

Lesson 1**Basic Machining Practices Required For CNC Machining Centers**

The more a person knows about the basic machining practices that apply to CNC machining centers, the easier it will be to become a proficient setup person or operator. We cannot over stress the importance of understanding shop practices needed for successful machining center use.

In this lesson, we introduce you to some of the most important principles you must understand *before* you begin learning about CNC machining center usage. The more you know about these topics, the easier it will be to learn to program, setup, or operate a CNC machining center.

There are five basic-machining-practice-related topics discussed in this lesson:

- ✍ Shop safety
- ✍ Shop math
- ✍ Blueprint reading
- ✍ Tolerance interpretation
- ✍ Measuring tools

Any one of these topics could probably fill a manual larger than this one. So keep in mind that we are merely *introducing* you to each topic. Our intention is to provide enough information so that a newcomer to the machine shop environment will understand the presentations made throughout this text. But this will not be enough to fully master any of these topics.

If you are a novice, you will surely need more than this text offers about basic machining practices. At the very least, you will need the help of an experienced machinist when you begin working in the shop. You can also learn more by reading books on each of these topics – or you can attend related courses offered by technical schools.

Shop Safety

A machine shop is a very dangerous place. There are hazards everywhere – and you must be very careful with everything you do and everywhere you go. While we offer some basic suggestions for keeping safe, we cannot possibly address every hazard in a machine shop. Most companies have safety instruction as part of the orientation for new-hires. We bow to an experienced person in your company or school to provide more specific safety instruction than what we show in this text. But at least this presentation should make you aware of some of the most important safety issues.

Safety equipment

Protective eyewear – most shops demand that you wear safety glasses at all times when you are in the shop – regardless of whether you are actually working on a machine or not. Most companies have signs on every shop entry door that say: **“Safety glasses required beyond this point.”** In areas where debris may be flying everywhere (like welding and grinding departments), people are often required to wear full-face shields.

Clothing – The clothing you wear when working in a machine shop has an impact on your safety. Most shops require that you wear long sleeve shirts and long pants (no tee-shirts or shorts) to protect your arms and legs from simple hazards. Your clothes should be close-fitting – don’t wear any loose fitting clothes that could get snagged or hooked in the shop – or worse – grabbed by a moving machine part. Neck ties are banned from most machine shops. Remove all jewelry (finger-rings, ear-rings, bracelets, necklaces, etc.) when

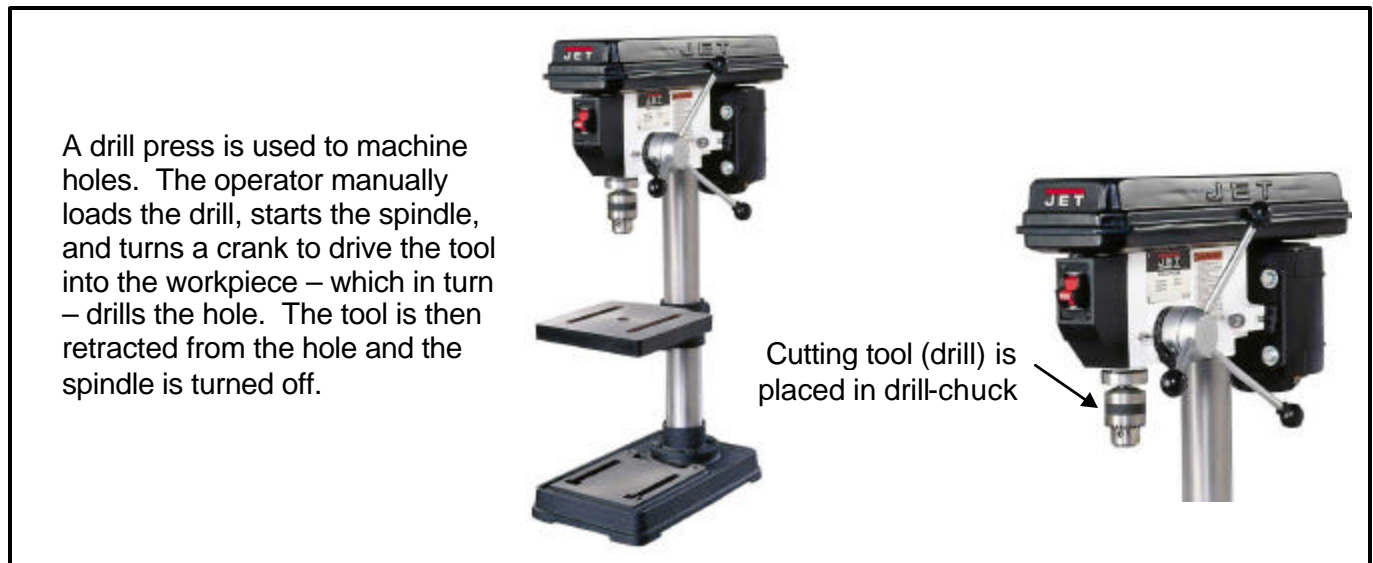
Lesson 2**What Does A CNC Machining Center Do?**

A CNC machining center is the most popular type of metal cutting CNC machine because it is designed to perform some of the most common types of machining operations. It is important to understand these machining operations in order to properly use a CNC machining center.

In this lesson, we will describe the function of CNC machining centers. We will start by comparing CNC machining centers to other types of machines with which you may have some experience. We will then introduce the cutting conditions that are related to machining operations. Finally, we will describe – in detail – the two most basic kinds of machining operations that can be done on machining centers, including hole machining operations and milling operations. At the completion of this lesson, you will know what a CNC machining center is designed to do.

Comparing a CNC machining center to other types of machines

CNC machining centers replace certain *conventional* machine tools. While you may not have experience with CNC machining centers, it is likely that you have had experience with at least one of the conventional machines they replace: the drill press. At the very least, you probably know what a drill press is and/or have seen one in use. If so, you know that the primary function of a drill press is to drill holes. The next illustration shows a typical home-shop drill press.



A typical drill press

A drill-bit is but one of many cutting tools that can be used to machine holes. And while a drill press can be used to machine holes with other types of cutting tools, most people in a home-shop don't do much more than drilling operations. We'll discuss other types of hole-machining tools a little later.

Lesson 3**General Flow Of The CNC Process**

The tasks of programming, setup, and operation are but three of the things that must be done in order to actually get a CNC job up and running. It really helps to understand how these tasks fit into the bigger picture of a company's manufacturing environment.

CNC machine tools are being used by all sorts of companies. Indeed, if a company manufactures anything, it is likely that they are using at least some CNC machine tools. With the diversity of companies and applications, there comes diversity with what is expected of CNC people. Understanding where your company fits in to this diverse group should help you understand what will be expected of you.

Companies that use CNC machining centers

There are many factors that contribute to how a CNC-using company applies its CNC machining centers. These factors include (among others) lot sizes, lead times, percentage of new jobs, size of tolerances held, materials machined, and company type. The most important of these factors is company type.

When it comes right down to it, there are only four types of companies that use CNC machine tools:

- ✍ **Product producing companies** – get revenue from the sale of a product
- ✍ **Workpiece producing companies** – (also called job-shops or contract-shops) get revenue from the sale of workpieces to product producing companies
- ✍ **Tooling producing companies** – get revenue from the sale of manufacturing tooling (fixtures, jigs, molds, dies, gauges, cutting tools, etc.) to product producing and workpiece producing companies
- ✍ **Prototype producing companies** – get revenue from the sale of prototypes to product producing companies

There are also overlaps in company type. For example, some product producing companies have a tool-room in which CNC machine tools are used – or they may have a research and development department that produces prototypes. Some workpiece producing or tooling producing companies have a product of their own.

While there will be exceptions to what we say here, some pretty good generalizations can be made based upon the company type alone, especially when it comes to what CNC people will be doing.

Product producing companies tend to have more resources than workpiece producing, tooling producing, and prototype producing companies. Since their profit is one step removed from manufacturing (a product won't come to market unless the company can make a profit), they tend to engineer all facets of the manufacturing environment. For this reason, they commonly break up the tasks related to CNC machine tool usage. People will specialize in the tasks they perform.

This will maximize machine tool utilization. It ensures that machines are running for as high a percentage of time as possible. Many CNC tasks will be performed while the machine is running production (like programming for upcoming jobs, gathering components needed for future setups, and assembling cutting tools, among others). So while a CNC operator is running a job on a CNC machine, other people are getting ready to run the next (and other upcoming) jobs. Again, this minimizes the amount of time that the CNC machining center is down between production runs.

On the other hand, workpiece producing, tooling producing, and prototype producing companies tend to charge an hourly rate for machine usage time. Their resources will be much smaller – and they will require

Lesson 4**Machine Configurations**

While you don't have to be a machine designer in order to run a CNC machining center, it helps to understand the configuration of the machine/s you'll be working with. At the very least, you must be able to recognize the major components and be able to identify the moving components (called axes).

Most beginners tend to be a little intimidated when they see a machining center in operation for the first time. Admittedly, there will be a number of new functions to learn. The first point to make is that you must not let the machine intimidate you. As you go along in this text, you will find that a machining center is very logical and is easy to understand with proper instruction.

As stated in Lesson Two, you can think of any CNC machine as being much like than the standard type of equipment it is replacing with very sophisticated and automatic motion control added. Instead of activating things manually by hand-wheels and manual labor, a *program* tells the machine what to do. Virtually anything that needs to be done on a machining center can be activated through a program.

There are two basic types of machining centers that we will be addressing in this text. They are vertical machining centers and horizontal machining centers. Because they are the more popular of the two, we'll be placing the emphasis on vertical machining centers. Let's start by describing the most common components of each.

Vertical machining centers

The classifications *vertical* and *horizontal* refer to spindle orientation. A vertical machining center has its spindle oriented in the vertical position (just as it is with the drill press and knee-style milling machine introduced in Lesson Two). The cutting tool will be pointing downward toward the machine's table. Indeed, a vertical machining center is such a popular type of CNC machining center because it closely resembles a knee mill – and again – a knee mill is a very popular conventional machine tool. For anyone having experience with a knee mill, a vertical machining center should be pretty familiar.

C-frame style

Figure 4.1 shows a C-frame style vertical machining center.

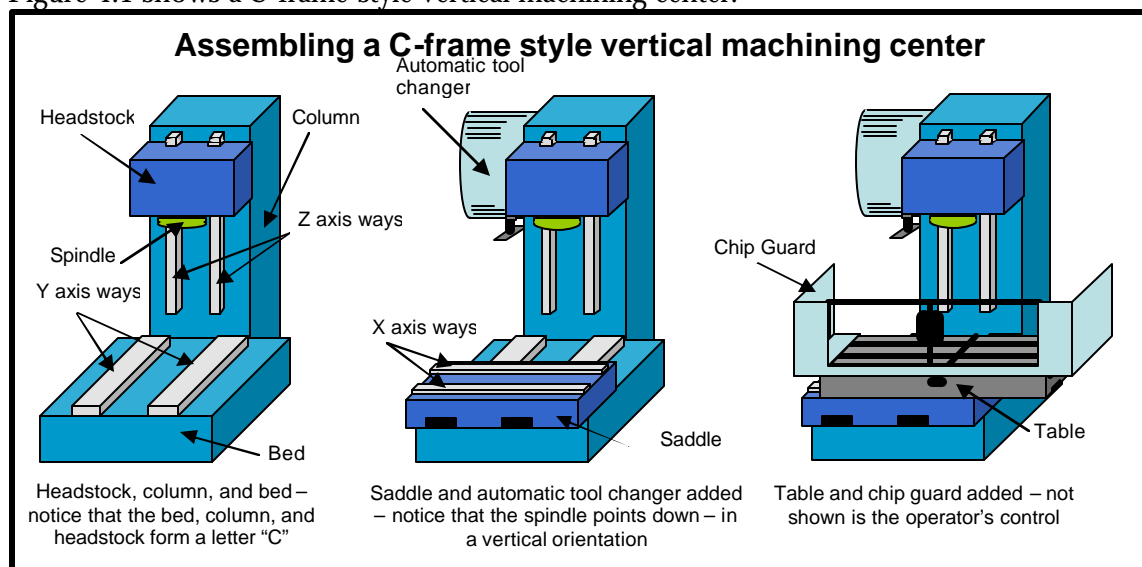


Figure 4.1 – Components that make-up a C-frame style vertical machining center

Lesson 5**Buttons And Switches On The Operation Panels**

While there are many buttons and switches on a CNC machining center, you must try to learn the reason why each one exists. If you don't, you may be overlooking a helpful – if not necessary – machine function. Worse, you may have a switch in the wrong position and not even know it.

You now know the configurations for the most popular types of CNC machining centers. You know the most basic components, the axes, and the programmable functions. However, we have not addressed some very important machine components – the operation panels. These panels provide the access you need to activate machine functions, and are the topic of this lesson.

*A proficient setup person or operator knows the function of **all** buttons and switches on their machine/s. While some may be seldom or never used, you must not consider yourself fully capable of running a machine until you know the function of all buttons and switches. If we don't cover a given button or switch in this text, ask someone in your company what it is – or look it up in the operation manual. Don't stop until you know why every button and switch is on your machining center.*

If you are just starting out, this may be quite a challenge. You will be striving just to learn the most important buttons and switches. Learning all buttons and switches may seem like too much to ask. While we sympathize, we cannot stress enough the importance of learning all of the access you have to activate machine functions. Again, don't stop until you know all the buttons and switches on your machine/s.

The two most important operation panels

Most machining centers have at least two distinct operation panels. We'll be calling them the *control panel* (designed by the control manufacturer) and the *machine panel* (designed by the machine tool builder).

For machining centers that have a Fanuc control (Fanuc is by far the most popular control in the industry), the control panel will be remarkably similar from one machining center to the next. Indeed there are few variations, even among different Fanuc control models.

But since the machine panel is designed by the machine tool builder, machining centers that have been manufactured by different machine tool builders will have substantially different machine panels. To compound this problem, machine tool builders can't seem to agree on the specific functions needed by CNC setup people and operators to run their machines. So one machine may have an important button or switch while another may not.

While we can be pretty specific about the function of buttons and switches found on the (Fanuc) control panel, we will be a little vague about machine panel buttons and switches. Also, there may be buttons and switches on your machining center's machine panel that we do not cover in this text. If you find one, be sure to ask an experienced person or reference the machine tool builder's operation manual to determine the function for the button or switch.

The control panel buttons and switches

Figure 5.1 shows the control panel for one popular control model. Again, this operation panel is made by the control manufacturer (Fanuc in our case). For the most part, it is used in conjunction with the display screen (on the left side of this control panel).

Notice the power buttons on the left side. The power-on button is usually the *second* switch used to power up the machine (the first is usually a main breaker switch that is placed behind the machine). The power-off button is used, of course, as part of the procedure to turn off the machine.

Lesson 6**The Key Operation Procedures**

Step-by-step procedures can keep you from having to memorize every function that you must perform on your CNC machining center. You will soon memorize procedures for task that you perform on a regular basis – but written procedures will always help you perform lesser used tasks.

While you have been introduced to the various buttons and switches on a typical machining center, it is unlikely that you have memorized them all. It is also unlikely that you know the appropriate order by which you should press buttons and activate switches to do anything on the machine.

For example, think of what it takes to perform one of the most basic tasks a setup person or operator must do – to power up the machine. Just because you know the related buttons and switches, does not mean you know the step-by-step *procedure* needed to turn on a CNC machining center. The procedure for a given machining center might look something like:

- ? Step 1: Turn on the main breaker (in back of the machine).
- ? Step 2: Press the power on button on the control panel.
- ? Step 3: Press the hydraulic system on button on the machine panel.
- ? Step 4: Follow the procedure to do a manual zero return.

With such a procedure, turning on the machine is not at all difficult. While you should still know the function of the related buttons and switches, this procedure provides you with sequential order (steps) you need. And similar procedures for other important tasks will be just as easy to follow (if you have them).

Admittedly, the most often used procedures will be soon memorized – and because most experienced setup people and operators have them memorized, they haven't written them down. And unfortunately, most companies don't have a written set of procedures for their CNC machines. Though this may be the case, nothing stops you from developing a written set of procedures for yourself. While you'll need the help of an experienced person, you can easily develop a set of procedures that will help you (and others) remember the steps necessary to do just about anything on the machine.

We divide the procedures needed to run a CNC machining center into five categories:

- ✍ Manual procedures
- ✍ Setup procedures
- ✍ Manual data input (MDI) procedures
- ✍ Program editing procedures
- ✍ Program operation procedures

Here are specific procedures that we recommend you write down in each category:

Manual procedures:

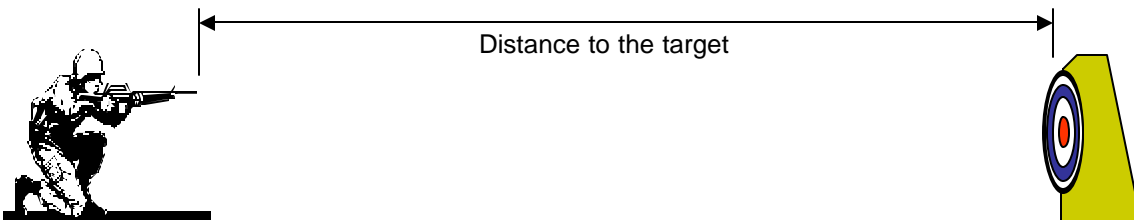
- To power up the machine
- To do a manual zero return
- To manually start the spindle
- To manually jog the axes
- To use the handwheel
- To manually load and remove cutting tools in the spindle

Lesson 7**Introduction To Compensation**

An airplane pilot must compensate for wind direction and velocity when setting a heading. A race-car driver must compensate for track conditions as they negotiate a turn. A marksman must compensate for the distance to the target when firing a rifle. And a CNC programmer must compensate for certain tooling-related concerns as programs are written. Setup people and operators must address these compensations – which in turn – will marry the CNC program to the physical setup that is being made.

What is compensation and why is it needed?

When you compensate for something, you are allowing for some unpredictable (or nearly unpredictable) variation. A *race car driver* must compensate for the condition of the race track before a turn can be negotiated. In this case, the variation is the condition of the track. An *airplane pilot* must compensate for the wind direction and velocity before a heading can be set. For them, wind direction and velocity are the variations. A *marksman* must compensate for the distance to the target before a shot can be fired – and the distance to the target is the variation. The marksman analogy is remarkably similar to what happens with most forms of CNC compensation. Let's take it further...



Before a marksman can fire a rifle, they must judge the distance to the target. If the target is judged to be one-hundred-fifty yards away, the sight on the rifle will be adjusted accordingly. When the marksman adjusts the sight, they are *compensating for the distance to the target*. But even after this preliminary adjustment and before the first shot is fired, the marksman cannot be *absolutely* sure that the sight is perfectly adjusted. If they've incorrectly judged the distance – or if some other variation (like wind) affects the adjustment – the first shot will not be perfectly in the center of the target.

After the first shot is fired, the marksman will know more. If the shot is not perfectly centered, another adjustment will be needed. And the second shot will be closer to the center of the target than the first. Depending upon the skill of the marksman, it might be necessary to repeat this process until the sight is perfectly adjusted.

With *all* forms of CNC compensation, the setup person will do their best to determine and enter certain *compensation values* needed to perfectly machine the workpiece (just as the marksman does their best in judging the distance to the target and adjusting the sight). But until machining actually occurs, the setup person cannot be absolutely sure that their initial compensation values are correct. After machining, they may find some imperfections in their initial entries. Or they may find that another variation (like tool pressure) is causing the initial adjustment to be incorrect. Depending upon the tolerances for the surfaces being machined, a second adjustment will probably be required. After this second adjustment, machining will be more precise.

Lesson 8

Fixture Offsets And Assigning Program Zero

The programmer will choose an origin for the program – which is called the program zero point. The use of a program zero point simplifies the task of programming; it is usually the setup person who must determine where the program zero point is located in the setup – and who must enter this position in fixture offsets.

You know that a CNC machining center has three linear axes: X, Y, and Z. It may additionally have one or two rotary axes. The CNC program tells the machine how to move these axes in a way that causes a cutting tool to machine the workpiece. To this end, the programmer specifies a series of *coordinates* through which the tool will move. Coordinates are simply positions specified within the program.

Think of the global positioning satellite (GPS) system in an automobile. It works by monitoring the car’s current position on the earth. And this position is specified with coordinates – using longitude and latitude.

With CNC machines, coordinates are specified in each axis to cause a positioning movement. Consider these X and Y coordinates:

X0.5 Y0.5

These coordinates may be specifying the position at which a hole must be drilled – and they send the machine to a position of 0.5 in each axis (X and Y). When this command is completed, the center of the spindle will be positioned directly above the hole-location. But in order to cause such a motion, the machine must know the *origin* for the coordinates – that is, the location from which the coordinates are taken. In CNC terms, we call this origin location the *program zero point*.

How is the program zero point determined?

It is the programmer who determines the position of the program zero point – and most programmers use a logical method for deciding where it is placed. The method is based upon how the blueprint is dimensioned. They will pick the location in each direction (axis) from which the dimensions begin. Consider the drawing shown in Figure 8.1 that shows an example of how program zero is determined in the X and Y axes.

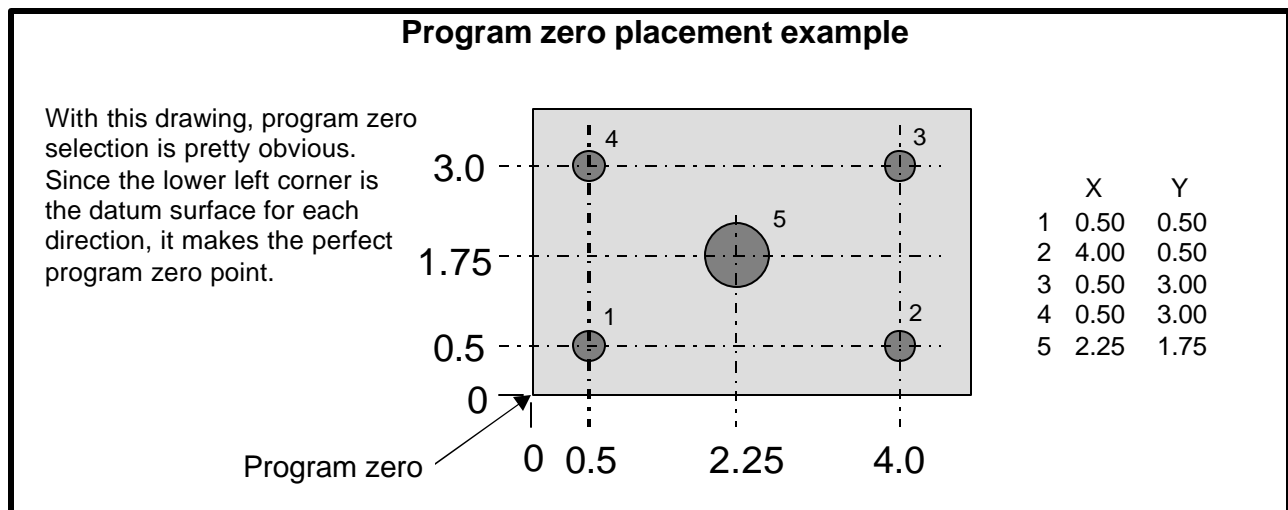


Figure 8.1 – An example of program zero point placement in the X and Y axes

Lesson 9**Tool Length Compensation**

Tool length compensation allows a programmer to ignore the precise length of each tool as a program is written. Each tool's length will be determined during (or just prior to) setup. The setup person is often responsible for measuring each tool's length and entering the length value into the machine's offset table. The operator must be able to repeat this process when dull cutting tools are replaced. Additionally, tool length compensation allows setup people and operators to make sizing adjustments to control the precise depth to which cutting tools machine.

You know from Lesson Seven that tool length compensation is one of the three compensation types – and that it is used for every cutting tool in every job. You also know that the tool length compensation value for each tool is placed in a tool offset. And you know that tool length compensation will allow you to make sizing adjustments to control the depth to which a cutting tool will machine. In this lesson, we will be taking an even closer look at this very important CNC feature.

The reasons why tool length compensation is needed

Cutting tools used on machining centers differ from one to the next. For one thing, and as you know from Lesson Two, there are a variety of cutting tool types that are used on machining centers, including (among others) center drills, spot drills, drills, taps, reamers, boring bars, end mills, and face mills. Each type of cutting tool requires its own type of tool holder. Most straight-shank tools use a collet holder. Others (like end mills) use a tool holder having a set-screw to clamp the cutting tool in place. Yet others (like face mills and taps) require a very special style of tool holder – designed specifically for the cutting tool type.

No two tools will have exactly the same length

Given the assortment of cutting tools available for use on CNC machining centers, it is unlikely that any two tools used by a program will be exactly the same length. And as the program is written, the programmer will not know the precise length of any given tool. Figure 9.1 illustrates this point with five different tools.

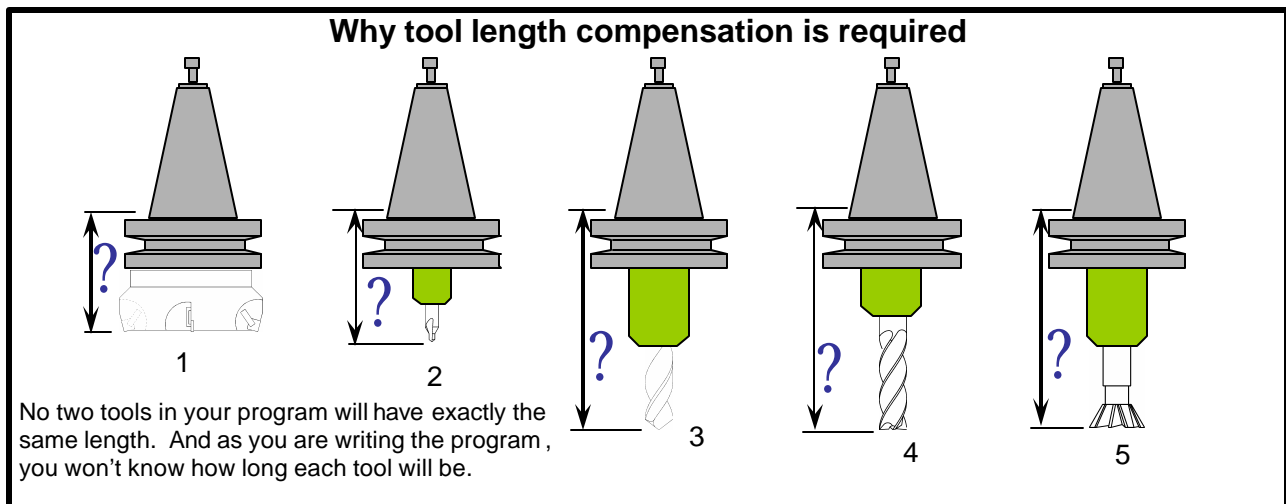


Figure 9.1 – Five cutting tools that might be used in by a CNC machining center program

The most important reason for tool length compensation has nothing to do with setup and operation. Tool length compensation will allow a programmer to write programs even though they don't know how long the cutting tools will be. But there are other reasons why tool length compensation is required – and most do apply to setup people and operators.

Lesson 10**Cutter Radius Compensation**

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.

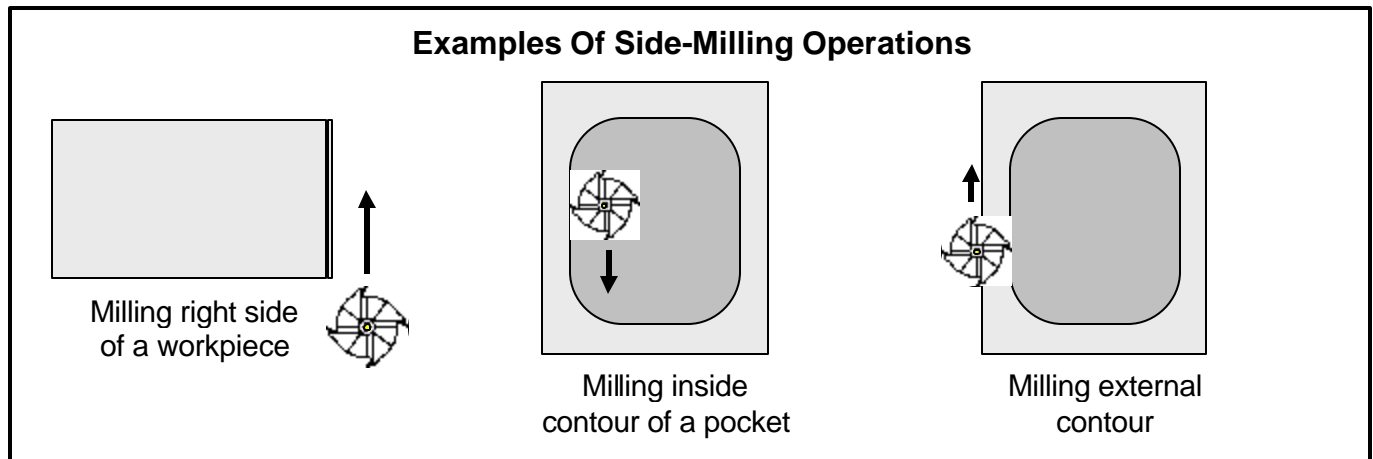


Figure 10.1 – Side-milling with an end mill

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.

Lesson 11**Tasks Required To Setup A CNC Machining Center**

We define setup time as the total time a machine is down between production runs. We define cycle time as the total time it takes to complete a production run divided by the number of good workpieces produced. If you think about it, there are only two general tasks that occur on CNC machining centers – machines are either in setup or they are running production. In this lesson, we will address those things that occur while the machine is down between production runs.

It is important to understand the distinction between making a setup and running production. The tasks you perform during setup are *getting the machine ready* to run production. Only when the setup is completed and after a workpiece has passed inspection is it possible to run production. (Note that that we include the running of the first good workpiece – and its inspection – as part of setup.) The person making the setup is called the *setup person* – the person completing the production run is called the *CNC operator*. In many companies – especially many tooling- and workpiece-producing companies – the same person makes the setup *and* completes the production run.

Here is a general list of tasks that are related to setup. We will be presenting them in detail during this lesson. Note the order we use to list the tasks. This is the approximate order by which a setup is made (though each setup person will have his or her own way of doing things).

Setup-related tasks

- ✍ Tear down previous setup and put everything away
- ✍ Gather the components needed to make the setup
- ✍ Make the workholding setup
- ✍ Assign program zero point/s
- ✍ Assemble cutting tools
- ✍ Measure tool length and cutter radius compensation values
- ✍ Enter tool length and cutter radius compensation values
- ✍ Load cutting tools into the machine's automatic tool changer
- ✍ Load the CNC program/s
- ✍ Verify the correctness of the program and setup
- ✍ Cautiously run the first workpiece – ensure that it passes inspection
- ✍ If necessary, optimize the program for better efficiency (new programs only)
- ✍ If changes to the program have been made, save the corrected version of the program

Some of these tasks should sound familiar. We have been addressing bits and pieces of what is required to make setups as part of our building-blocks approach throughout this text. Many of these general tasks require more specific tasks to be done. *Assign program zero point/s*, for example, will often require the setup person to first measure the program zero assignment values and then enter them into the appropriate fixture offset. In this lesson, we intend to *put-it-all-together*, providing you with a logical flow for making setups. Some of the material in this lesson has been presented before – especially material that is related to the more complicated topics. Consider repeated material as review. It should help to solidify your understanding.

Lesson 12**Tasks Required To Complete A Production Run**

Once a job is set up and the first good workpiece is efficiently machined, the rest of the workpieces must be run. Completing a production run is the job of a CNC operator – though the same person who sets up the machine is often the person who completes the production run. A misconception exists about the difficulties related to completing a production run. Many manufacturing people feel that running out a job simply requires part loading, cycle activation, and workpiece removal. In reality, there is usually a lot more to it.

With an understanding of what it takes to set up a CNC machining center, let's turn our attention to what it takes to run the rest of the workpieces in the job. Here is a list of the related tasks:

Done during every cycle:

- ✍ Load a workpiece
- ✍ Activate the cycle
- ✍ Monitor the cycle to ensure that cutting tools are machining properly (first few workpieces only)
- ✍ Remove the workpiece
- ✍ Clean/de-burr the workpiece
- ✍ Perform specified measurements (if required)
- ✍ Report measurement results to statistical process control (SPC) system

Not required in every cycle:

- ✍ Make sizing adjustments for critical workpiece attributes
- ✍ Replace worn tools
- ✍ Remove chips from work area (if required)
- ✍ Machine maintenance

At first glance, the tasks related to completing a production run probably look pretty simple. And as long as everything is going smoothly, keeping the machine running good workpieces may be as simple as loading the workpiece, pressing the cycle start button, and removing the completed workpiece when the cycle is completed. This may be the case for the first few workpieces machined in a proven job (one run before).

It is quite common, however, that operators must make adjustments during the production run to ensure that workpiece are produced consistently. The more workpieces in the production run, the tighter the tolerances, and the more abrasive the workpiece material, the more likely it will be that adjustments must be made. The frequency of these adjustments (how often they must be made) is also related to these factors.

In this lesson, we will be taking a close look at what it takes to complete a production run once the setup is completed. As we did in Lesson Eleven, we will present the related tasks in the approximate order that a production run is completed.

Remember from Lesson Eleven that there are only two things that occur on a CNC machining center. The machine is either in setup or running production. In this lesson, of course, we will be addressing thing that occur during the production run. And this includes just about anything that happens between setups.