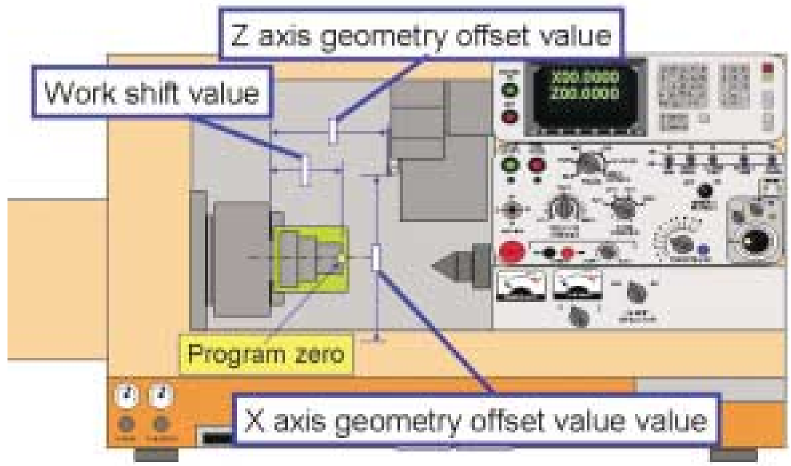


Figure one

Assigning program zero for each tool

Each tool used on a turning center is in a different location. A turning tool's cutting edge, for example, is in a completely different location that a boring bar's cutting edge – or a drill's. For this reason, each tool requires its own program zero assignment. In essence, the setup person must align the setup that's been made to the program that's been written. Geometry offsets are used for this purpose. Two program zero assignment values (one for X and one for Z) specify the distances from the tool tip at the zero return position to the program zero point (X is specified in diameter).

The measure function

Most controls that have geometry offsets allow the setup person to easily determine and enter each geome-

In This Issue:

Subscription Information:

The Optional Stop is published quarterly by CNC Concepts, Inc. and is distributed free of charge to people downloading it from our website (www.cncci.com). Back issues are available, but a $10.00 charge per issue will apply to any back isues not currently published on our site. Back issues 1-50 are available for a total of $199.00.

try offset's values. While the specific procedure varies among Fanuc control models, here is one common scenario. For the X axis, the setup person makes the cutting tool machine a small amount of material from a diameter. They then back the tool off in Z (not moving X) and measure the diameter just machined. They bring the cursor to the geometry offset to be set (the tool station number) and type MX (for measure X) along with the diameter they just measured. When they press the INPUT key, the control will automatically calculate the tool tip's diameter at the zero return position in X and enter it into the geometry offset's X register as a large negative value.

The procedure is much the same for the Z axis program zero assignment. The setup person first brings the tool to the Z axis program zero surface (possibly skim cutting). Next, they bring the cursor to the geometry offset to be set. They type MZ0 and press the INPUT key. The control automatically calculates the distance from the tool tip at the zero return position in Z to the Z axis program zero surface and enters it into the geometry offset as a large negative value.

While the measure function dramatically simplifies the program zero assignment task, program zero must be assigned for every tool in the turret. And if Z axis geometry offsets program represent the distance from zero return to the face of the finished workpiece, program zero assignment must be repeated in Z for all tools when a new setup must be made since the workpiece for the next job will be of a different length.

Eliminate all geometry offset tasks for tools that remain in the turret from job to job.

The scenario just described works well if all tools will be removed from the turret from job to job. But must turning center users leave a "core set" of tools in the turret on a semi-permanent basis. Tool number one, for example, may be the rough turning tool. Tool number two may be the finish turning tool. Tool number three may be a threading tool. And so on. For tools that remain in the turret from job to job, there will be a great deal of duplicated effort. While the X axis geometry offsets will remain consistent from job to job, the setup person must repeat the Z axis program zero assignment value measurements and geometry offset entries.

Your goal will be to eliminate all geometry offset measuring and entry tasks for any tools that remain in the turret for the next job. In fact, for any tool that remains in the turret (even if it is not used in the next job), its related geometry offsets will remain correct until the tool is removed from the turret!

This can be easily accomplished by re-thinking the reference position for your geometry offset entry. Instead of working from the workpiece face (a location that changes from setup to setup), choose to work from a more consistent Z axis surface. Since most companies do not remove the chuck from the machine from setup to setup, the face of the chuck makes an excellent point of reference for Z axis geometry offset entries.

Note, however, that many turning centers don't have enough Z axis stroke to let a turning tool reach the face of the chuck. If this is the case, simply use a gauge block (like a 1-2-3 block) between the tool tip and the face of the chuck when determining Z axis geometry offset values. After touching the tool tip to the three inch side of a 1-2-3 block (which is flush with the chuck face), type MZ3.0 and press the INPUT key. The geometry offset will be properly calculated and entered.

The work shift value

The programmer will, of course, still want program zero to be the workpiece face to keep programs simple (and to keep from having to modify all programs written to this point). Fanuc controls have a special offset display page called the work shift page. You'll never need to modify the X value on this page (X axis program zero will always be the center of the workpiece – which doesn't change from setup to setup). But in the Z axis register of the work shift page, you'll enter the distance between the chuck face and the true Z axis program zero point on the face of the workpiece. Note that the polarity for this work shift value changes from one control model to another. With some, it is the distance from the chuck face to the program zero point (+). With others, it is the distance from the program zero point to the chuck face (-). A little trial and error may be needed to determine what's needed for your control/s.

Figure one (on page one) is a drawing that shows the program zero point, work shift value, and X & Z geometry values as we have recommended to this point.

Going from job to job

As long as your programmer is consistent, programming the mean value of every tolerance for every job, your setup people will not have to do anything to geometry and wear offsets for tools that were used in the previous job. Truly, if a tool was cutting on size in the last job, it will continue to machine on size in the next job. The only exception to this statement might be dramatic workpiece material variations from job to job. But with material changes, it is likely that cutting tools (inserts) must also be changed between jobs.

Only new tools being placed in the turret for the new job will require geometry offset measurements and entries. And again, for as long as a tool remains in the turret, its geometry and wear offsets will remain correct. By the way, we recommend that setup people get in the habit of clearing (setting to zero) geometry offsets only as they remove tools from the turret. With this method, they can rest assured that geometry and wear offsets for any tool currently in the turret will be correct.

Determining the work shift value for each new job

The work shift value must, of course, be changed from setup to setup. If workholding tooling is not consistent, as is commonly the case with soft jaws, the setup person must physically measure the work shift value. After making the workholding setup, the setup person can measure the work shift value. If it's not critical, as is commonly the case when machining the first operation (lots of stock left on the other end), the setup person may be able to measure the work shift value with a scale.

If it's more critical (second operation and workpiece length is determined by the accuracy of the work shift measurement), more precise measuring will be necessary. While there are many ways to do this, we show one simple example. Say the workpiece must be precisely 6.000 inches long. Yet there is approximately 0.100 inch of stock on the face yet to be removed. After the workholding setup is made, the operator can skim cut the face to clean up (still leaving stock). They will then set the Z axis display to zero. Next, they measure the overall length of the workpiece to determine precisely how much excess stock is still to be removed. We'll say that it's currently 0.054 inch. By using the handwheel and monitoring the Z axis display, they'll move the tool precisely 0.054 inch in the negative direction. The tool tip is now flush with the Z axis program zero point. The setup person will now set the Z axis display to zero again. When they move the tool tip over to the chuck face, the Z axis display will be showing the work shift value.

Programming the work shift value

Note that when you use solid (fixed) top tooling or stopper blocks, the work shift value will be predictable and consistent from one time the job is run to the next. Consider, for example, using hard jaws that incorporate Z axis stop pads. For our example, the stop pads, when the hard jaws are mounted, are precisely 0.75 inch from the chuck face. The work shift value will be 0.75 plus the rough stock on the chuck side of the workpiece plus the workpiece length. If you know the work shift value you can program it. This will keep the setup person from having to measure and enter it. And by the way, for companies that use the same cutting tools in every job, this will help you to eliminate all program zero assignment tasks from job to job. A G10 command is used to enter work shift values. The command

```
G10 P0 Z6.75
```

will place the value 6.75 in the Z axis work shift register.

Sizing the first workpiece

To this point, we have been discussing tasks that get you ready to machine the first workpiece. With the setup made and the geometry offsets determined and entered, let's now turn our attention to the best way to size in a new tool that's just been placed in the turret. Experienced setup people know that just because they correctly measured and entered geometry offsets, doesn't mean a tool is going to machine perfectly on the very first try. Even if skim cutting was done to determine geometry offset values, there will likely be some tool pressure differences when the tool actually machines the workpiece using the cutting conditions specified in the program. Here is a simple scenario:

A rough turning tool has machined a 3.000 inch outside diameter, leaving 0.030 stock on the side. Currently this diameter is 3.060 inch. The finish turning tool comes in and machines the 3.000 inch diameter. But after measuring, the setup person finds that the 3.000 diameter is actually 3.0012 inches, meaning it is not perfectly to its targeted size. A small discrepancy exists. But this discrepancy may be enough to through the workpiece out of tolerance – or even if the workpiece is within its tolerance band, the setup person will want to bring this dimension to its target value before they allow the production run to begin. What does the setup person do?

Most setup people will make the needed adjustment in the X axis wear offset register, reducing it in this case by 0.0012 inch. The next time the tool cuts, it will machine 0.0012 inch smaller, cutting the 3.000 inch diameter to its target (mean) value. While this works just fine, consider what will happen during the cutting tool's life. As the tool continues to machine workpieces, it will wear. After 50 workpieces, for example, the 3.000 inch diameter may be 3.0008. While it's still within its tolerance band, this diameter may be getting close to its high limit, and the operator will eventually want to make a sizing adjustment. So they reduce the wear offset by another 0.0008 inch (now -0.002 inch in the offset).

This pattern of tool wear will be repeated several times during the tool's life. After several sizing adjustments, the total wear offset is now -0.0043 – and the tool is dull and in need of replacement or indexing. Say they're indexing the insert (not replacing it). When the operator indexes the insert, the wear offset must also be adjusted. If the operator forgets to do this, the next part will be machined undersize (scrapped). But to what value must the wear offset be set? In our case, the wear offset started out at -0.0012 inch. And this is a value the operator must remember if they are to have any hope of machining the next workpiece on size without trial machining.

Make the initial adjustment in the geometry offset!

What caused the initial 0.0012 variation? Did it have anything to do with tool wear? Obviously, the answer is no. The initial variation was caused by an imperfection with program zero assignment. And the initial offset adjustment should be done in the geometry offset (reduce the geometry offset by 0.0012). With this technique, the geometry offset will be perfect, and the wear offset can start off at zero. Any value in a wear offset will be related to tool wear. When an insert must be indexed, it's related wear offset can simply be set back to zero and the next workpiece can be run without trial machining.

What about insert inconsistencies?

Again, we've been talking about indexing inserts. Since the same insert is being used, there should be no inconsistencies related to insert size. However, all inserts have a tolerance. It is specified by the third letter of the insert's designation. With a CNMG-432 eighty degree diamond-shaped insert, for example, the letter M designates the insert's tolerance. M specifies that this insert has a plus or minus 0.002 inch tolerance on the insert's included circle, which will have a dramatic impact on the inserts placement in its tool holder (almost a 0.004 inch variation). For most turning applications, this variation alone will be enough to through a workpiece out of tolerance.

Note first of all that you can buy more repeatable inserts. A CNBG-432, for example has only a plus/minus 0.0002 tolerance on its included circle. While it's more expensive, its additional price may be justifiable if you can eliminate trial machining when inserts are replaced.

If you cannot justify buying more repeatable inserts, you can still take advantage of the technique discussed earlier whenever your index inserts. Since most inserts have at least four cutting edges, this means you'll only have to trial machine once in every four times you perform tool maintenance. Just remember, every time you replace an insert (possibly requiring trial machining), make the initial adjustment in the geometry offset. By the way, when a geometry offset is entered or modified, its corresponding wear offset will be automatically set to zero on most controls. So again, the geometry offset will be perfect for the new insert and the wear offset will start out at zero.

What about operator inconsistencies?

Admittedly, operators will have to be consistent in the method by which they change/index inserts. Cam clamps must always be tightened in the same direction so the insert seats against the same tool holder surface. And clamping pressure must be consistent. But with a little practice (and testing) operators should be able to (especially) index inserts in a manner consistent enough to eliminate the need for trial machining when inserts are indexed.

What about small lots?

If your cutting tools last for the entire production run without having to be indexed or replaced, how the initial adjustment is made will have no impact upon machine usage. In this case, most companies will perform the initial adjustment in the wear offset.

More on cutting tools used in the last job

Since machine time is so very precious in most companies, you should strive to eliminate any duplicated effort. By using the techniques described in this article, you should be able to go from job to job with as little duplication of effort as possible. New work will only be required for the tasks related to new cutting tools being placed in the turret for the next job. And of course, the new work shift value must be determined and entered (but again, remember that it can be programmed with fixed locators).

Once again, make sure setup people and operators understand that if cutting tools were cutting properly in the last job, they will continue to cut properly (at least they will machine to size) in the next job. They can move right on to the next job, confident that tools used in the last job will continue to machine on size.

This requires, of course, that programmers program mean values of every tolerance band. If they do not, program discrepancies will require trial machining for critical tools every time a new job is run.

M01

CNC CONCEPTS, INC.
44 Little Cahill Road
Cary, IL 60013
Ph: (847) 639-8847
Fax: (847) 639-8857

The Optional Stop Newsletter Enclosed!

Mailroom: PAID SUBSCRIPTION ENCLOSED. Please assure delivery to addressee. Thank You!

Products That Address The Needs Of CNC Technology Today!

Use this handy form to request more information about our products.

CNC Video Courses:
❑Machining Center Programming And Operation
❑Turning Center Programming And Operation
❑Conversational Turning Center Programming & Op
❑Custom Macro Programming
❑Four Axis Turning Center Programming

CD-rom Courses:
❑Machining Center Programming, Setup, & Operation
❑Turning Center Programming, Setup, & Operation
❑Setup Reduction for CNC
❑Cycle Time Reduction for CNC
❑Parametric Programming
❑CNC Router Programming, Setup, & Operation
❑Advanced Techniques With Basic CNC Features

CNC Mini-Vids:
❑Related To Selected CNC Topics Like Setup Time Reduction, Cycle Time Reduction, Probing

CNC Course Curriculums For Instructors:
❑ Machining Center Programming And Operation
❑ Turning Center Programming And Operation
❑ CNC Setup Reduction for CNC
❑ CNC Router Programming And Operation
❑ Maximizing CNC Utilization (NEW!)

CNC Publications:
❑Machining Center Self Study Course
❑Turning Center Self Study Course
❑CNC Router Self Study Course
❑Managing CNC Operations
❑CNC For Machining
❑CNC Advanced Techniques
❑CNC Accessory Devices
❑The Optional Stop Subscription & Back Issues

CNC In-Plant Training:
❑Basic CNC Courses
❑CNC Tune-Up - For your *experienced* People

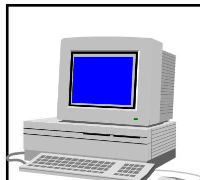
If other than above:

Name: ______________________

Company Name: ______________________

Address: ______________________

City, State, Zip: ______________________



Check Out Our Website!
http://www.cncci.com/
Information about each of
our products is on line!