Cutter radius compensation is only used for milling cutters. Just as tool length compensation lets the programmer ignore the precise length of cutting tools as they write programs, cutter radius compensation allows them to ignore the precise diameter of milling cutters used for contour milling. And also like tool length compensation, setup people and operators must enter and manipulate the related offsets during the tools initial setup, and for trial machining and sizing.

You know from Lesson One that milling cutters (like end mills) can be used for side-milling operations. With a side-milling operation, the milling cutter is machining on its periphery (around its diameter). Figure 10.1 shows some examples.

![Examples Of Side-Milling Operations](image)

Figure 10.1 – Side-milling with an end mill

Some people refer to these side-milling operations as contour milling operations. But as you can see from Figure 10.1, a milling cutter may be milling but one straight surface - and it is still be considered to be a side-milling operation.

**Do you need to learn about cutter radius compensation?**

Unlike tool length compensation - which is used for every cutting tool in every program - cutter radius compensation is only used for milling cutters, and only when side-milling. Some companies do not perform side-milling operations - they may, for instance, only perform hole-machining operations on their machining centers. In this case, there will not be much of a need to learn about cutter radius compensation.

Even if you work for such a company, you will still want to know the reasons why cutter radius compensation is required for side milling. So at the very least, skim this lesson to make sure you know what is included. If the need ever arises, you can always come back to this lesson and dig in.
Similarities to tool length compensation

From a setup person’s or operator’s viewpoint, cutter radius compensation is very similar to tool length compensation. We will minimize our presentations during this lesson when discussing topics that are similar to tool length compensation. We will assume that you have read and understand Lesson Nine. Here is a list of the similarities.

- The setup person will measure the cutting tool and enter an offset value for the cutting tool. With cutter radius compensation, they will enter the milling cutter’s radius into the cutter radius compensation offset register. The cutter’s radius, of course, is half its diameter. So the setup person will first measure the cutter’s diameter (commonly with a micrometer) and then divide the diameter by two.

- The initial offset entry may not be perfect. The cutter may be running out (wobbling slightly) in its tool holder, which will cause it to machine too much stock. Or tool pressure may cause the milling cutter to deflect from the surfaces being machined. If tolerances are small, trial machining must be done to ensure that the milling cutter properly machines the first workpiece.

- The production run will begin with all machined surfaces at their target values.

- Tool wear will cause the milling cutter’s diameter to shrink. Sizing adjustments may be required during the production run to deal with this shrinkage. The more abrasive the material, the smaller the tolerance, and the more workpieces that must be produced, the more likely it will be that sizing adjustments will be required.

- The polarity for making adjustments remains the same. If more material must be removed, the adjustment polarity will be negative. If less material must be removed, the adjustment polarity will be positive.

- The plus-input key can be used during adjustments to eliminate the need to calculate offset values.

- When the milling cutter is dull, it must be replaced. Many of the same tasks required during the initial setup of the milling cutter (assembly, measurement, loading, and trial machining) will be required for the replacement milling cutter.

This should sound pretty familiar. While we will cover what you need to know about cutter radius compensation in detail in this lesson, again, we will assume you understand many of the points made in Lesson Nine.

Reasons why cutter radius compensation is required

Let’s begin by discussing why cutter radius compensation is needed – and why you must understand cutter radius compensation. Some of the reasons are similar to the ones given for tool length compensation.

Coordinate calculations are simplified

While this reason is more important to CNC programmers, it is also helpful to setup people that are required to verify and optimize the CNC program. In most companies, however, when a problem is found with the program, the programmer is called in to correct it. But there are companies in which setup people are responsible for making minor program changes. This means, of course, that they must be familiar enough with the related programming words and commands in order to do so.

Though it is not our intention to present programming techniques (a programming manual is sold separately that will), we do want to describe this reason for cutter radius compensation. When performing contour milling operations without cutter radius compensation, the programmer must specify programmed coordinates for the milling cutter’s centerline path. This requires them to consider the size of the milling cutter in every coordinate calculation – complicating each calculation. With cutter radius compensation, the programmer can specify coordinates that are right on the work surface, ignoring the size of the milling cutter. Figure 10.2 shows the two different tool paths.
Again, we are not going to present programming techniques in this text. But you should be able to see from Figure 10.2 that it is much easier to calculate coordinates (the positions of points 1, 2, 3, 4, 5, etc.) with the drawing shown on the right side of Figure 10.1 than it is for the one on the left side. For the drawing on the right, many of the needed coordinate positions will be specified right on the blueprint. For the drawing on the left, every coordinate position requires the cutter’s radius to be added-to or subtracted-from dimensions specified on the blueprint.

**Range of cutter sizes can be used**

This reason for using cutter radius compensation applies to setup people and operators, as well as programmers. Just as tool length compensation allows the length of a given cutting tool to vary without requiring a program change, so does cutter radius compensation allow the milling cutter’s diameter (and of course, its radius) to vary without requiring a program change.

Without cutter radius compensation, the setup person must use the precise milling cutter called for by the programmer. If the programmer specifies a 1.0 in (25.4 mm) cutter, the setup person must use a 1.0 in cutter. If any other cutter size is used, the cutter will not mill along the correct path. All machined surfaces will be larger-than or smaller-than their intended sizes (by the difference in radius between the planned cutter size and the size of the cutter being used). Figure 10.3 illustrates this.
Cutter radius compensation allows a range of cutter sizes to be used for the side milling operation. This range should be specified in the setup documentation, but frankly speaking, it seldom is. Many programmers assume that the setup person will use a milling cutter very close to the specified size – so it will be within the intended range.

If the cutter size range is not specified, you can always use a smaller cutter without having to worry about machine problems. But you must be careful with larger cutters. If the cutter is too large, the machine will generate an alarm during its motions to machine the surfaces.

A programmer may, for example, program a side milling operation in a way that allows any milling cutter from 0.75 in to 1.0 in (19.05 mm to 25.4 mm) in diameter to be used. Again, they should specify this range in the setup documentation.

The small end of the range (0.75 inch in our case) is usually the smallest cutter that can effectively machine the surfaces and this size has nothing to do with machine problems. If a smaller cutter is used, the machine will make the correct motions, but the cutter may not be strong and rigid enough to mill the surfaces. It may break during the machining operation.

The large end of this range (1.0 inch in our case) is the largest cutter that can be used without machine problems. If a larger cutter is used, it may violate some of the surfaces (like a recess or undercut), causing the machining center to generate an alarm.

Though there are limitations to the range of cutter sizes that can be used with cutter radius compensation (especially when compared to tool length compensation), having the ability to use a slightly different cutter than intended is still an important benefit. It keeps the programmer from having to create a new tool path in the program just because you don’t have a cutter of the intended size.

**Do you use sharpened (re-ground) milling cutters?**

Many companies sharpen their dull milling cutters and use them again. When a milling cutter is sharpened, its diameter will become smaller by about 0.020 inch or so (0.50 mm). Figure 10.4 shows some sharpened milling cutters. The milling cutter’s tool path must, of course, be modified whenever a sharpened cutter is used – due to its smaller diameter. Again, cutter radius compensation will automatically modify the milling cutter’s tool path based upon the current size of the sharpened cutter. If you do not use cutter radius
compensation, you cannot use sharpened cutters without the programmer changing the cutter’s motions in the program.

![Sharpened cutters](image)

**Figure 10.4 – Many companies use sharpened cutters**

---

**Trial machining and sizing**

Just as tool length compensation allows trial machining and sizing for depth surfaces, so does cutter radius compensation allow trial machining and sizing for XY surfaces (surfaces milled with the periphery of the milling cutter). Like depth surfaces, many XY surfaces have close tolerances. If you do not use cutter radius compensation, there will be no way to make sizing adjustments without actually changing the tool path coordinates in the program (which is usually difficult to the point of being infeasible - and we never recommend changing programmed coordinates to make sizing adjustments).

Also as with machined depth surfaces, tool pressure will affect how XY surfaces are milled. Again, just because a milling cutter’s coordinates have been programmed perfectly (to the mean value for each tolerance) - and just because a milling cutter of exactly the intended size is being used - it is no guarantee that the milling cutter will machine the XY surface/s perfectly to size. The tighter the tolerance/s to be held and the smaller the milling cutter’s diameter, the more likely the milling cutter will not correctly machine the surface/s.

There is also a tooling-related problem that affects the accuracy of contour milling operations. Milling cutters have a tendency to run-out in their holders. That is, the milling cutter will not be perfectly concentric with the tool holder and spindle. Run-out can be as much as 0.002 in (0.05 mm) and will cause the milling cutter to machine more material than it should - having the same effect as using a slightly larger milling cutter.

Milling cutters also have a tendency to wear during their lives. As the cutter wears, a small amount of material will be removed from its outside diameter. In effect, the milling cutter becomes smaller in diameter. With tight tolerances, this small amount of variance will cause problems. Sizing must be done during the tools life to keep the milling cutter machining surfaces to size.

For these reasons, you must have the ability to perform trial machining and sizing with XY milling operations. Whether the cutter is simply milling one straight surface (like the right end of the workpiece), or machining an elaborate contour, cutter radius compensation will allow you to do so.

---

**Rough and finish milling with the same set of coordinates**

When a machining operation requires a great deal of material to be removed - and especially when the machined surfaces have small tolerances - it is common practice to rough machine the surfaces. A separate cutting tool (called the roughing tool) will machine the surfaces within a small distance from their final size. That is, after roughing, there will still be material on the surfaces that must be machined. This material is call finishing stock. The amount of finishing stock left on the surfaces is based upon the kind of machine operation being performed, and ranges from about 0.01 in to 0.03 in (0.25 mm to 0.76 mm). A different cutting tool (called the finishing tool) will then machine the rest of the material, bringing the surfaces to their final sizes.

Roughing then finishing in this manner allows efficient rough machining (a special cutting tool is commonly used that is specifically designed for rough machining) and extended tool life for finishing (the finishing cutting tool is removing only a small amount of material - which allows it to last longer). Note that this concept applies to many kinds of machining operations, but for our discussions, we’ll focus on side-milling operations.
When programming rough and finish side milling operations, it is necessary for the programmer to come up with two different sets of motions - one for the rough milling cutter and the other for the finish milling cutter. Figure 10.5 shows this.

![Figure 10.5 – The rough and finish pass required for a side-milling operation.](image)

It can be difficult enough for the programmer to calculate the coordinates required for the finishing path - even when cutter radius compensation is used and when programming the work-surface path. It becomes even more difficult for the programmer to calculate another set of coordinates for the roughing path. The roughing path must be the finishing stock distance away from the finishing path. With cutter radius compensation, the programmer can use the same set of coordinates for both sets of tool paths.

When the programmer uses the same tool path for both rough milling and finish milling, it should be specified in the setup documentation. The cutter radius compensation offset value for the rough milling cutter must be larger than its actual value by the amount of finishing stock that is left on all surfaces. If, for example, the programmer intends to leave 0.03 in (0.76 mm) finishing stock on all surfaces, the cutter radius compensation value for the rough milling cutter must be 0.03 larger than its actual value. For a 1.0 in diameter rough milling cutter, the cutter radius compensation value will be 0.53 in. This will keep the rough milling cutter 0.03 in away from the programmed (finishing) path and leave 0.03 in finishing stock on all surfaces.

The cutter radius compensation value for the finish milling cutter, of course, will be its actual size.

As you can imagine, using this method to allow finishing stock can lead to mistakes. If the setup person is unaware of the fact that the programmer is using this technique, they will place the actual cutter size for the rough milling cutter in the cutter radius compensation offset. When this cutting tool machines, of course, it will not leave any finishing stock - which could result in a scrap workpiece.

For this reason, this method of leaving finishing stock is becoming less popular. More and more programmers are generating two sets of tool paths, one for the rough milling cutter and another for the finish milling cutter. This allows the setup person to place the actual cutter size in the cutter radius compensation offset for both tools - eliminating the potential for mistakes.

**The setup person's responsibilities with cutter radius compensation**

In the setup documentation, the programmer must specify the milling cutters in a program that use cutter radius compensation. They should also specify the range of acceptable cutter sizes for each milling cutter (as discussed earlier). And they should specify which cutter radius compensation offset number/s are being used.

The setup person must be alert when studying the setup documentation. They must be able to spot the milling cutters that use cutter radius compensation and act accordingly.
Which offsets are involved?
Setup people and operators must know, of course, the offset register number into which a given milling cutter’s radius must be entered. Again, cutter radius compensation offset numbers should be specified in the setup documentation. Most programmers use a consistent technique for determining the cutter radius compensation offset number used for a given milling cutter. The cutter radius compensation offset number will be tied in some way to the cutting tool’s automatic tool changer magazine station number.

In Lesson Seven, we introduce the two types of tool offset display screens. Again, with one type (commonly found on older machines), there is only one offset register per offset. With this kind of offset table, two separate offsets must be used for milling cutters that use cutter radius compensation. One is used for the tool length compensation value and the other is used for the cutter radius compensation value. We have stated that most programmers will use the cutting tool’s tool station number as the tool length compensation offset number. Tool number four, for example, will use offset number four for the tool length compensation offset value.

For cutter radius compensation offset numbers with this kind of offset table, most programmers will add a constant number that is equal to or greater than the number of cutting tools that can be placed in the machine’s automatic tool changer magazine. If the magazine can hold 25 tools, for example, the constant number might be thirty. In this case, the cutter radius compensation offset number for a milling cutter placed in station number four will be offset number thirty-four.

Most machining center made today have two registers per offset. One (commonly labeled length) is used for tool length compensation and the other (commonly labeled radius) is used for cutter radius compensation. Determining offset numbers for this kind of offset table is much easier. Both offset types (tool length and cutter radius compensation) will use the same offset number – and most programmers make the offset number match the cutting tool’s station number. A milling cutter placed in station number four will use offset number four for both the tool length and cutter radius compensation values.

The two ways to use cutter radius compensation
As with tool length compensation, there are two ways to use cutter radius compensation. And just as with tool length compensation, the setup person is probably not the person who decides which method is used. This means you will have to adapt to the method your company is using – and it means you should understand both methods. But unlike tool length compensation, there is no clear-cut advantage for using one method over the other. Generally speaking, manual programmers (those that write programs in G code) tend to use the first method we show. Programmers that use computer aided manufacturing (CAM) systems tend to use the second method we show.

To setup people and operators, the only difference is related to the value that is placed in the cutter radius compensation offset.

When the work surface is the programmed path
With this method, the programmer specifies coordinates right on the work surface. The cutter radius compensation offset value will be the cutter’s radius. If a 1.0 inch (25 mm) cutter is used, the cutter radius compensation offset value will be 0.5 inch (12.5 mm).

When the cutters centerline path is programmed
With this method, the programmer specifies coordinates based upon the centerline path of a planned cutter size. The cutter radius compensation offset value will be the radial difference from the planned cutter size to the cutter size that is actually being used. There is a polarity to the offset entry. If the actual cutter size is larger than the planned cutter size, the polarity will be positive. If the actual cutter size is smaller than the planned cutter size, the polarity will be negative. Here are three specific examples.

? The programmer plans to use a 1.0 inch diameter cutter. During setup, the setup person has a cutter that is precisely 1.0 inch in diameter. In this case the cutter radius compensation offset value will be zero.
The programmer plans to use a 1.0 inch diameter cutter. During setup, the setup person cannot find a 1.0 inch diameter cutter. But they do find a sharpened cutter that originally started out as a 1.0 inch diameter cutter. Its current diameter is 0.980 inch. In this case the cutter radius compensation offset value will be -0.01. This value is negative because the actual cutter is smaller than the planned cutter. The value is 0.01 is calculated by first subtracting the actual cutter size (0.98) from the planned cutter size (1.0). The result is 0.02. This value (0.02) is then divided by 2 to come up with the radial difference in cutter size (0.01).

The programmer plans to use a 1.0 inch diameter cutter. During setup, the setup person cannot find a 1.0 inch diameter cutter. But they do find a 1.125 diameter cutter and decide to use it. In this case the cutter radius compensation offset value will be 0.0625. This value is positive because the actual cutter is larger than the planned cutter. The value is 0.0625 is calculated by first subtracting the planned cutter size (1.0) from the actual cutter size (1.125). The result is 0.125. This value (0.125) is then divided by 2 to come up with the radial difference in cutter size (0.0625).

The programmer must, of course, specify the planned cutter size for each milling cutter that uses cutter radius compensation in the setup documentation.

**Relationship between the cutter radius compensation value and the machined surfaces**

If you don’t perfectly measure cutter radius compensation values, the cutting tool will not go exactly to the programmed XY surfaces. For example, say the cutting tool is machining the overall length of a six inch long workpiece. The programmer correctly specifies a coordinate of X6.0 in the program to bring the cutting tool to one end of the workpiece.

If the cutter radius compensation value is smaller than it should be, the cutting tool will machine the workpiece shorter than the programmed depth. In this case, the machine thinks the milling cutter is smaller than it really is. If the cutter radius compensation value is larger than it should be, the milling cutter will machine the workpiece longer than it should. In this case, the machine thinks the milling cutter is larger than it really is.

**What is the polarity for an adjustment?**

With almost every milling cutter you measure, there will be small deviation between the milling cutter’s actual radius and the radius you measure. Additionally, tool pressure – which we have already defined as being the cutting tool’s tendency to push away from the surface it machines - will affect the positions to which a cutter will actually go during machining. This means that the measured size of a workpiece attribute will almost never perfectly match the programmed size. It also means you must be able to make adjustments. And every adjustment will have a polarity (plus or minus).

Here is the rule-of-thumb for adjustment polarity: If a milling cutter must remove more material, the adjustment must be in the negative direction. If the milling cutter is removing too much material (you want it to remove less material), the adjustment will be in the positive direction.

**Trial machining with cutter radius compensation**

Just as tool length compensation allows trial machining and sizing for depth (Z) surfaces, so does cutter radius compensation allows trial machining for XY surfaces. Again, trial machining is required when the milling cutter machines for the first time if surfaces have a small tolerance. This technique ensures that the cutter will not machine too much material on its very first try and gives the setup person the ability to perfectly size-in the surfaces being machined. If trial machining is done for each critical machining operation, the first workpiece will pass inspection.

Trial machining for XY surfaces requires the same five steps as it does for depth surfaces. We’ll be referring to the drawing shown in Figure 10.6 during these descriptions.
1: **Recognition of a tight tolerance that worries you** – If you’re worried that a surface’s tolerance is so small that your initial cutter radius compensation offset setting is not accurate enough, trial machining must be done if you want to guarantee that the cutting tool machines the first workpiece correctly. In our case, a milling cutter is milling the left end of this workpiece. As you are running the very first workpiece, you notice the very tight +/- 0.0004 inch tolerance for the overall length of the workpiece. You’re worried that your initial cutter radius compensation offset setting is not accurate enough to machine the surface within its tolerance band. If the cutter is running-out in its holder even a tiny amount, this dimension will be machined undersize, scrapping the workpiece.

2: **Increase the value of the cutter radius compensation offset by about 0.010 inch (0.25 mm)** – Since we’re using a 1.000 inch diameter milling cutter, we’ll increase the initial offset value to 0.510, 0.010 larger than the true radius of this cutter.

3: **Let the milling cutter machine under the influence of the trial machining offset and stop the cycle after it is finished** – In our example, the milling cutter will be milling the left end of the workpiece. When it is finished, you stop the machine. The trial machining offset will force the milling cutter to leave excess stock on the machined surface.

4: **Measure the surface and reduce the cutter radius compensation offset accordingly** – If everything is perfect, the current overall length of this workpiece will be exactly 4.510. But when you measure, say you find it to be 4.508. It is currently only 0.008 inch oversize – and it’s a good thing you trial machined. Either you did not measure the diameter of the end mill correctly or the end mill is running out in its holder. Either way, you must reduce the cutter radius compensation offset by 0.008, making it 0.502. Note that if you had not trial machined, the surface would have been milled 0.002 undersize, which would scrap the workpiece.

5: **Re-run the milling cutter under the influence of the adjusted offset** – This time, the milling cutter will machine the overall length perfectly to size. You measure just to be sure. It comes out to 4.9999. The tiny 0.0001 inch deviation is caused by the difference in tool pressure from the first time the cutter machined (the normal amount of stock is being removed) to the smaller depth-of-cut it took after trial machining. The next workpiece (when the milling cutter takes its full depth-of-cut) will be perfect.

---

**Figure 10.6 – Drawing for trial machining example**

- **Milling the left side of a workpiece**
- **Stock to be machined**
- **1.000 end mill**
- **0.500 is in the offset**
- **Vise with end stop**
- **4.500 +/-0.0004**
- **3.0**
How do you know when to trial machine?

We’ve said that when tolerances are under about 0.002 inch (overall), your initial tooling measurements may not be accurate enough to ensure that the first workpiece is machined correctly. But this is largely a matter of your own personal experiences — and some trial-and-error.

When in doubt, it’s best to trial machine. And when you’re trial machining, you can learn a lot. When you increase the initial offset to trial machine, by say 0.010, the cutter should machine leaving exactly 0.010 too much stock. Did it? If it did not, how close was it?

This difference is the amount of error in your initial setting — caused mostly by imperfect measuring — with possibly a little tool pressure. It tells you whether you needed to trial machine in the first place.

If the difference is well less than your tolerance, the workpiece would have been machined properly — without trial machining. If this happens consistently, you shouldn’t have to trial machine for this tolerance with this kind of tool and workpiece material in the future.

Warning about milling multiple surfaces
In our example for trial machining, the milling cutter is only machining one end of the workpiece — and it’s pretty easy to make sizing adjustments. But keep in mind that milling cutters often machine on two sides of the workpiece. Say, for example, the program is milling both ends of the workpiece shown in Figure 10.6. When you increase the offset value by 0.01 for trial machining, you may be in for a bit of a surprise. After machining, an extra 0.01 (approximately) inch will be left on both surfaces. When you measure the 4.500 inch part length, you will find it to be close to 4.520. In the specific example just given (having 0.002 inch of error on each surface), it will come out to 4.516. The cutter radius compensation offset must still be reduced by 0.008, so that it will remove 0.008 more material from each end of the workpiece.

Whenever a milling cutter is machining multiple surfaces, you must be very alert when making adjustments. Remember that your adjustments will affect every surface machined by the milling cutter.

How important is it that you make your first workpiece a good one?
We’ve been talking a great deal about trial machining to ensure that your first workpiece gets machined properly. There are companies, however, that aren’t overly concerned if setup people scrap the first workpiece — as long as they learn enough in the process to make the next workpiece properly. When procuring raw material, these companies actually purchase more than the quantity needed in the production run (I’ve heard people refer to the extra material as setup pieces or practice pieces).

While these companies do exist, the vast majority of CNC using companies expect their setup people to make every workpiece a good one. They know that if their setup people use trial machining techniques for dimensions with critical tolerances as they machine the first workpiece, the first workpiece will be an acceptable workpiece. While mistakes are bound to occur, there is really no excuse for scrapping the first workpiece if trial machining techniques are used (assuming the program and process are correct).

When trial machining is not required
Again, only tight XY tolerances require trial machining with cutter radius compensation. If the milled surface has a large tolerance, you should be able to measure the cutter radius compensation value accurately enough. The cutter will machine somewhere within the tolerance band on its first try (the workpiece will be acceptable). But after machining, it is still important to measure the milled surface/s and adjust the cutter radius compensation offset in such a way that the next workpiece machined will have the dimension come out right at its target value.
In our example for milling the left side of a workpiece (Figure 10.6), say the overall length tolerance is plus or minus 0.005 inch. In this case trial machining is not required – so you let the milling cutter machine with the initial offset setting (0.500 in our case). Once the cutter is finished you measure the overall length and find it to be 4.502. It is 0.002 larger than its target value, but not out of tolerance (it is a good workpiece). In this case, you should decrease the offset by 0.002 inch to make the milling cutter machine 0.002 inch more material on the next workpiece. When a production run begins, all machined surfaces should be being machined at their target values.

### Sizing with cutter radius compensation

Sizing is required when the wear a cutting tool experiences during its life affects the position of the surfaces it machines. With very tight tolerances (say, under about 0.0004 overall or so), you may need to perform several sizing adjustments during a milling cutter’s life. The example shown in Figure 10.6 may require this kind of sizing. Let’s discuss it further.

After trial machining for the first workpiece, the setup person has the 4.500 overall-length-dimension coming out perfectly to size. They turn the job over to a CNC operator to run production. After thirty workpieces or so, the operator measures the 4.500 dimension and finds it to be 4.5001. The cutter has worn a tiny amount. It is still cutting properly and does not have to be replaced - but it is showing signs of wear. After thirty more workpieces, the operator measures again and finds the overall length to be 4.5002. It is still within the tolerance band, but if this trend continues for much longer, workpieces will be machined with this dimension out of its tolerance band.

At this point, the operator decides to make a sizing adjustment. They reduce the milling cutter’s cutter radius compensation value by 0.0002. The next workpiece to be machined will come out at the 4.500 target value. This sizing adjustment may have to be repeated several times (based upon the size of the tolerance and how much the milling cutter wears) before the milling cutter is completely dull and must be replaced.

**Does the method of using cutter radius compensation affect the way sizing is done?**

Making offset adjustments for trial machining and sizing will be exactly the same, regardless of which method is used. You’ll just be working with different offset values. When you want to machine more material, reduce the offset value (regardless of its current value). When you want to make the cutting tool remove less material (as is required when trial machining), increase the offset value.

**Can you change programmed coordinate/s in the program to make sizing adjustments?**

As we said during our discussions of tool length compensation, most CNC controls allow you to modify offset values incrementally. Fanuc calls this feature input plus (actually INPUT+ 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 + key. The control will automatically calculate the new value for the offset and enter it.

**Dull tool replacement**

As stated in Lesson Nine, many of the tasks required during the initial setup of the cutting tool must be repeated for the replacement tool. This just expands to include cutter radius compensation tasks (measuring cutter radius, entering the offset value, and trial machining if required). Please see Lesson Nine for other tasks that must be done.
Key points for Lesson Ten:

- Cutter radius compensation is used only for milling cutters – and only when contour milling.
- Cutter radius compensation allows the programmer to ignore the size of the milling cutter as the program is being written.
- Cutter radius compensation simplifies calculations for programming, allows a range of cutter sizes to be used, allows trial machining and sizing, and allows one tool path to be used for both rough milling and finish milling.
- The setup person must store the cutter’s radius in the cutter radius compensation offset (based upon our recommended method for manual programmers).
- Trial machining can be done when the setup person is worried that the initial offset entry isn’t accurate enough.
- Sizing adjustments may be necessary during the production run to keep machined surfaces on size.
- For sizing, if you want the milling cutter to remove more material, reduce the offset value.

Conclusion to Key Concept Number Three:

It is imperative that CNC setup people and operators understand the three compensation types.

Fixture offsets are especially important to setup people. Since most companies make at least some unqualified setups, setup people will have to assign program zero on their own for at least some of the jobs they run. And many companies make very few qualified setups, meaning setup people must assign program zero for most or all of the jobs they run. Once program zero is assigned for a job, there should be no need to deal with fixture offsets again until the next job, meaning CNC operators shouldn’t have to modify program zero assignment values once a production run begins.

Tool length compensation is important to both setup people and operators. Every cutting tool in every program requires tool length compensation to be used. Setup people must initially determine and enter the tool length compensation value for each tool into the appropriate offset register. During the machining of the first workpiece, they must then manipulate the tool length compensation offset value to make each workpiece depth surface come out to its target value – so the production run begins with each attribute being nearly perfectly machined. Operators may have to manipulate tool length compensation values in order to keep depth surfaces coming out within their tolerance bands. And when cutting tools get dull, they will have to repeat many of the tasks a setup person performs with the cutting tool during the initial setup.

Cutter radius compensation, when used, is also important to both setup people and operators. It is only required for milling cutters that perform side-milling operations. Setup people must initially determine and enter the cutter radius compensation value for each tool into the appropriate offset register. During the machining of the first workpiece, they must then manipulate the cutter radius compensation offset value to make each milled XY surface come out to its target value – so the production run begins with each attribute being nearly perfectly machined. Operators may have to manipulate cutter radius compensation values in order to keep XY surfaces coming out within their tolerance bands. And when cutting tools get dull, they will have to repeat many of the tasks a setup person performs with the cutting tool during the initial setup.

We cannot overstress the importance of the topics presented in this Key Concept. While there are several other physical tasks that must be performed during setups and production runs, setup people and operators will be working with compensation in every job. Your ability to run good workpieces is based upon your understanding of this Key Concept.