

Lesson 4.2–Tool Length Compensation

Tool length compensation allows a programmer to ignore the precise length of each tool as a program is written. It is used for every tool in every program you write—so you must understand this important CNC feature.



Objectives

After completing this lesson, students should be able to:

- ✓ Describe the reason tool length compensation is needed
- ✓ Describe how to program tool length compensation
- ✓ Describe how to select which tool compensation offset to use for each tool
- ✓ Describe two methods for measuring tool length offset values
- ✓ Describe the advantages to using the recommended method of measuring tool length offset values
- ✓ Describe the typical mistakes programmers, setup people and operators make with tool length compensation
- ✓ Describe trial machining, how to do it and when it is needed

Introduction

You know that workpiece coordinate system offset values are entered into workpiece offsets. For the X and Y axes, the values are the distances between the workpiece coordinate system zero and the spindle center (while the machine is at its zero return position). So when you specify a position of X1.0 Y1.0 in a program, the machine will be able to send the spindle center (and tool center) to this position—relative to program zero. For the Z-axis the value is the distance between the Z-axis workpiece coordinate system zero surface and the spindle nose (again, while the machine is at its zero return position).

But for cutting tool positioning, you don't want to specify Z-axis positions from the spindle nose. This would be very cumbersome—and it would require that you know the precise length of each tool before you could even write the program. You want to specify Z-axis positions to the tip of each cutting tool so that when you specify a position of Z0.1, the tool tip will move to this position. In order to be able to program the tool tip in the Z-axis, a feature called tool length compensation must be used. Mastering tool length compensation is the focus of this lesson.

The Reasons Why Tool Length Compensation is Needed

Cutting tools used on machining centers differ from one another. For one thing, there are a variety of cutting tool types that are used on machining centers, including center drills, spot drills, drills, taps, reamers, boring bars, end mills, and face mills (among many others). Each type of tool requires a different way of gripping the actual cutting tool in its holder. Some tools (like some straight shank tools) use a collet system. Others (like end mills) use a set-screw to hold the cutting tool in place. Yet others (like face mills and taps) require a very special style of tool holder—designed especially for the cutting tool.

No Two Tools Will Have Exactly the Same Length

Given the vast assortment of cutting tools available for use on CNC machining centers, it is unlikely that any two tools used in a program will be exactly the same length. And you will not know precisely how long each tool will be when you write the program. The figure below shows five different types of cutting tools to illustrate this point.

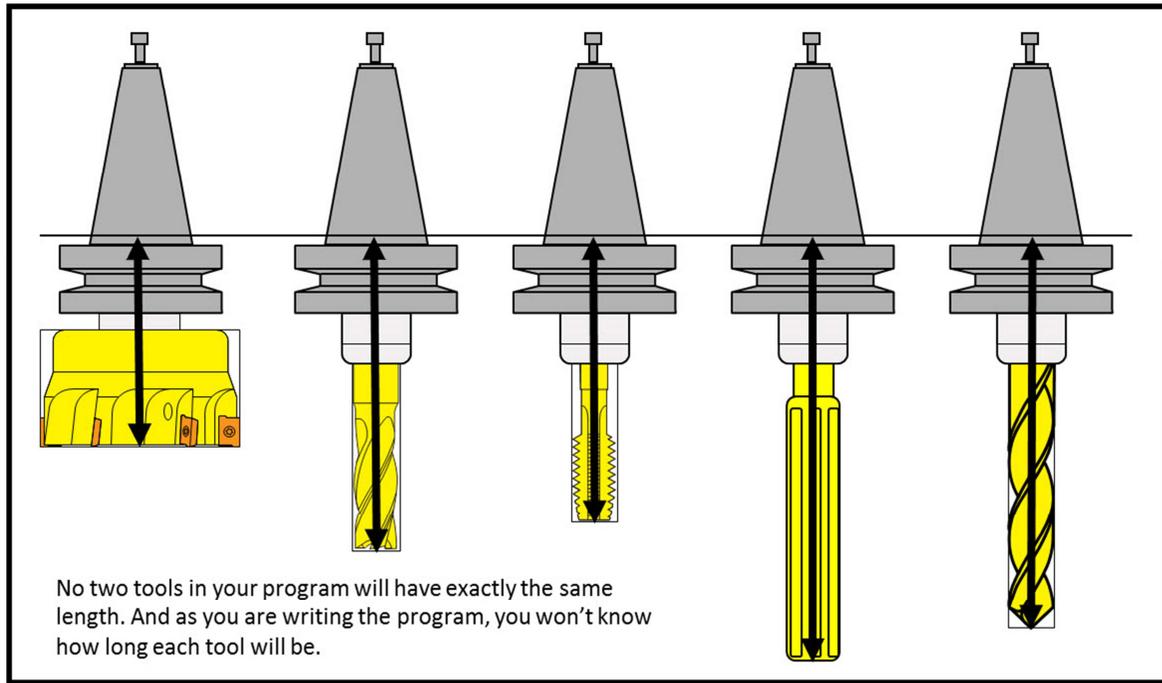


Figure 4.5: Five cutting tools that might be used in by a CNC machining center program

Tool length compensation will allow you to write programs even though you don't know how long the cutting tools will be at production time.

Tool's Length Will Vary from Each Time it is Assembled

When a cutting tool is assembled more than once (even with the same components), its length will usually vary. Consider, for examples, straight shank tools that are placed in collet holders. Each time you assemble the tool, it will be of a different length. Tool length compensation will allow you to use the same program over and over again, even though each tool's length changes from one time the job is run to the next.

Tool Data is Entered Separately from the Program

The same program will work regardless of how long each cutting tool is. The program tells the control where to look for the length of each tool. During setup, the setup person (or someone) assembles and measures each cutting tool. The length of each tool is then placed in the appropriate location (a tool offset register).

Sizing and Trial Machining Must Often be Done

In lesson 4.1, we discuss the importance of being able to trial machine in order to machine the first workpiece correctly. And during a given tool's life a tool will wear and cause the surface being machined to change. Tool length compensation allows the setup person and operator to easily hold size for Z-axis related dimensions (pocket depths, hole-depths, etc.). The program need not be changed when workpiece dimensions must be adjusted.

What about interference and reach?

As is discussed in lesson 2.1, you must be concerned with whether or not your cutting tools will reach the surfaces to be machined—without over-traveling—and without interfering with the fixture of other obstructions on the workholding device. Tool length compensation will not help with these concerns.

Programming Tool Length Compensation

There two popular methods for measuring and setting tool length compensation. As long as you use workpiece coordinate system offsets to assign program zero, the programming remains exactly the same regardless of which method you choose.

Tool length compensation is instated with a G43 word. Included within the G43 command is an H-word that specifies the offset number in which the tool length compensation value is stored. You must also include a Z-word in the G43 command, telling the machine where you want the tool tip to be positioned. This initial move allows the tool offset value to be instated without a negative Z-axis motion.

The G43 command will always be the cutting tool's first Z-axis motion. Said another way, you instate tool length compensation during each tool's first Z-axis motion as the tool approaches the workpiece in the Z-axis.

Once tool length compensation is instated, it remains in effect until the next tool is selected. All Z-axis motions you need to the tool to make will be relative to the tool tip.

Since you will instate tool length compensation during the next tool's first Z-axis motion—using the appropriate offset of course—and since offsets are not accumulative—you need not cancel tool length compensation. G49 cancels tool length compensation, but if you use the techniques shown in this course, you need not use G49 in your programs. Be warned, if you do program G49 in your program it will cause the Z-axis to move by the tool offset amount to uninstall the tool offset.

Choosing the offset number to be used with each tool

Offsets are storage registers for values. Each offset to be used with tool length compensation will contain a tool length compensation value for one tool. To keep from entering the tool length compensation value in the wrong offset, the programmer must use a logical approach for selecting offset numbers.

Keep it simple. Use the offset number that corresponds to the cutting tool's magazine station number. For example, use offset number one for the tool in tool station number one. Use offset number two for the tool in tool station number two. And so on.

This tool length compensation offset will be instated in the program during each tool's first Z-axis motion. An H-word is used to specify the offset number. And again, we recommend that you make the H-word number match the tool station number (the T-word number for each tool).

An example program

The figure below shows the drawing to be used for this example.

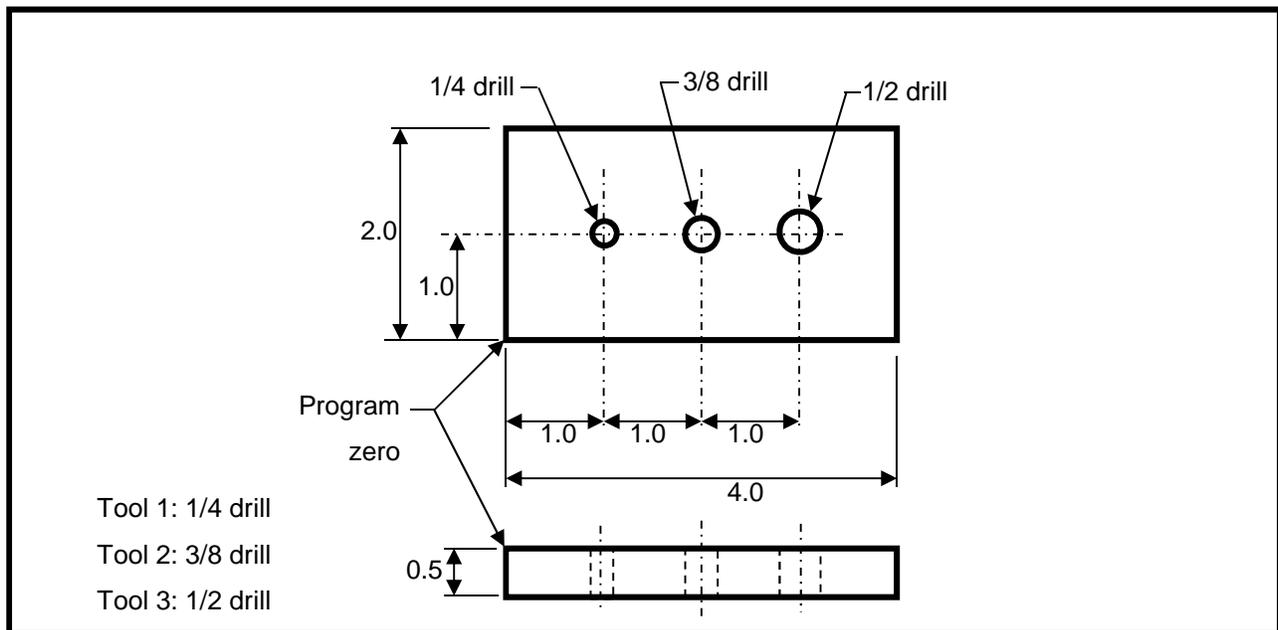


Figure 4.6: Drawing for example program

Program with comments:

```

00003 (Program number)
N010 G20 G90 G54 (Select inch & absolute modes, fixture offset #1)
N020 T01 M06 (Load tool #1 in spindle - 1/4 drill)
N030 S1200 M03 T02 (Start spindle fwd at 1200 rpm, get tool #2 ready)
N040 G00 X1.0 Y1.0 (Rapid to hole location in X and Y)
N050 G43 H01 Z0.1 (Instate tool length compensation for tool one, approach
in Z to just above work surface)
N060 M08 (Turn on the coolant)

```

```

N070 G01 Z-0.65 F4.0 (Drill hole)
N080 G00 Z0.1 M09 (Rapid out of hole, turn off coolant)
N090 G91 G28 Z0 M19 (Rapid to tool change position, orient spindle)
N100 M01 (Optional stop)

N110 T02 M06 (Load tool #2 in spindle - 3/8 drill)
N120 G54 G90 S1000 M03 T03 (Select fixture offset #1, absolute mode, start
  spindle fwd at 1000 RPM, get tool number three ready)
N130 G00 X2.0 Y1.0 (Rapid to hole position in X and Y)
N140 G43 H02 Z0.1 (Instate tool length compensation for tool 2, approach in
  Z to just above work surface)
N150 M08 (Turn on coolant)
N160 G01 Z-0.7 F5.0 (Drill hole)
N170 G00 Z0.1 M09 (Rapid out of hole, turn off coolant)
N180 G91 G28 Z0 M19 (Rapid to tool change position, orient spindle)
N200 M01 (Optional stop)

N210 T03 M06 (Place tool number three in spindle)
(1/2 Drill)
N220 G54 G90 S800 M03 T01 (Select fixture offset #1, absolute mode, start
  spindle fwd at 800 RPM, get tool number one ready)
N230 G00 X3.0 Y1.0 (Rapid to hole in X and Y)
N240 G43 H03 Z0.1 (Instate tool length compensation for tool three, approach
  in Z to just above work surface)
N250 M08 (Turn on coolant)
N260 G01 Z-0.75 F6.0 (Drill hole)
N270 G00 Z0.1 M09 (Rapid out of hole, turn off coolant)
N280 G91 G28 Z0 M19 (Rapid to tool change position, orient spindle)
N290 M30 (End of program)

```

Blocks N050, N140, and N240 instate tool length compensation for each of the three tools. Notice that each of these commands is the first Z-axis movement for the tool (its approach movement in Z). Each instating command includes the G43 word, the appropriate H-word (that matches the tool station number that is currently in the spindle), and a Z-word. Once tool length compensation is instated, it will remain in effect until the next tool. Again, it never has to be canceled using this style of programming as the G28 automatically suspends the tool length compensation during the move to the zero return position.

The Setup Person's Responsibilities with Tool Length Compensation

Before this program can be run, the setup person must perform several tasks. For example, that they must mount the workholding device (probably a vise in this example), measure the workpiece coordinate system offset values (techniques shown in lesson 1.5) and enter them into work offsets (lesson 1.6). They must also load the program.

Cutting tools must be assembled and loaded into the machine's automatic tool changer magazine—tool station 1 for the 1/4 drill, tool station 2 for the 3/8 drill, and tool station 3 for the 1/2 drill. Though we haven't shown how yet, tool length compensation values must also be determined and entered into the appropriate offsets. Let's now discuss how tool length compensation values are determined.

There are two popular ways to use tool length compensation. We'll first show our recommended method.

Using the Tool's Length as the Tool Length Compensation (offset) Value

This is our recommended method. The Z-axis workpiece coordinate system offset value (work offset Z-value) is the distance between the Z-axis workpiece coordinate system zero and the spindle nose when it is at the Z-axis zero return position to—just as is shown in lessons 1.5 and 1.6. With our recommended method, the length of each cutting tool will be its tool length compensation value.

The length of the cutting tool is the distance from the tool tip to the spindle nose and will always be a positive value. This will be the value that is entered into the tool's tool length compensation offset register. Figure 4.6 shows a typical tool's length.

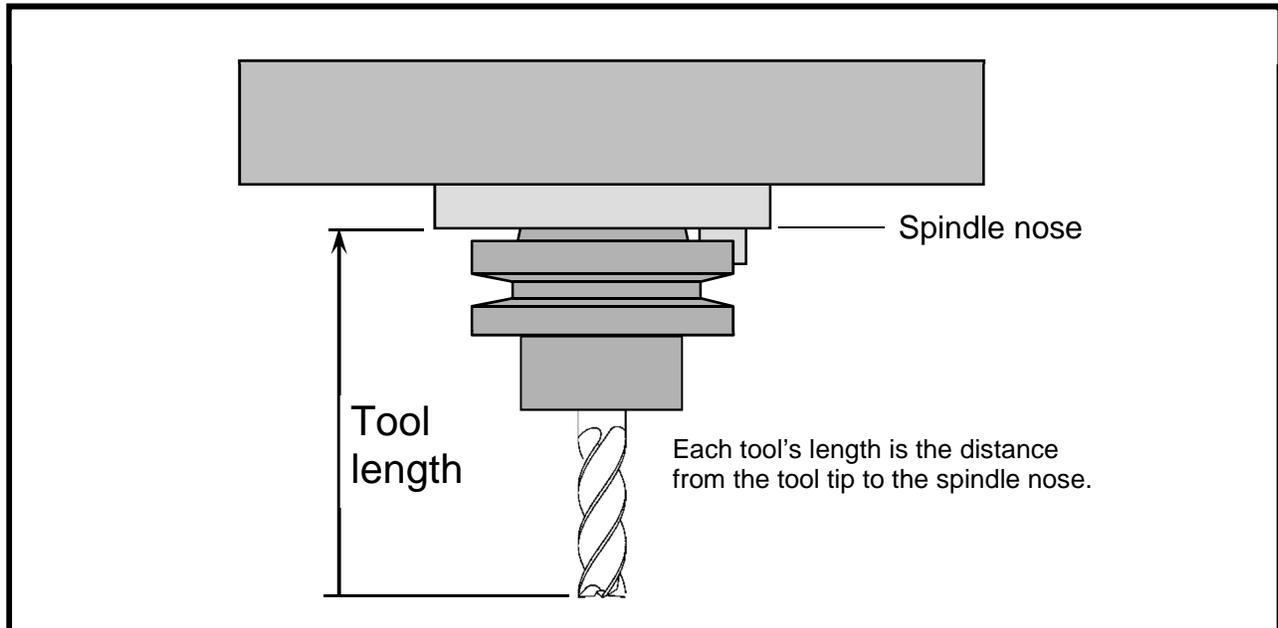


Figure 4.7: Tool length is the distance from the tool tip to the spindle nose—always a positive value

Most machining centers use tool holders that have a tapered shank (CAT-40, CAT-50, BT-40, and BT-50 tool holders are very common examples of tapered shank tool holders). Figure 4.5 shows some examples. The shank taper matches the taper in the spindle itself—and the tool locates in the spindle against this taper. From the illustration in Figure 4.7, notice that there is a small gap between the spindle nose and the flange of the tool holder. This gap is about 0.125 inch.

We're pointing this out because you might incorrectly assume that tool length is the distance from the tool tip to the end of the flange. When holding a tool holder in your hands, this might appear to be logical. But again, the tool's length is the distance from the tool tip to the spindle nose, not to the end of the flange.

Determining tool length compensation values

The tool length for each tool can be measured right on the machine during setup, or it can be measured off-line using a tool length measuring device. Measuring on the machine during setup takes time—and companies that are highly concerned with reducing setup time prefer to measure cutting tools off line. Let's look at both methods.

Even if your company does measure tool lengths off-line, there will be times when you must still measure tool lengths on the machine (maybe after a dull tool is replaced)—so it is quite important that all setup people and operators know how to measure tool lengths on the machine. Techniques to do so are quite similar to those used to measure program zero assignment values (shown in lesson 1.5).

Step 1: Make the workholding setup—or place some kind of flat block on the machine table

It is important to have a nice, flat surface on which to work. The top of a vise works nicely.

Step 2: Place a gauge block on the flat surface

The three inch side of a 1-2-3 block works nicely. However, low cost dial and electronic tool touch-off gauges are also available.

Step 3: Without a tool in the spindle, make the spindle nose touch the block or offset gauge.

Use incremental jog and/or a handwheel to carefully touch the spindle nose to the gauge. At this point, reset (set to zero) the Z-axis relative position display—select the relative display screen, press the Z letter address and then the [ORIGIN] soft key.

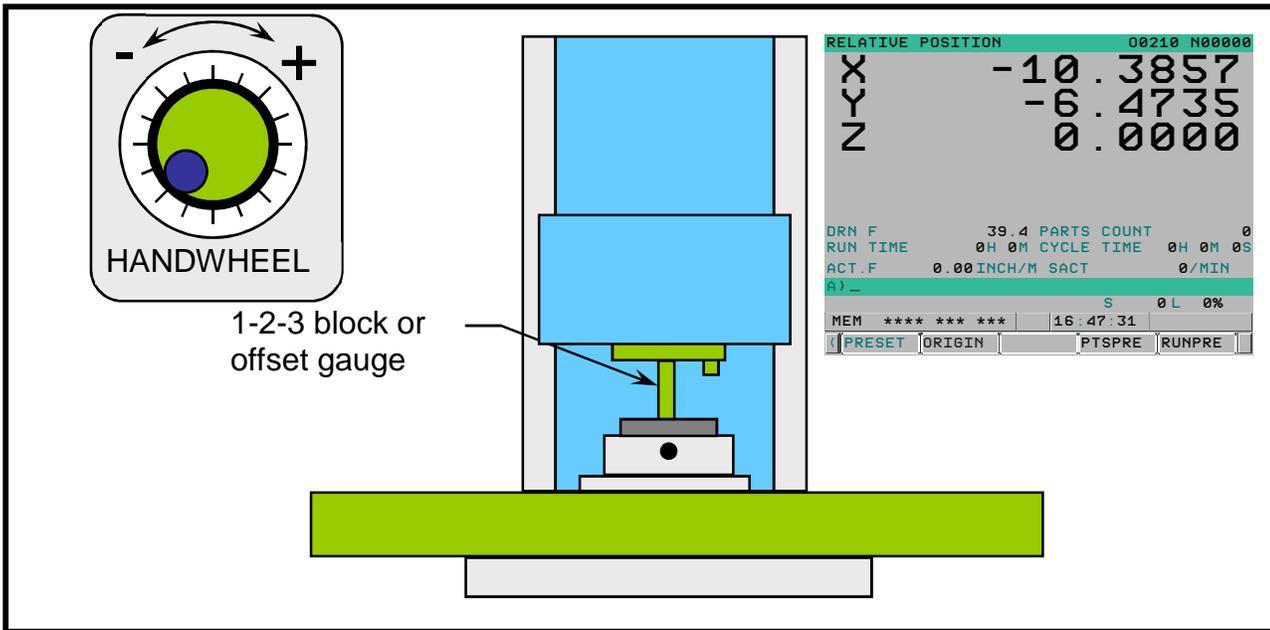


Figure 4.8: Spindle nose touching gauge block or offset gauge

Step 4: Retract the Z-axis and load a cutting tool to be measured. Bring the tool tip to the same block.

Use continuous jog and/or the handwheel to cautiously move the spindle nose away from the block. Load a cutting tool to be measured. Now, carefully move the tool tip to the same block that was just touching the spindle nose. The Z-axis relative display will follow along. When the tool tip is touching the block, the Z-axis display will be showing you the tool's length. This is the value that must be entered into the tool length compensation offset register for this tool. Enter the value into the appropriate tool length offset register.

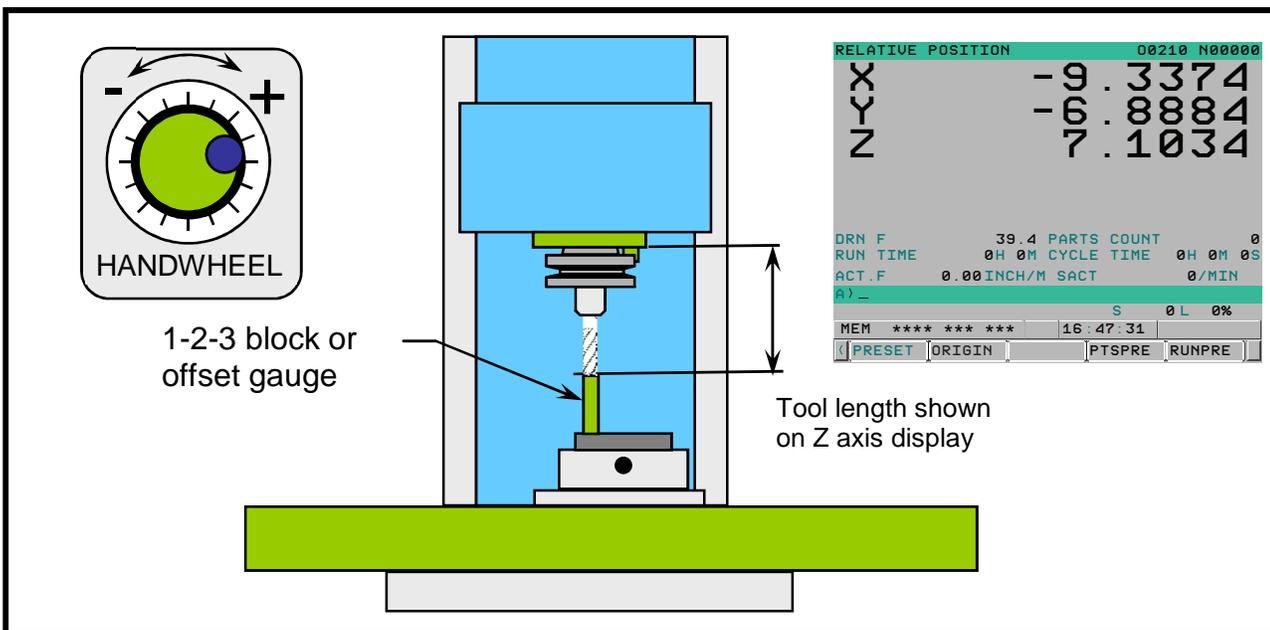


Figure 4.9: Measured tool touching gauge block or offset gauge

Step 5: Repeat step four for all cutting tools that must be measured.

Measuring tool lengths off line—with a tool length measuring gauge

There are many suppliers that can provide devices specially designed for tool length measuring. These devices make it quite easy to measure tool lengths on a bench—away from the CNC machining center. And of course, the machine can be running production while a person assembles cutting tools and measures tool lengths in preparation for up-coming jobs. In this manner, downtime between production runs can be reduced.

But you don't have to buy a fancy tool length measuring gauge. With a little effort, just about any height gauge can be used. The figure below shows an example.

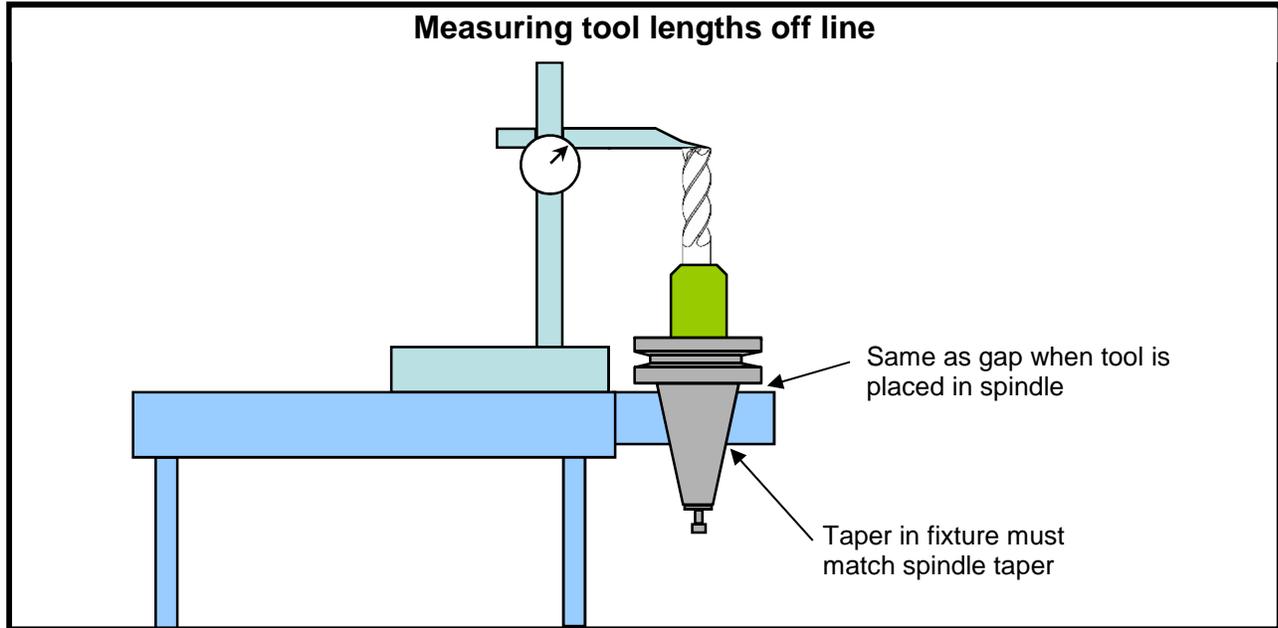


Figure 4.10: A simple and inexpensive height gauge can be used to measure tool lengths.

Entering tool length compensation offsets

Regardless of which method is used to determine tool length values (measuring on the machine or off-line), the tool length compensation values must be entered into tool offsets before the program can be run. One benefit of measuring on the machine is that most machining centers provide a way to transfer the Z-axis position display value right to the tool length compensation offset. This eliminates the possibility for data entry mistakes.

How accurate are your tool length measurements?

The accuracy of your tool length measurements directly affects the accuracy of the machining operation. While some depth dimensions have wide open (large) tolerances, others have very tight (small) tolerances.

For tight tolerances (under about 0.002 inch or so), it is likely that you will not be able to measure tool lengths accurately enough. In this case, if no special consideration is given to how the program is run, the dimension machined by the tool will probably *not* be within its tolerance band the very first time the tool cuts.

Once the tool has machined, the workpiece can be measured. You can then determine *how much* to adjust the tool length offset in order to machine the *next* workpiece correctly. But you will have probably scrapped a workpiece. The technique **trial machining** (shown later in this lesson) will ensure that the cutting tool will machine accurately on the first workpiece.

Using the Distance from the Tool Tip to Workpiece Coordinate System Zero as the Tool Length Compensation (offset) Value

While we don't recommend using this method, it is quite popular with vertical machining centers (it is difficult to apply this method to horizontal machining centers when rotary devices are used). If your company uses this method, you'll probably have to conform. This technique does not (feasibly) allow tool length compensation values to be determined off line, which is one of the reasons we don't recommend it. But there are many situations in which tools cannot be measured off line:

- One person is responsible for all CNC tasks—this person doesn't have time to measure tool lengths off line.
- Lot sizes are very small and cycle times are very short—and no one can keep up with the number of tools that must be measured off line.
- Short lead times—no one knows what cutting tools will be needed in upcoming jobs.
- There are not enough cutting tool components—tools needed in upcoming jobs cannot be assembled.

In these situations, tool length compensation values must be measured on the machine during setup. But do keep in mind that our recommended method can still be used. If you are new to CNC machining centers and will have control of how things are done, we urge you to use our recommended method. But again, many companies are using this second method—and in this case—you'll probably have to adapt.

With this method, the Z-axis workpiece coordinate system offset value is zero. That is, the work offset Z register will be set to a value of zero. The tool length compensation value (that is entered into the tool length compensation offset register) is the distance from the tool tip (at the Z-axis zero return position) to workpiece coordinate system zero. The polarity is negative—all of your tool length compensation offsets will be very large negative values. The figure below shows the offset value.

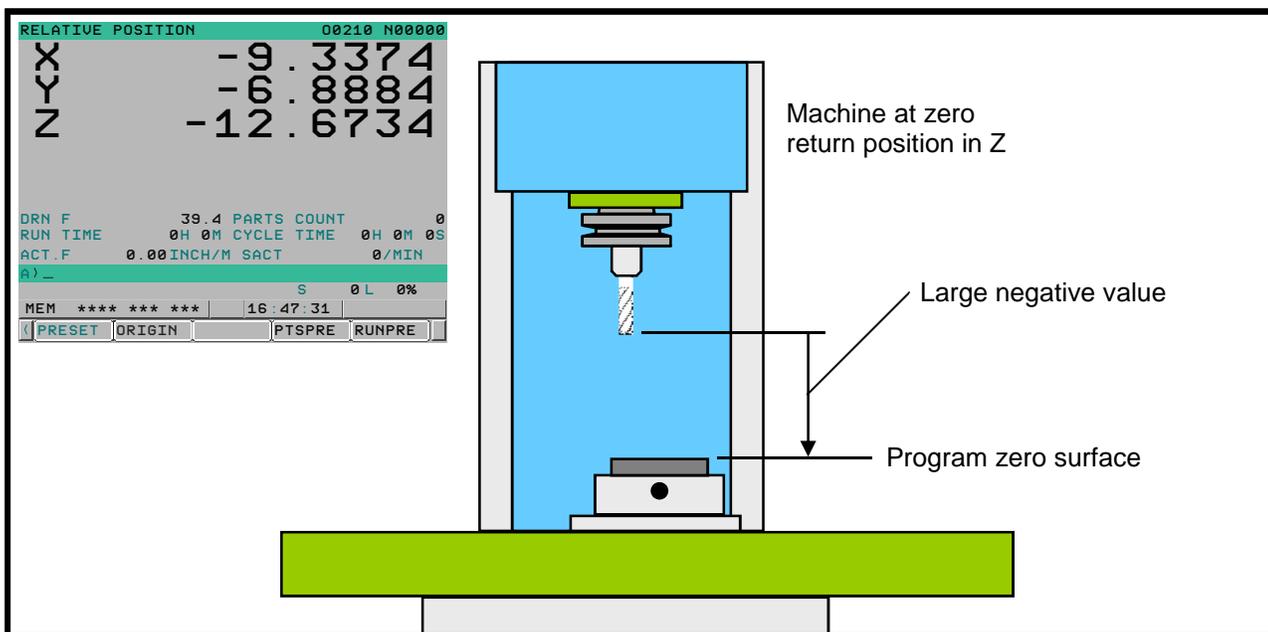


Figure 4.11: Tool length compensation value when tool tip to program zero is used as the offset

Measuring tool length compensation values with this second method

Again, when you use this method, tool length compensation values must be measured on the machine during setup. Here's how.

Step 1: Make the workholding setup and load a workpiece

Program zero will be the top surface of the workpiece.

Step 2: Send the machine to the Z-axis zero return position and reset (set to zero) the Z-axis relative display.

This sets the point of reference for your measurement.

Step 3: Load a tool to be measured and manually bring the tip of the tool to the Z axis program zero surface.

Using incremental jog and/or handwheel, cautiously bring the tool tip to the program zero surface. The Z-axis display follows along. With the tool tip touching the workpiece coordinate system zero surface in Z, the Z-axis relative position display will be showing you the tool length compensation value for this tool. This is the value that must be entered into the tool length compensation offset register.

Reasons for Using the Recommended Method

Here we list a few reasons to use the recommended method. Note that if your company uses the second method shown, these reasons might help you convince people in your company to change.

Tool length compensation values can be measured off line - We've already mentioned this. Minimizing down time between production runs is a very important goal in many companies.

Cutting tools can be used from job to job without being re-measured—This is a very important reason. The length of a cutting tool will not change (unless it is disassembled) between production runs—but the Z axis program zero surface probably will. If using our recommended method, offset values will remain the same for tools used from job to job. If you use the second method, the tool length compensation value for *all* tools must be re-measured in every setup. Consider how many tools get used in consecutive jobs (center drills, spot drills, common drill-and-tap combinations, and common milling cutters). Indeed, many companies assign standard tool stations to most often-used cutting tools. Using the second method can result in a great deal of duplicated effort.

Cutting tools can be used from machine to machine—In similar fashion, a cutting tool's length does not change when it is placed in a different machine. If a tool is not disassembled after a production run (and if it is not dull), write down the tool's length and keep it with the tool. The next time the tool is needed, you won't have to re-measure it—regardless of which machining center uses it.

The work holding setup doesn't have to be made before tool lengths can be measured—You can measure tool lengths at any time—even on the machine.

Multiple identical tools can be setup and kept ready for action—For those tools that are most prone to wearing out—or for those tools that you use the most, you can keep several identical tools ready to go. When a tool dulls, simply replace it in the machine and enter the new tool length compensation value (that has been measured off line).

Working with multiple program zero points is much easier—Though we haven't shown any applications yet, if your company uses a rotary device to expose several surfaces to the spindle for machining during the CNC cycle, you'll find it much easier to do so with our recommended method. This is the case with most horizontal machining centers.

Typical Mistakes with Tool Length Compensation

Mistakes with tool length compensation can have some pretty severe consequences—so we want to prepare you for what can happen when mistakes are made.

Forgetting to instate tool length compensation

As you know, you must instate tool length compensation during every tool's first Z-axis movement, including a G43 and H-word along with the Z-axis departure. If you forget to do so, here's what can happen. Remember that tool length compensation is modal. If you forget to instate tool length compensation for the fourth tool in your program, the control will use the tool length compensation value from the third tool. If the fourth tool is shorter than the third, at least the machine will not try to crash the tool into the workpiece—but it still won't send the tool to the correct Z-axis position.

If you forget to instate for the first tool in the program (and if tool length compensation has not been instated since power-up), tool length compensation is in its cancelled state. If using our recommended method (tool length is offset value), the machine will think that the nose of the spindle is the tool tip. The machine thinks you have a tool with a zero length. The machine will bring the spindle nose to the programmed Z surface, crashing the tool into the workpiece along the way.

Forgetting to enter the tool length compensation value

When going from job to job, the setup person must remember to enter all tool length compensation values for the up-coming job. If they forget to do so, and if there is a value in the offset from the previous job, the machine will

(incorrectly) use this offset value for the new job. If the current value of the offset is zero, the machine will think your tool has zero length—and again—it will try to bring the spindle nose to the programmed Z surface.

Mismatching offsets

Remember, the H-word must correspond to the tool station number. As you write the H word in your program, get in the habit of looking back up into the program for the T word in the most recent tool change (M06) command. The H word must match the tool station number.

What if a Machine Doesn't have Workpiece Coordinate System Offsets?

As you know, the method of using tool length compensation you use depends upon the way you assign program zero in the Z-axis. Everything discussed to this point in this lesson has assumed that your machine has workpiece coordinate system offsets. If you use our recommended method (offset value is the tool's length), the Z-axis register for the workpiece coordinate system offset must be the distance from the spindle nose to the Z-axis program zero surface (a large negative value). If you use the second method shown (offset value is the distance from the tool tip to program zero for each tool), the workpiece coordinate system offset Z register value will be zero.

But as you know from lesson 1.6, very old machines don't have workpiece coordinate system offsets. If your machine does not have workpiece coordinate system offsets, you know you must use G92 to assign program zero (technique to do so is shown in lesson 1.6). If you use our recommended method, the G92 Z value must be the distance from the Z-axis program zero surface to the spindle nose (the same value entered into the fixture offset Z register, only positive). If you use the second method shown, the G92 Z-value must be zero.

Trial Machining with Tool Length Compensation

Tool length compensation will allow trial machining and sizing for depth (Z) dimensions that cutting tools machine. Again, trial machining is required on the first workpiece when a dimension machined by a cutting tool has a very tight tolerance. This technique ensures that the tool will not machine too much material on its very first try. If done for each tool that has a tight Z axis tolerance, the first workpiece will pass inspection.

Trial machining for depth dimensions involves five steps:

1: Recognition of a tight tolerance that worries you—If you're worried that a dimension's tolerance is so small that your initial tool length offset measurement is not accurate enough to make the tool machine the dimension within the tolerance band, then trial machining must be done. For example, say you notice a 0.500 deep pocket with a very tight depth tolerance of plus or minus 0.0002 inch. You're worried that your initial offset setting is off by more than 0.0002 inch—or that tool pressure will cause the tool to machine improperly.

2: Increase the value of the tool length compensation offset by about 0.010 inch (0.25 mm)—Again, this is done after you have measured and entered the tool length compensation offset value. In a pocket milling application, say you have measured the milling cutter's length and found it to be 5.2376 inch long—and you have initially placed this value (5.2376) in the tool length compensation offset register. You'll increase this value to 5.2476 to perform the trial machining operation.

3: Let the tool machine under the influence of the trial machining offset and stop the cycle after the tool is finished—In our example, you will allow the milling cutter to machine the pocket. With the increased offset, the machine will keep the milling cutter 0.010 inch further away from the workpiece during machining (forcing our pocket to come out too shallow).

4: Measure the current dimension and reduce the tool length compensation offset accordingly—In our example, say you measure the pocket depth and find it to be 0.492 inch deep. It is currently 0.008 inches undersize. You must reduce the offset by 0.008 inch, making it 5.2396 (5.2476 minus 0.008). (By the way, if you had not used trial machining techniques and just let the tool cut, it would have machined the pocket about 0.002 inches too deep. And since the tolerance is only plus or minus 0.0002 inch, this workpiece would have been scrap.) If the CNC has a separate tool length geometry and wear offsets, you can enter the variation in the wear field.

5: Re-run the tool under the influence of the adjusted offset—This time, the cutting tool will machine the pocket depth properly—very close to the target dimension (0.500 deep in our case). Any tiny deviation you notice will be caused by the difference in tool pressure from the first time the cutter machines (the normal amount of stock is being removed) to the smaller depth-of-cut after trial machining.

When trial machining is not required

Only tight depth tolerances require trial machining with tool length compensation offsets. You typically do not need trial machining for drilling, tapping, reaming, and most other hole-machining operations.

But just because a cutting tool does not require trial machining techniques doesn't mean you don't have to measure and adjust after the tool has machined for the first time. If the depth dimension has a large tolerance, the tool will machine somewhere within the tolerance band on its first try (the workpiece will be acceptable). But it is still important to adjust the offset in such a way that the next workpiece machined will have the dimension come out to its target value.

In our pocket example, say the pocket depth dimension (0.500 inch) has a tolerance of plus or minus 0.005 inch–0.010 inch overall. In this case trial machining is not required—so you let the milling cutter cut with the initial offset setting. Once the tool is finished you measure the pocket depth and find it to be 0.498 inch deep. This depth is well within the tolerance band, but not right at its target value. In this case, you should decrease the offset by 0.002 inch to make the milling cutter machine 0.002 inch deeper on the next workpiece—making the pocket depth come out right to its target dimension. If this technique is used for all cutting tools, every workpiece attribute will be machined at its target value when you begin the production run.

Sizing with Tool Length Compensation

Sizing is required when the wear a cutting tool experiences during its life affects the surfaces it machines. Frankly speaking, this rarely occurs with depth dimensions (controlled with tool length compensation). If you do notice changes in depth dimensions during a cutting tool's life, remember that you can make the tool machine deeper by reducing the tool length compensation value by the amount of the deviation caused by tool wear.

A tip for remembering which way to adjust the offset

A common mistake made by beginning setup people and operators is adjusting the tool length compensation offset in the wrong direction. Here's a way to remember which way is which.

With our recommended method, think about what will happen if a tool length compensation offset is set to zero. In this case, the machine will think the cutting edge is at the spindle nose, and bring the spindle nose to the programmed surface. So, reducing the offset value will make the tool go deeper.

What if I use the second method shown for tool length compensation?

If you use the distance from the tool tip to program zero as the offset (large negative values in the offset registers), believe it or not, techniques used for adjusting offsets are exactly the same—if you understand polarity.

Again, if you want a tool to go deeper you must reduce the tool length compensation offset value. If you already have a negative offset value in the offset, as is the case with the second method shown, this means you must make the already negative value more negative by the amount of your desired adjustment. If for example, the tool length compensation offset is currently -12.2726 when you measure the pocket depth and find it to be too shallow by 0.002 inch, you must reduce -12.2726 by 0.002 inch, making it -12.2746.

Do I have to make all these calculations when adjusting offsets?

Most CNC controls allow you to modify offset values incrementally. FANUC calls this feature input plus (actually the [INPUT+] soft key on the display screen). With this feature, you need only know the amount of needed offset adjustment. If you need an offset to be reduced by 0.002 inch, you simply type -0.002 and press the [INPUT+] soft key. The control will automatically calculate the new value for the offset and enter it.

Why can't I just change the Z coordinate/s in the program to make sizing adjustments?

There are three reasons to use offsets to make all sizing adjustments. Never change programmed coordinates to make sizing adjustments.

All programmed coordinates must specify mean values

First, and maybe most importantly, it is important that all programmed coordinates specify the mean value of the tolerance band they machine. If milling a 0.500 deep pocket with a tolerance of plus or minus 0.002, the mean value is 0.500, and a position of Z-0.5 must be specified in the program (assuming program zero in Z is at the top of the pocket). Programming mean values is important because it provides some consistency throughout the program, it provides consistency from one time the job is run to the next, and it provides consistency among programs.

Say you machine this pocket and find it to be 0.002 inch too shallow (0.498 deep). If you change the programmed Z coordinate at the pocket bottom from Z-0.500 to Z-0.502, admittedly, this pocket will be machined 0.002 inch deeper.

But let's go a little further. What caused the 0.002 inch deviation? In this case, you must not have correctly measured the cutting tool's length (to be off by this much). Additionally, tool pressure may be affecting the way the tool machines. But the program is correct. Changing the program to deal with tooling problems doesn't make sense.

Also, consider what will happen when this tool eventually dulls and gets replaced. It's likely that you will correctly measure its new length. But when it machines, the incorrect program coordinate (Z-0.502) will cause this tool to machine the pocket too deep.

The same thing goes for the next time the job is run. If the program still has the Z-0.502 coordinate from the last time it was run, the setup person will have no idea how the cutting tool will machine on its first try.

And when a cutting tool is used again in an up-coming job, it will continue machining workpiece attributes to their target values.

Programs cannot be changed while machine is running.

Many sizing adjustments, especially during a production run, can be done while the machine is running. Tool offsets can be modified while the machine is in cycle. Most machining centers do not allow you to modify the active program while the machine is running it.

Lots of program modifications may be required

In our example, there is only one pocket to machine. But say you must machine fifty pockets. This means there will be fifty Z-0.500 words in the program to change. With offset adjustments, only one value must be changed to affect how all fifty pockets will be machined.

Again, never modify programs to make sizing adjustments!

Key points for lesson 4.2:

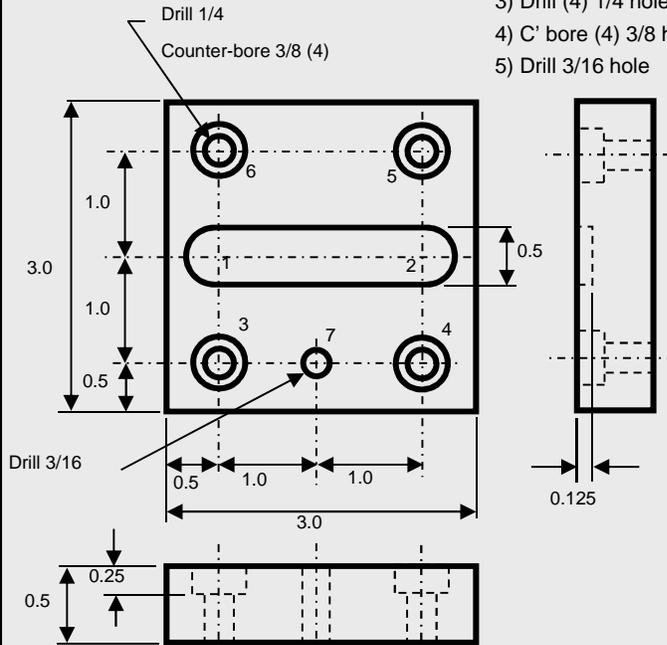
- ✓ Tool length compensation lets the programmer ignore the precise length of each tool prior to writing the program.
- ✓ Tool length compensation is used for every tool in every program and is instated on each tool's first Z axis approach movement to the work surface.
- ✓ Tool length compensation is instated with a G43 word that includes an H word and a Z word. The H word specifies the offset number to use.
- ✓ Make the H word number (offset number) the same number as the tool station number.
- ✓ With our recommended method, the tool offset will contain a value equal to the tool's length (from tool tip to spindle nose).
- ✓ With our recommended method, the fixture offset Z value must be the distance from the spindle nose to the Z axis program zero surface (a large negative value).

Practice with tool length compensation

To mill the slot, simply plunge 1/2 end mill at pt 1, mill to pt 2, and retract.

Process:

- | | | | |
|--------------------------|-----------------|-----------|---------|
| 1) Mill 0.5 wide slot | 1/2 end mill | 611 rpm | 5.5 ipm |
| 2) Center-drill 5 holes | #4 center drill | 1,200 rpm | 5.0 ipm |
| 3) Drill (4) 1/4 holes | 1/4 drill | 950 rpm | 4.5 ipm |
| 4) C' bore (4) 3/8 holes | 3/8 end mill | 600 rpm | 4.0 ipm |
| 5) Drill 3/16 hole | 3/16 drill | 1,630 rpm | 3.6 ipm |



Fill in the coordinates:

	X	Y	Z
1:	_____	_____	_____
2:	_____	_____	_____
3:	_____	_____	_____
4:	_____	_____	_____
5:	_____	_____	_____
6:	_____	_____	_____
7:	_____	_____	_____

N210 _____ M09

N215 G91 G28 Z0 M19

N220 M01

N225 T04 M06 (3/8 end mill)

N230 G54 G90 _____ M03 T05

N235 _____ (pt 3)

N240 _____ M08

N245 _____

N248 _____

N250 _____ (pt 4)

N255 _____

N260 _____

N265 _____ (pt 5)

N270 _____

N275 _____

N280 _____ (pt 6)

N285 _____

N290 _____ M09

N295 G91 G28 Z0 M19

N300 M01

N305 T05 M06 (3/16 drill)

N310 G54 G90 _____ M03 T01

N315 _____ (pt 7)

N320 _____ M08

N325 _____

N330 _____ M09

N335 G91 G28 Z0 M19

N340 M30

Fill in the blanks for this program:

O0001

N005 T01 M06 (1/2 end mill)

N010 G54 G90 S _____ M03 T02

N015 _____ (pt 1)

N020 _____ M08

N023 _____

N025 _____ (pt 2)

N030 _____ M09

N035 G91 G28 Z0 M19

N040 M01

N045 T02 M06 (#4 center drill)

N050 G54 G90 _____ M03 T03

N055 G00 _____ (pt 3)

N060 _____ M08

N065 _____ Z-0.12 _____

N070 _____

N075 _____ (pt7)

N080 _____

N085 _____

N090 _____ (pt 4)

N095 _____

N100 _____

N105 _____ (pt 5)

N110 _____

N115 _____

N120 _____ (pt 6)

N125 _____

N130 _____ M09

N135 G91 G20 Z0 M19

N140 M01

N145 T03 M06 (1/4 drill)

N150 G54 G90 _____ M03 T04

N155 _____ (pt 3)

N160 _____ M08

N165 _____

N168 _____

N170 _____ (pt 4)

N175 _____

N180 _____

N185 _____ (pt 5)

N190 _____

N195 _____

N200 _____ (pt 6)

N205 _____

Answer program is shown on next page.

Answer program for exercise on previous page

O0001	N115 G00 Z0.1	N225 T04 M06 (3/8 end mill)
N005 T01 M06 (1/2 end mill)	N120 X0.5 (pt 6)	N230 G54 G90 S600 M03 T05
N010 G54 G90 S611 M03 T02	N125 G01 Z-0.12	N235 G00 X0.5 Y0.5 (pt 3)
N015 G00 X0.5 Y1.5 (pt 1)	N130 G00 Z0.1 M09	N240 G43 H04 Z0.1 M08
N020 G43 H01 Z0.1 M08	N135 G91 G20 Z0 M19	N245 G01 Z-0.25 F4.0
N023 G01 Z-0.125 F5.5	N140 M01	N248 G00 Z0.1
N025 X2.5 (pt 2)		N250 X2.5 (pt 4)
N030 G00 Z0.1 M09	N145 T03 M06 (1/4 drill)	N255 G01 Z-0.25
N035 G91 G28 Z0 M19	N150 G54 G90 S950 M03 T04	N260 G00 Z0.1
N040 M01	N155 G00 X0.5 Y0.5 (pt 3)	N265 Y2.5 (pt 5)
	N160 G43 H03 Z0.1 M08	N270 G01 Z-0.25
N045 T02 M06 (#4 center drill)	N165 G01 Z-0.605 F4.5	N275 G00 Z0.1
N050 G54 G90 S1200 M03 T03	N168 G00 Z0.1	N280 X0.5 (pt 6)
N055 G00 X0.5 Y0.5 (pt 3)	N170 X2.5 (pt 4)	N285 G01 Z-0.25
N060 G43 H02 Z0.1 M08	N175 G01 Z-0.605	N290 G00 Z0.1 M09
N065 G01 Z-0.12 F5.0	N180 G00 Z0.1	N295 G91 G28 Z0 M19
N070 G00 Z0.1	N185 Y2.5 (pt 5)	N300 M01
N075 X1.5 (pt7)	N190 G01 Z-0.605	
N080 G01 Z-0.12	N195 G00 Z0.1	N305 T05 M06 (3/16 drill)
N085 G00 Z0.1	N200 X0.5 (pt 6)	N310 G54 G90 S1630 M03 T01
N090 X2.5 (pt 4)	N205 G01 Z-0.605	N315 G00 X1.5 Y0.5 (pt 7)
N095 G01 Z-0.12	N210 G00 Z0.1 M09	N320 G43 H05 Z0.1 M08
N100 G00 Z0.1	N215 G91 G28 Z0 M19	N325 G01 Z-0.586 F3.6
N105 Y2.5 (pt 5)	N220 M01	N330 G00 Z0.1 M09
N110 G01 Z-0.12		N335 G91 G28 Z0 M19
		N340 M30