Key Concept	*Know the Machining Center from a Programmer's Viewpoint*
1	*You must come to know a CNC turning center from two distinctly different perspectives. In Key Concept 1, we look at the machine from a programmer's viewpoint. Later, during Key Concept 7, we will look at the machine from a setup person's or operator's viewpoint.*

Key Concept 1 is the longest of the Key Concepts. It contains several lessons:

- 1.1: Machine configurations
- 1.2: Turning center feeds and speeds
- 1.3: CNC work flow
- 1.4: Visualizing program execution
- 1.5: Workpiece coordinate system
- 1.6: Understanding geometry offset and work shift values
- 1.7: Entering geometry offsets
- 1.8: Introduction to programming words

A CNC programmer need not be as intimate with a specific CNC turning center as a setup person or operator–but they must understand enough about the machine to:

- ✓ Create programs
- ✓ Instruct setup people and operators
- ✓ Create related setup and production documentation.

First and foremost, a CNC programmer must understand what the CNC turning center is designed to do. That is, they must:

- ✓ Understand the machining operations a turning center can perform
- ✓ Be able to develop a workable process (sequence of machining operations)
- ✓ Select appropriate cutting tools for each machining operation
- ✓ Determine cutting conditions (feeds and speeds) for each cutting tool
- ✓ Design a workholding and cutting tool setup

All of these skills, of course, are related to basic machining practice–which as we state in the preface–are beyond the scope of this course. For the most part, we'll be assuming you already possess these important skills. That said, we do include some important information about the machining operations that can be performed on CNC turning centers throughout this course. For example, we discuss how to develop tool paths for machining operations in lessons 2.1 and 3.1. We provide a description of rough and finish turning and boring in lesson 6.2. Threading is discussed in lesson 6.4. And in general, we provide suggestions about how machining operations can be programmed when it is appropriate. This information should be adequate to help you understand enough about machining operations to begin working with CNC turning centers.

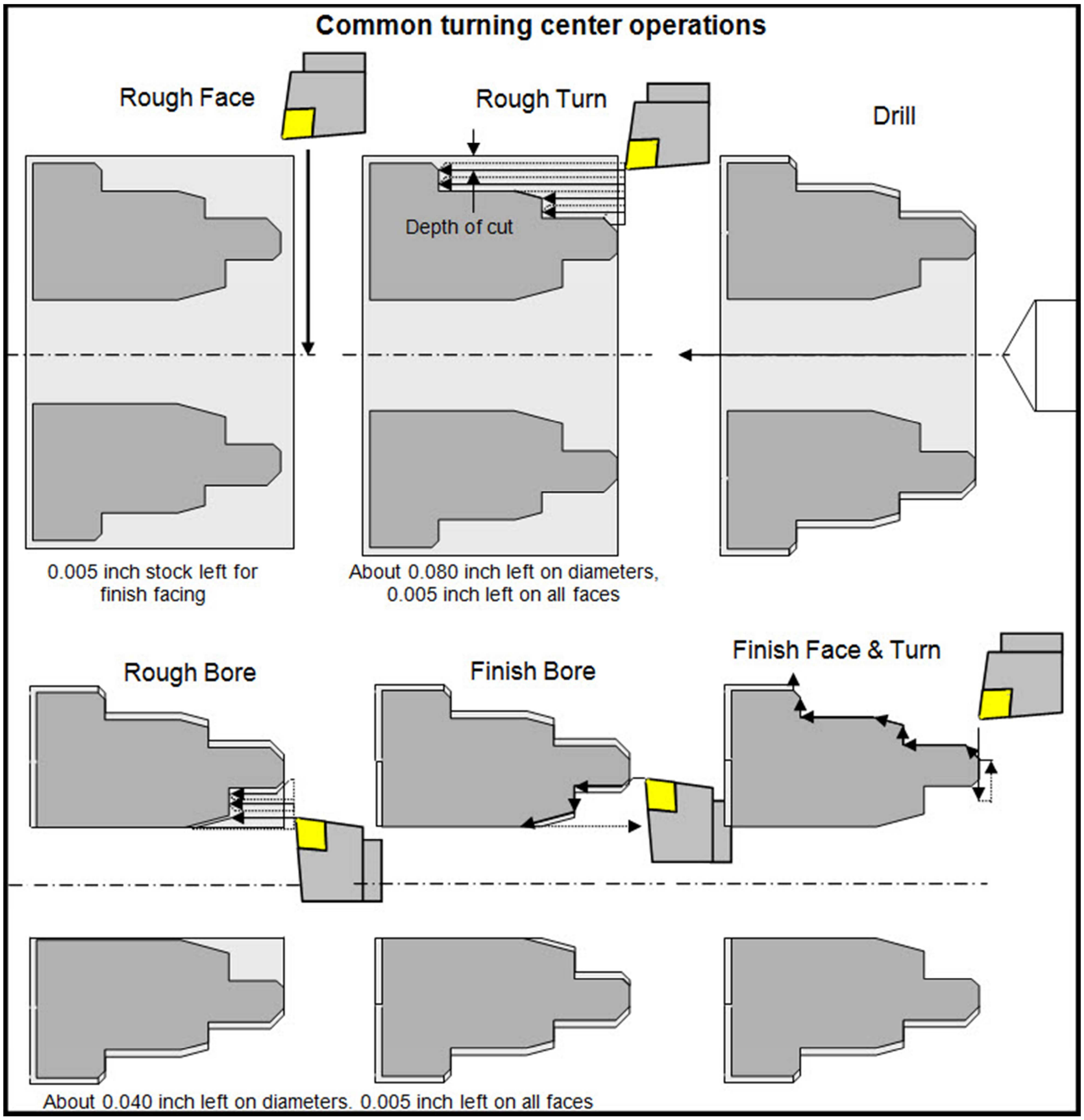
Experienced with Conventional (non-CNC) Machine Tools?

A CNC turning center can be compared to an engine lathe (or any kind of conventional lathe). Many of the same operations performed on a conventional lathe are performed on a CNC turning center. If you have experience with manually operated lathes, you already have a good foundation on which to build your knowledge of CNC turning centers.

This is why machinists make good CNC programmers. With a good understanding of basic machining practice, you can easily learn to program CNC equipment. You already know what you want the CNC machine to do. It is a relatively simple matter of learning how to tell the CNC machine to do it.

If you have experience with machining operations like rough and finish turning, rough and finish facing, drilling, rough and finish boring, necking and threading, and if you understand the processing of turned workpieces, you are well on your way to understanding how to program a CNC turning center. Your previous experience has prepared you for learning to program a CNC turning center.

We can also compare the importance of knowing basic machining practice in order to write CNC programs to how important it is for a speaker to be well versed with the topic they will be presenting. If not well versed with their topic, the speaker will not make much sense during the presentation. In the same way, a CNC turning center programmer who is not well versed in basic machining practice will not be able to prepare a program that makes any sense to experienced machinists.



Lesson 1.1 - Machine Configurations

As a programmer, you must understand the characteristics of a CNC turning center. You must be able to identify its basic components – you must understand the moving components of the machine (called axes)–and you must know the various functions of your machine that are programmable.

Objectives

After completing this lesson, students should be able to:

- ✓ Describe several types of turning centers
- ✓ Describe the basic components of a turning center
- ✓ Describe the primary moving axes and their polarity
- ✓ Describe the basic programmable functions of the spindle
- ✓ Describe the basic functions of the turret
- ✓ Describe the basic programmable functions of a turning center

Introduction

Beginners can be a little intimidated when they see a turning 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 course, you will find that a turning center is very logical and easy to understand with proper instruction.

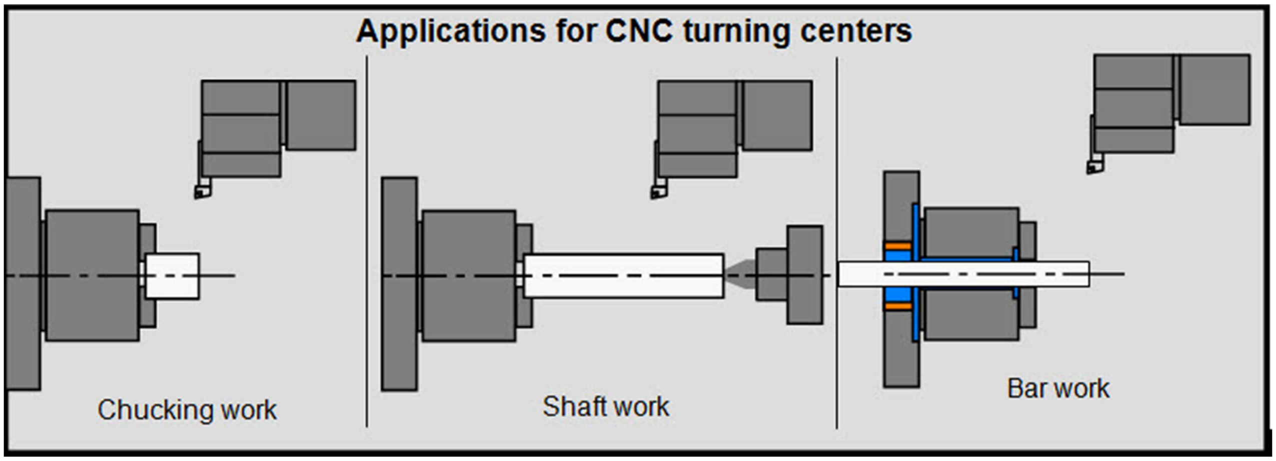
You can think of any CNC machine as being nothing more than the standard manual machine it is replacing with motion control and other machine functions automated. Instead of moving the axes manually using hand-wheels and selecting or activating functions using switches and levers, you will be preparing a program that tells the machine what to do. Virtually anything that needs to be done on a true turning center can be controlled in a program–meaning anything you need the machine to do can be commanded in a program.

Types of CNC turning centers

There are several types of CNC turning centers. While at first glance there may appear to be substantial differences among the various types, all turning centers share several commonalities. We'll begin by describing the most popular type of CNC turning center – the *universal style slant bed turning center*. Because it is so popular, this is the machine type we will use for all examples in this text. We will then introduce several other types of turning centers, comparing them to the universal style slant bed turning center.

Universal style slant bed turning center

This style of turning center is called a *universal style* turning center because it can perform all three forms of turning applications – chucking work, shaft work, and bar work. This explains why it is the most popular type of turning center – it provides the most flexibility to CNC turning center users.



When raw material comes to the machine in the form of short slugs (like round bars cut to length), the application is called chucking (or chucker) work. The raw material is secured solely by the workholding device (commonly a three-jaw chuck).

With longer slugs (longer than about three to four times the raw material diameter), the workholding device by itself will not be sufficient to secure the workpiece for machining. For these applications, some form of work support device/s must be used (commonly a tailstock and/or steady-rest). This application is called shaft work.

With bar work, the raw material comes to the turning center in the form of a long bar (from four to fifteen feet long [1.2-5 meters], depending upon the type of bar feeder being used). Bar work requires a special bar support and feeding device (called a bar feeder). The bar is fed through the headstock and spindle into the working area. A workpiece is machined and cut off from the bar. The bar is then fed again for another workpiece to be machined.

Figure 1.1 shows a universal style slant bed turning center. The headstock houses a spindle to which the workholding device is mounted. Our illustration shows a three-jaw chuck, but other types of workholding devices can be used (collet chuck, expanding mandrel, etc.). To the right of the workholding device is the tailstock, which is used to support the right end of long workpieces – again, for shaft work. The turret of the turning center is used to hold cutting tools and it can be quickly rotated from one tool station to another. Current turning centers have turrets that hold from six to twelve cutting tools.

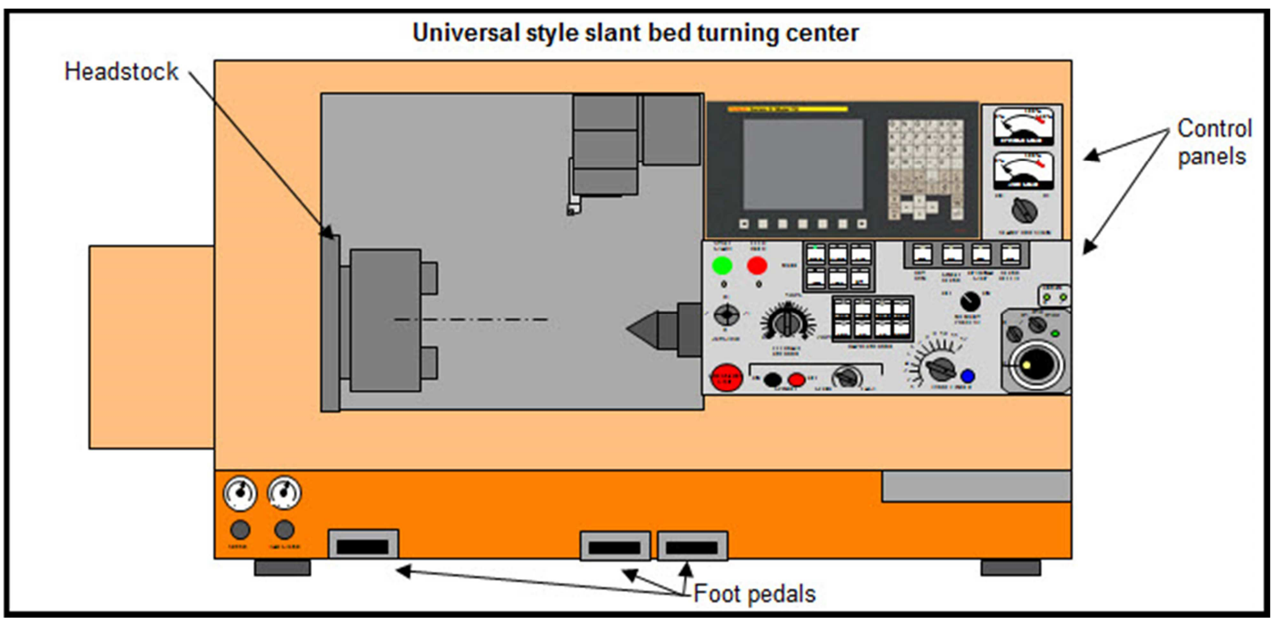


Figure 1.1 – A universal style slant bed turning center with its door removed

Directions of motion (axes) for a universal style slant bed turning center
All turning centers have at least two linear *axes* of motion. The turret (and cutting tool) will move along with these two axes. By *linear*, we mean the axis moves along a straight line.

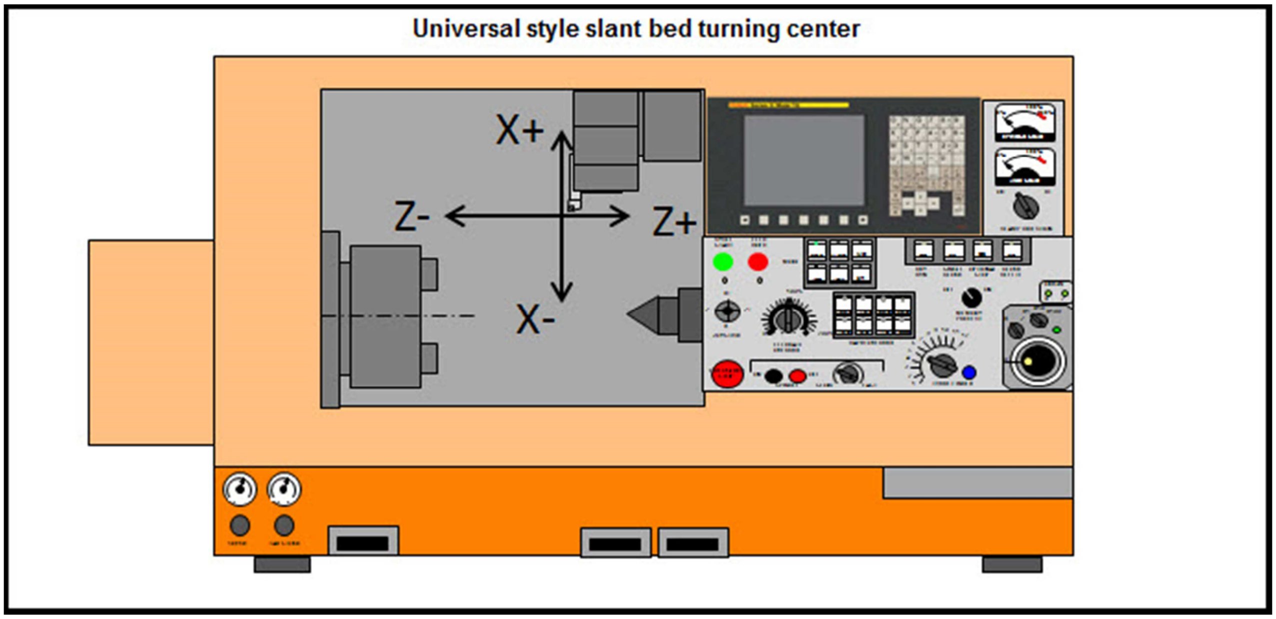


Figure 1.2 – The two most basic axes of motion for a CNC turning center

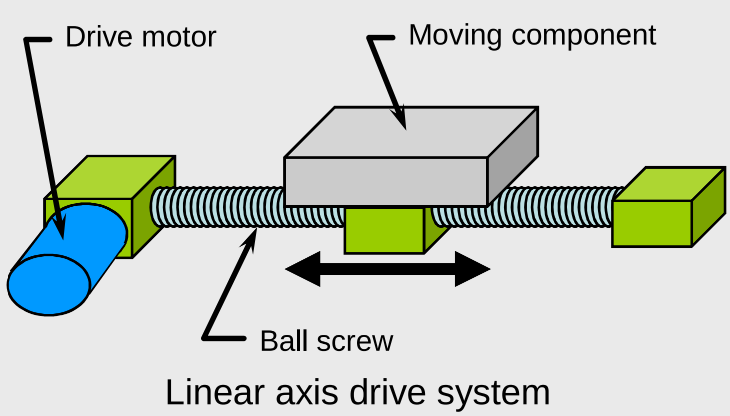
The *diameter-controlling* axis (up/down motion of the turret as shown in Figure 1.2) is the *X-axis*. The *length-controlling* axis (right/left motion of the turret as shown in Figure 1.2) is the *Z-axis*. Figure 1.2 shows these directions of motion along with the polarity (+/-) for each.

These two most basic directions of motion will remain exactly the same for almost all types of turning centers (only a handful of turning center manufacturers stray from what we show in Figure 1.2.) *The X-axis will always be the diameter-controlling axis* – and X-minus is always the direction that causes the cutting tool to move to a smaller diameter (toward the spindle centerline). *The Z-axis will always be the length controlling axis* – and Z-minus will always be the direction that causes the cutting tool to move toward the workholding device.

X is specified in diameter
Though we may be a little ahead of ourselves, the X-axis is designated in *diameter* for almost all turning centers. That is, if a diameter of 3.0 inches must be machined, the designation for the X-axis will be X3.0. There are some (especially older) turning centers that require the X-axis to be specified with radial values. For these machines, the word X1.5 will cause the tool to be positioned to a 3.0 inch diameter. Note that it is much easier to work with a turning center when the X-axis if it is designated in diameter – which is why most current model turning centers do so.

What's in a linear axis?

As you know, the X- and Z-axes are *linear axes*. They cause motion along a straight line. Though you don't have to understand every detail of how an axis works, it helps to understand the basic components. A *drive motor* is connected to a *ball screw* that drives the *moving component* for the axis (turret in our case). The action is not unlike what occurs with a common table vise. As you turn the screw on the table vise, it causes the jaw on the vise to move along a straight line – clamping a component in the vise.



Live tooling for a universal style slant bed turning center
We have just described the most basic form of a universal style slant bed turning center. Again, this machine has two axes (X and Z) – and it can perform all three kinds of turning work (chucking work, shaft

work, and bar work). The majority of universal style slant bed turning centers that are in use today are of this configuration.

There is, however, a special accessory called live tooling that can be equipped on all types of CNC turning centers (including the universal style slant bed turning center). This accessory, which is becoming quite popular, makes it possible for a turning center to perform machining operations that are more commonly associated with CNC machining centers (or milling machines).

These operations include drilling, tapping, reaming, and milling (among others). In essence, turning centers equipped with live tooling can perform both turning center operations and machining center operations – giving them the ability to more completely machine a workpiece. For many applications, this eliminates the need to perform secondary operations on another machine tool. Figure 1.3 shows a workpiece that requires live tooling if it is to be completely machined on a turning center.

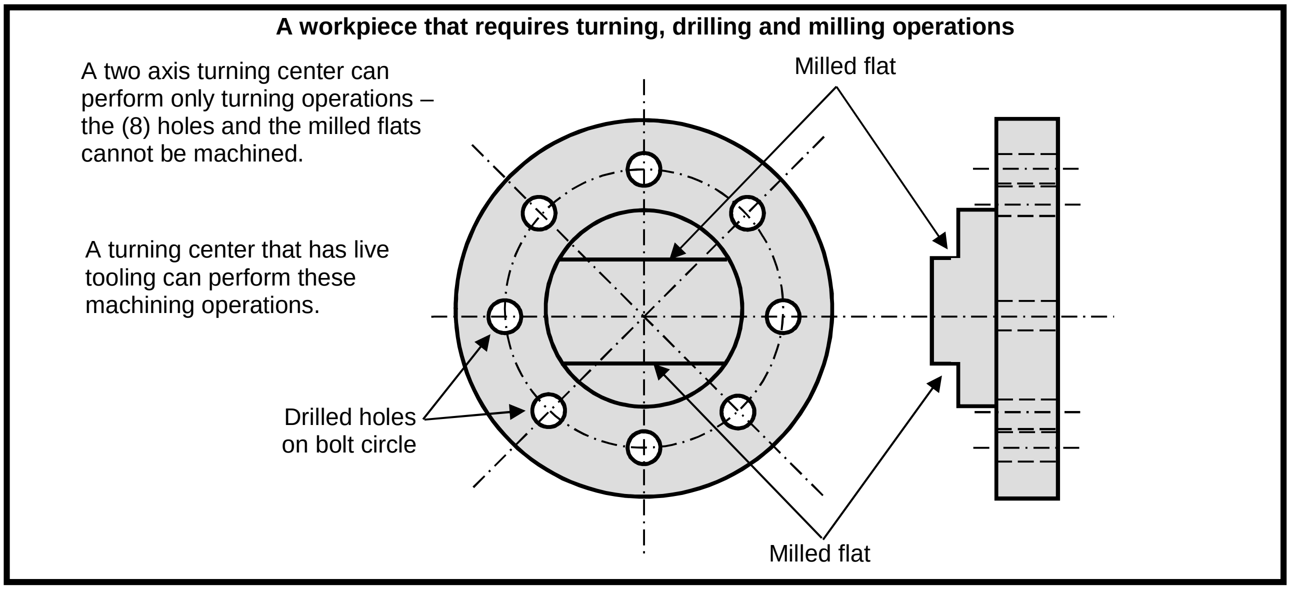


Figure 1.3 – A workpiece that requires live tooling

Turning centers that are equipped with live tooling have two additional features. First, as the name implies, they have a special device mounted within the turret that makes it possible to rotate cutting tools (again, like drills and end mills). Second, they have a special rotary axis (called the C-axis) built into the spindle drive system. Figure 1.4 shows a universal style slant bed turning center that has live tooling. These turning centers are sometimes referred to as mill/turn machines.

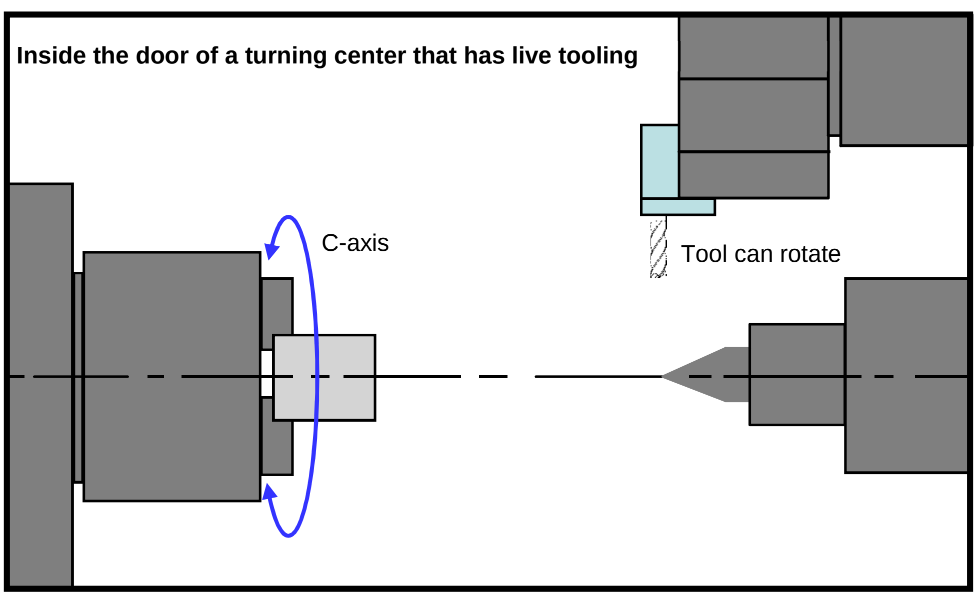


Figure 1.4 – Live tooling on a universal style slant bed turning center

The cutting tool depicted in Figure 1.4 points along the X-axis. Another style of tool holder is usually available that allows the cutting tool to point along the Z-axis (toward the chuck).

When the programmer selects the *live tooling* mode, the rotary axis within the spindle is engaged and the spindle within the turret will be used to rotate cutting tools. When the programmer selects the *normal turning mode*, these devices are disengaged.

The bulk of this text describes the programming of the normal turning mode (not using live tooling). We describe live tooling in detail in the Appendix after lesson 6.7.

Other types of CNC turning centers

The machine types we show from this point have a great deal in common with universal style slant bed turning centers. Commonalities include:

- **Axis direction** – X is always the diameter-controlling axis (toward and away from the spindle centerline). Z is always the length-controlling axis (toward and away from the workholding device).
- **X-axis polarity** – X-minus is always the direction toward the spindle centerline (getting smaller in diameter. Note: there are two machine builders that we know of that reverse X-axis polarity for some of the machines they have produced (X-minus is away from the spindle centerline).
- **Z-axis polarity** – Z-minus is always the direction toward the workholding device.
- **X is almost always specified in diameter** – If you need to machine a 4.0 inch diameter, the word X4.0 will be used. There are a few (especially older) machines that require X-positions to be specified as a radial value (to turn a 4.0 inch diameter the word X2.0 will be used).
- **Turret/tool moves in each axis** – With one exception (the Swiss-type turning center), the turret (and cutting tool) will move in each axis. That is, the workpiece will remain stationary in X and Z while cutting tools will move to perform machining operations.
- **Live tooling** – All forms of CNC turning centers can be equipped with live tooling (as an optional feature).

Chucking style slant bed turning center

Figure 1.5 shows another common form of CNC turning center. Notice that the only difference between this machine and the universal style is the absence of the tailstock. This, of course, means that the machine cannot perform shaft work – it is limited to chucker and bar work. These machines usually have a limited Z-axis stroke length.

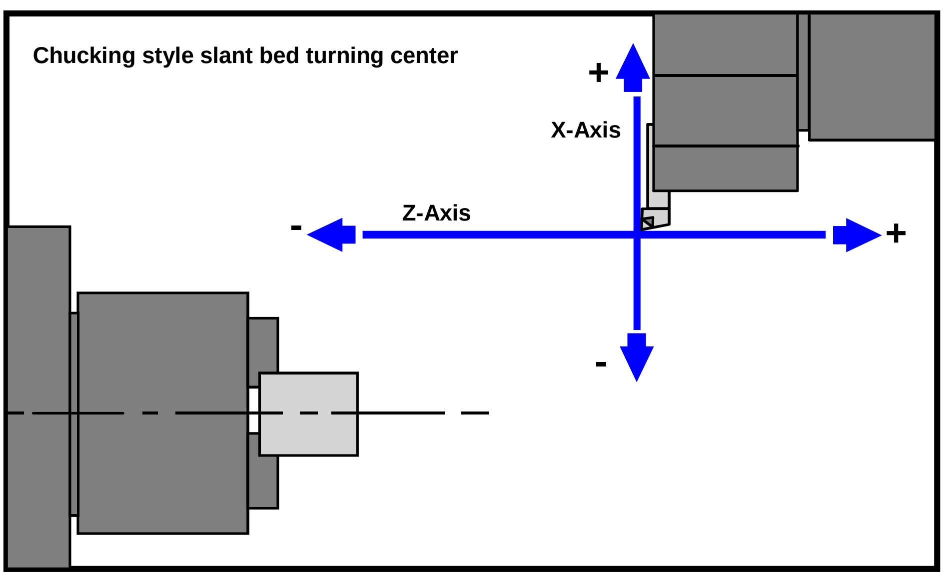


Figure 1.5 – Chucking style slant bed turning center (axis polarity reflects turret motion)

Twin spindle horizontal bed turning centers

This is a form of turning center that is quite popular. Its popularity stems from the fact that two cycles can be running at the same time. This effectively doubles productivity – in essence, it is like having two machines in one – but it takes less floor space and it is more convenient for one operator to run.

Though there are exceptions, these machines are not equipped with tailstocks, meaning they are limited to chucker and bar applications. Figure 1.6 shows the configuration.

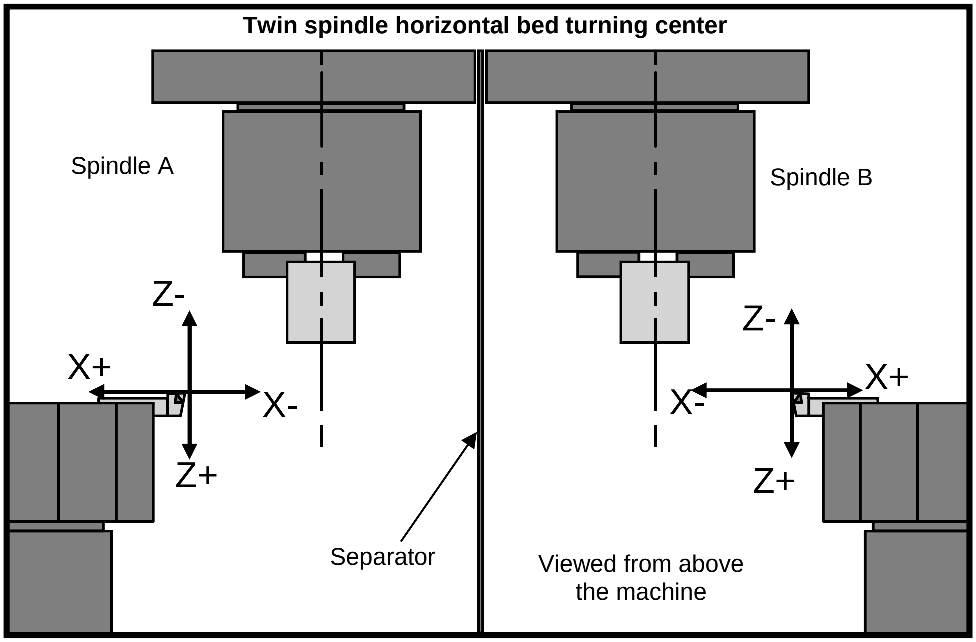


Figure 1.6 – Twin spindle horizontal bed turning center (axis polarity reflects turret motion)

Again, this kind of turning center is like having two machines in one. Each spindle is totally independent, meaning two different programs can be run at the same time. Examples of applications for this machine include running two identical operations, running first and second operation for a workpiece, and running two different workpieces. One or both spindles can even be used for bar work.

While this machine may look radically different from slant bed turning centers, notice that our given commonalities still apply. X is still the diameter controlling axis and X-minus is still the direction going toward the spindle centerline (for both spindles). Z is still the length controlling axis and Z-minus is the direction going toward the workholding device. And X is still specified in diameter. A program written for a chucking style or slant bed turning center could be run on either spindle of this machine.

Sub-spindle style turning centers

This form of two-spindle turning center has the spindles positioned in such a way that they face one another. Figure 1.7 shows one.

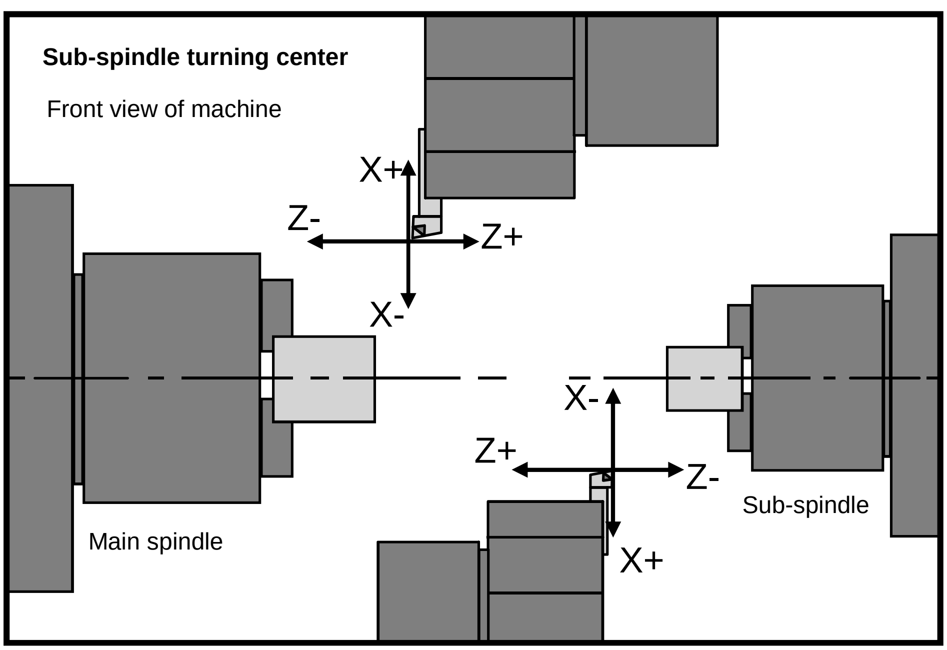


Figure 1.7 – A sub-spindle turning center

This design provides an advantage over the twin-spindle horizontal bed turning center just shown. The advantage has to do with transferring a workpiece from one spindle to the other. With many sub-spindle turning centers, the sub-spindle can move forward to (automatically) take a workpiece from the main spindle. This makes it possible to easily perform machining operations on the second end of the workpiece in the sub-spindle. Figure 1.8 shows how the transfer can take place.

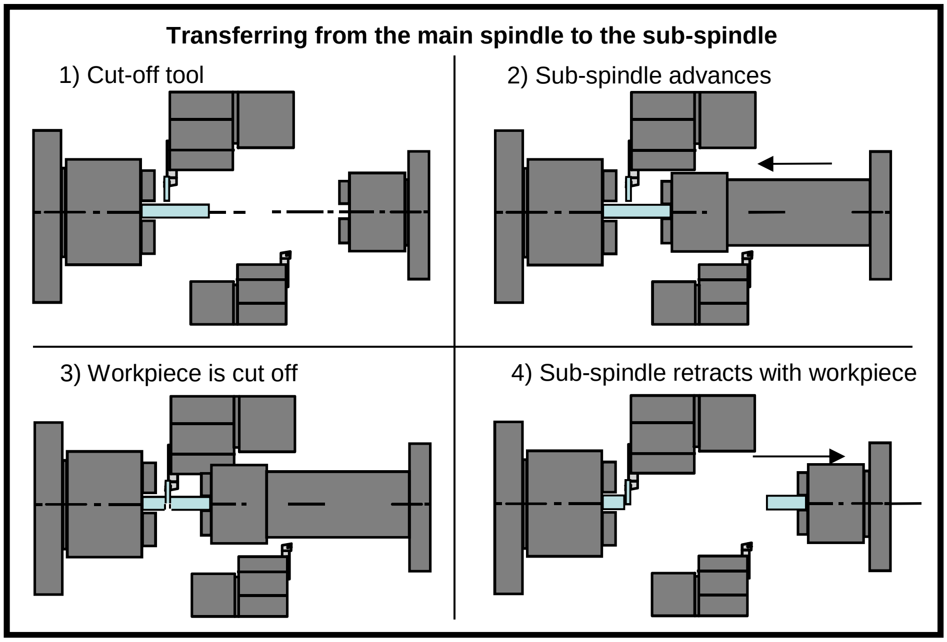


Figure 1.8 – Transferring a workpiece from the main spindle to the sub-spindle

In this application, the main spindle is usually being fed by a bar feeder (a bar application). After the bar feed, the main spindle machines the right side of the workpiece. When finished, the sub-spindle will advance to hold and support the workpiece during the cutoff operation (the sub-spindle runs at the same speed as the main spindle during the cutoff operation). After the cutoff, the sub-spindle retracts. The main spindle will feed another bar for the right side operation while the sub-spindle machines the left side of the previous workpiece. In this way, one completed workpiece is produced per cycle.

Vertical single spindle turning centers

This kind of turning center is commonly used to machine large diameter (but relatively short), heavy workpieces. When a workpiece is held in a horizontal orientation (as is the case with all turning centers discussed to this point), the weight of the workpiece will work against the work holding device (chuck). That is, workpiece weight will have a tendency to cause the workpiece to fall out of the chuck – making for unstable workholding. The heavier the workpiece, the more difficult it is to adequately hold it in a horizontal orientation.

While horizontal orientation may be fine for light workpieces, as the weight of the workpiece increases, it may become difficult (if not impossible) to properly hold and support the workpiece held in this manner. With a *vertical* turning center, the weight of the workpiece actually *helps* to stabilize it in the work holding device.

Figure 1.9 shows an example of a single spindle vertical turning center.

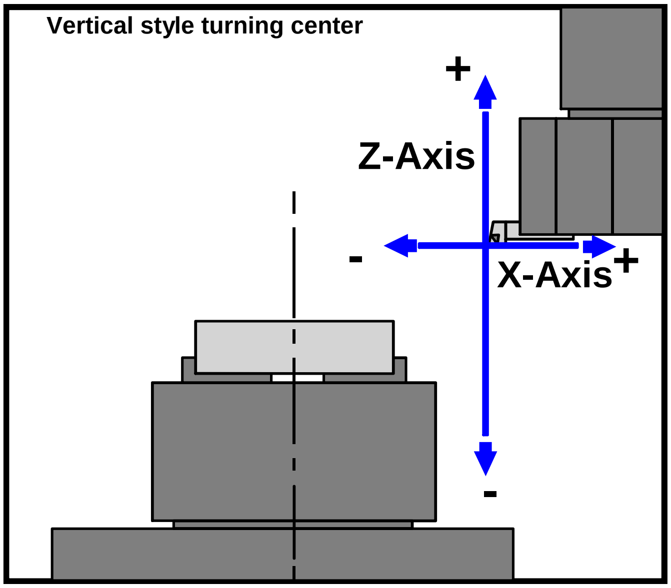


Figure 1.9 – Single spindle vertical turning center (axis polarity reflects turret motion)

Since workpieces to be run on this kind of machine are usually quite short, tailstocks are seldom equipped with this kind of turning center. Also, since the spindle is mounted vertically, this type of turning center cannot be used as a bar feed machine. Its only feasible application is chucking work.

Twin spindle vertical turning centers

The twin spindle vertical turning center has the same advantages as the twin spindle horizontal turning center discussed earlier. Figure 1.10 shows a drawing of one.

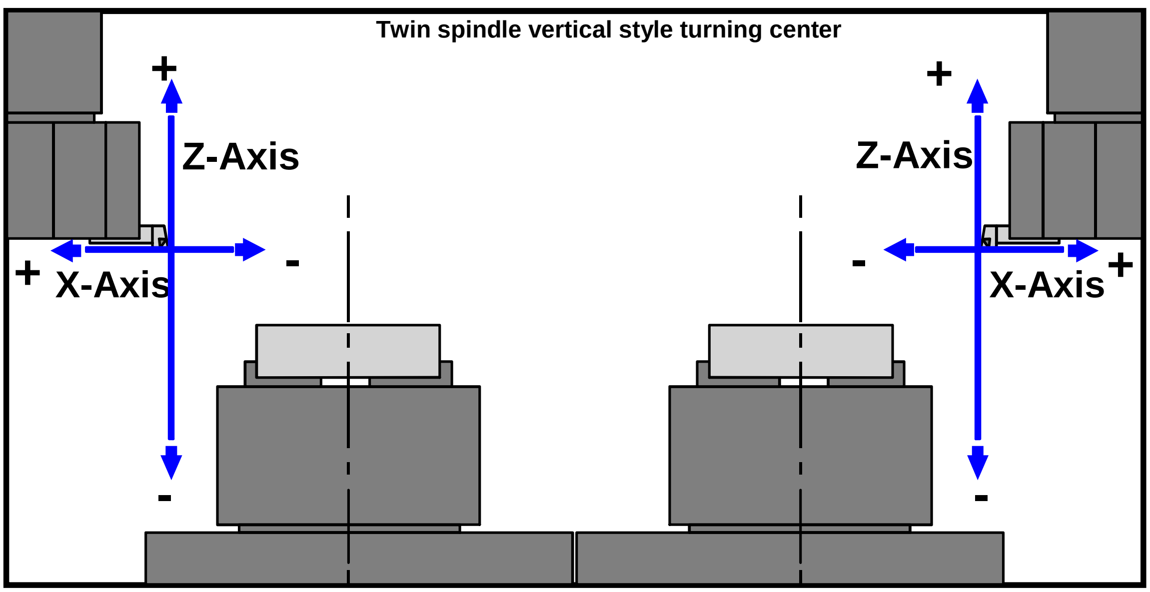


Figure 1.10 – Twin spindle vertical turning center (axis polarity reflects turret motion)

Again, think of this machine as like having two machines in one. A program written for one spindle can be run in the other.

Gang style turning centers

This form of turning center does not have a turret to hold cutting tools. Instead, the cutting tools are mounted on a table or *sub-plate*. While tool interference can sometimes be a problem, the advantage with this style of turning center is that there will be virtually no tool changing time. Figure 1.11 shows an example of this type of turning center.

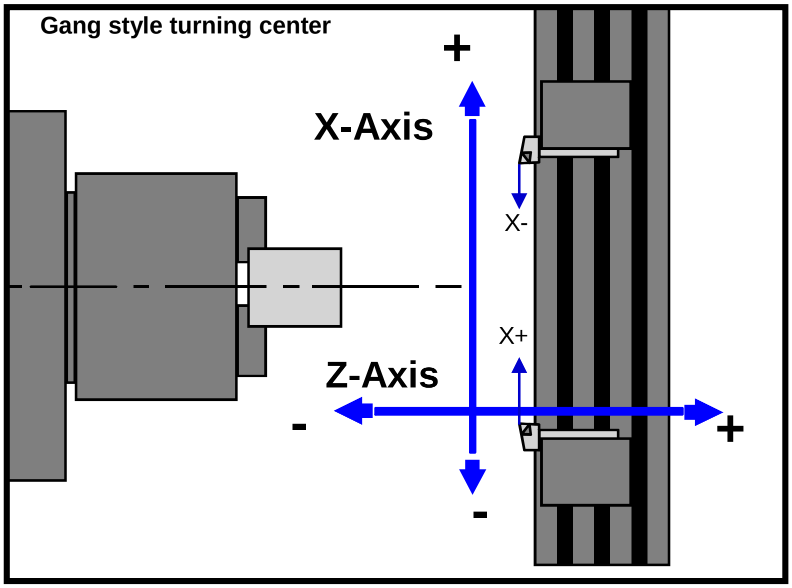


Figure 1.11 – Gang style turning center (axis polarity reflects tooling-table motion)

A gang style turning center is usually quite small (under 10 hp spindle motor) and is commonly used for bar work. Of course, the basic design of this machine eliminates the possibility for a tailstock, so shaft work cannot be done.

Notice that the entire table moves to form the X- and Z-axes. With X, this can be a little troublesome. For tools mounted on the far side of the spindle centerline, X+ will be as it is for all types of turning centers we have introduced (X+ is away from spindle centerline). But for tools on the near side of the spindle center, X+ is the direction toward the spindle center.

Swiss-type CNC turning centers (also called sliding headstock turning centers)
This machine is a little different. The tool (turret) only moves in the X-axis for this type of machine. The tool remains stationary in the Z-axis. Instead, the *workpiece* moves in the Z-axis. Figure 1.12 shows one.

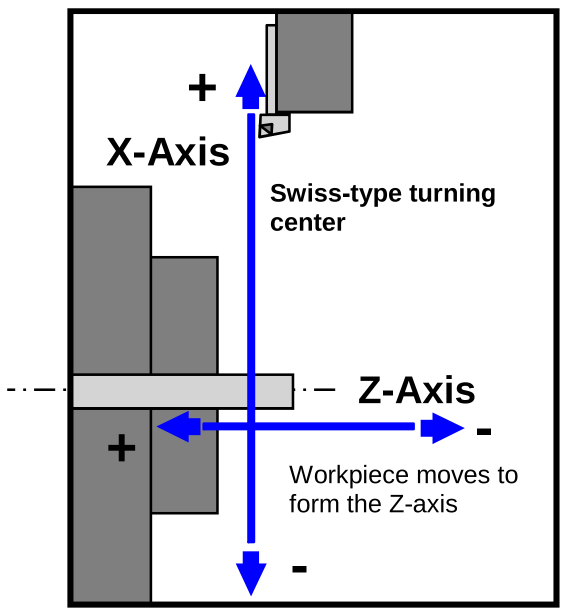


Figure 1.12 – Swiss type CNC turning center

While at first glance the Z-axis may appear to be reversed as compared to the machines we've shown so far, remember that it is the workpiece that is moving in the Z-axis. This kind of machine always uses a bar feeder. Since machining will always occur very close to the workholding device, machining will be very well supported. This allows the machining of very small diameter, yet quite long workpieces. (Think of a needle valve.)

A reminder about this text – Once again, we will be using the universal style slant bed turning center for all example programs shown in this text. When appropriate, we will point out special considerations for the other types of turning centers we have just shown.

Programmable functions of turning centers

A true CNC turning center will allow you to control just about any of its functions from within a program. There should be very little operator intervention during a CNC cycle. Here we list some common functions that can be programmed on all *true* turning centers. While we do show the related CNC words used to command these functions, our intention here is not to teach programming commands (yet). It is to simply make you aware of the kinds of things a programmer can control through a program.

Spindle

The spindle of all turning centers can be programmed in at least three ways, activation (start/stop), direction (forward/reverse), and speed (in either surface feet/meters per minute or revolutions per minute). Many turning centers additionally provide multiple power ranges (like the transmission of an automobile).

Spindle speed

You can precisely control how fast the spindle of a turning center rotates. An *S-word* is used for this purpose. There are two ways to specify spindle speed. When the spindle is in rpm mode, an S-word of S500 specifies a speed of 500 revolutions per minute (rpm). When the spindle is in *constant surface speed mode*, an S-word of S500 specifies a speed of 500 surface feet per minute (sfm), assuming you are working in the inch measurement system. (If you work in the Metric measurement system, S500 will specify 500 meters per minute when in constant surface speed mode.)

We will describe the two spindle speed modes – as well as how to determine how and when to use them – in lesson 1.2.

Spindle activation and direction

You can also control which direction the spindle rotates – *forward or reverse*. The forward direction is used for right hand tooling (when machining occurs toward the workholding device). It will appear as counter-clockwise when viewed from in front of the machine. The reverse direction is used for left hand tooling and will appear as clockwise when viewed from in front of the spindle.

Three *M-codes* control spindle activation. M03 turns the spindle on in the forward direction (used with right-hand tools). M04 turns the spindle on in a reverse direction (for left-hand tools). M05 turns the spindle off.

What is an M-code?

M-codes control many of the programmable functions of a turning center. In many cases, you can think of them as being like programmable on/off switches. "M" stands for "miscellaneous" or "machine" function.

M-codes are created by the machine tool builder, and will often vary from one turning center to another. Here we show some common M-codes, but you must look in your machine tool builder's programming manual to find the complete list of M-codes for a given turning center.

These M-codes don't vary:

M00: Program stop

M01: Optional stop

M03: Spindle on (forward)

M04: Spindle on (reverse)

M05: Spindle off

M08: Flood coolant on

M09: Coolant off

M30: End of program

These M-codes vary:

M___: Low spindle range
M___: High spindle range

M___: Tailstock quill forward

M___: Tailstock quill reverse

M___: Automatic door open

M___: Automatic door close

M___: Chip conveyor on

M___: Chip conveyor off

M___: ___________________

M___: ___________________

M___: ___________________
M___: ___________________

M___: ___________________

Spindle range

Many, especially larger turning centers, have two or more spindle ranges. Spindle ranges are like the gears in an automobile transmission. Generally speaking, lower ranges are used for power – higher ranges are used for speed. With most turning centers, spindle range selection is done with M-codes.

While the specific M-code numbers for spindle range selection will vary from one machine tool builder to another, many turning center use M41 to select the low range and M42 to select the high range. We'll use these two M-codes (M41: low and M42: high) to specify spindle range selection throughout this text.

Turning centers vary when it comes to what will actually happen when the spindle range is changed. Some, especially older machines use a mechanical gearbox that must be engaged for the low range. These machines commonly require that the spindle be stopped during the range change. While the spindle stoppage, range change, and spindle restart will occur automatically, these machines can take from three to ten seconds or more to change ranges.

Some, especially newer machines have spindle motors with multiple windings. Two or more sets of windings within the motor itself control range selection. With these machines, range changing is almost instantaneous – and the spindle does not have to stop when the spindle range is changed.

It is important to know the power characteristics for the turning center/s you will be working with in order to make the correct spindle range selection for a given machining operation. Every turning center manufacturer will provide a power curve chart like the one shown in Figure 1.13 to document a machine's spindle power characteristics. You will normally find this power curve chart in the machine tool builder's operation or programming manual.

Power curve chart for a turning center having two spindle ranges

Low range: 0 – 1,500 rpm

High range: 0 – 3500 rpm

M41: Selects the low range

M42: Selects high range

Notice that the high range completely overlaps the low range – this will be the case with most turning centers

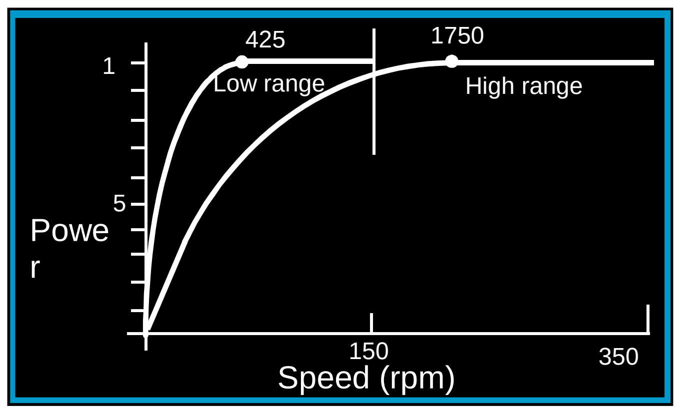


Figure 1.13 – An example spindle power curve chart

With this example spindle power curve chart, notice that the low range runs from zero to fifteen hundred rpm and achieves full power at 425 rpm. The high range runs from zero to thirty-five hundred rpm, completely overlapping the low range. Full power is achieved in the high range at 1,750 rpm.

Again, knowing your machine's spindle power characteristics is important for selecting the appropriate spindle range. We give a general rule-of-thumb for spindle range selection: *perform roughing operations in the low range and finishing operations in the high range*.

While this is a good rule of thumb, there are times when it isn't correct. Consider, for example, machining a steel workpiece that is less than 1.0 inch in diameter. Say the cutting tool manufacturer recommends a speed of 600 sfm for the rough turning operation. The formula to calculate rpm is as follows:

Rpm = 3.82 times speed in sfm divided by the diameter to be machined

In this case, the *slowest* speed needed for the roughing operation will be 2,292 rpm (3.82 times 600 sfm divided by 1.0). In this case, since full power is achieved at 1,750 rpm in the high range, and since the low range will peak out at 1,500 rpm – slowing the machining operation – the high range should be selected for this operation.

Feedrate

You know that all turning centers have at least two linear axes, X and Z. You also know that the cutting tool (for most turning centers) moves along with these two axes. It is the motion of the cutting tool while it is in contact with the workpiece that causes machining to occur. It is important that the motion *rate* (how quickly the tool moves) be appropriate to the machining operation being performed. In CNC turning center terms, this motion rate is called *feedrate*.

An F-word is used to specify feedrate. And like spindle speed, feedrate can be specified in two ways. It can be specified in *per minute* fashion or in *per revolution* fashion. As the names imply, when feedrate is specified in per minute fashion, it specifies how far the cutting tool will move during one minute. When feedrate is specified in per revolution fashion, it specifies how far the cutting tool will move during one spindle revolution.

Also as with spindle speed (at least in constant surface speed mode), feedrate specification is related to the measurement system you use. In the inch mode, feedrate is specified in either inches per minute (ipm) or inches per revolution (ipr). In metric mode, feedrate is specified in either millimeters per minute (mmpm) or millimeters per revolution (mmpr). Feedrate selection is discussed in much greater detail in lesson 1.2. For now, we'll simply introduce the related words and give a few examples.

- F-word – Feedrate specification
- G20 – Inch mode
- G21 – Metric mode
- G98 – Feed per minute mode
- G99 – Feed per revolution mode

Here are a few examples of feedrate specification:

N010 G20 G98 F4.0 (4.0 inches per minute)

N020 G20 G99 F0.015 (0.015 inches per revolution)

N030 G21 G98 F100.0 (100.0 millimeters per minute)

N040 G21 G99 F0.5 (0.5 millimeters per revolution)

What is a G code?

G codes are called *preparatory functions*. They prepare the machine for what is coming up – in the current command and possibly in up-coming commands. In many cases, they set *modes,* meaning once a G code is *instated* it will remain in effect until the mode is changed or cancelled.

Here we list a few common G codes, but don't worry if they don't make much sense yet. Upcoming discussions will clarify.

Common G codes:

G00: Rapid motion	G42: Tnr comp. right
G01: Straight line motion	G50: Spindle limiter
G02: Circular motion (CW)	G70: Finishing cycle
G03: Circular motion (CCW)	G71: Rough turning cycle
G04: Dwell	G72: Rough facing cycle
G20: Inch mode selection	G76: Threading cycle
G21: Metric mode selection	G96: CSS mode
G28: Zero return command	G97: RPM mode
G40: Cancel tool nose radius comp.	G98: Feed per minute
G41: Tool nose radius comp. left	G99: Feed per revolution

Look in your control manufacturer's manual for a complete list of G codes.

Turret indexing (tool changing)

With the exception of gang style turning centers, all turning center introduced in this lesson have a turret into which cutting tools are placed (twin spindle turning centers have two turrets). Specific turret design will vary from one machine tool builder to another. All turning centers (even gang style turning centers) must provide a way to specify cutting tool selection. Figure 1.14 shows the turret of a typical turning center.

Turret of a turning typical center

Figure 1.14 – A typical turning center turret

The turret shown in Figure 1.14 is currently holding turning tools (external machining tools) as well as boring tools (internal machining tools). Notice that the internal tools protrude quite a distance from the turret face (they stick out). This can cause interference problems with a large workpiece and/or the workholding device. You must always be concerned with the potential for interference problems as you choose the turret stations into which you place cutting tools.

With many turrets, for example, you cannot place a long drill or boring bar in a station that is adjacent to a turning tool that machines a small diameter (like a facing tool) without interference problems. As the turning tool faces a workpiece to center, the adjacent boring bar will be driven into the chuck or workpiece.

Turret station and offset selection

A T-word specifies which cutting tool will be used. For turning centers that have a turret, the T-word will actually cause the turret to index to the specified turret station. But there's a little more to the T-word than turret index.

For most machines, the T-word is a four-digit word. The first two digits specify the turret station and *geometry offset* to be used with the tool (geometry offsets assign *program zero* – and will be discussed in lessons 1.7 and 4.2). The second two digits of the T-word specify the *wear offset* to be used with the tool (wear offsets allow the operator to make minor adjustments – and will be discussed during lesson 4.3).

The command

N020 T0404

will cause these three things to occur:

- the turret to index to station number four (first two digits)
- geometry offset number four will be selected (first two digits)
- wear offset number four will be selected (second two digits)

Almost all current model turning centers have *bi-directional turrets*. That is, the turret can rotate in either direction. When a T-word is given, most machines will cause the turret to automatically rotate in a direction that that provides the shortest rotational distance to the specified tool

With gang style turning centers, of course, there is no turret to index. Only two things will happen with the previous command: geometry and wear offset number four will be selected.

Again, offset use is the topic of future lessons. For now, just remember that most programmers will make the wear offset number the same number as the turret station number and geometry offset number.

Coolant

All turning centers allow programmable control of *flood coolant*. Coolant is commonly used to cool the workpiece during machining and to lubricate the machining operation. Two M-codes are used to control coolant. Almost all turning center manufacturers use M08 to turn on flood coolant and M09 to turn it off.

Other possible programmable functions

The programmable functions introduced to this point are available for all current model turning centers. Those we list from this point are related only to certain machine types – or they are optional functions that are not supplied with all turning centers.

Tailstock

Turning centers that can perform shaft work (like universal style slant bed turning centers) are equipped with a tailstock. The tailstock is used to support the right end of a long workpiece during machining (the end opposite the workholding device). Most machinists would agree that when the length of the workpiece exceeds about three to four times its diameter, a tailstock should be used to provide support during machining.

Though most current model turning centers have *programmable* tailstocks, machine tool builders vary with how they cause the tailstock body and quill to move. Many use a series of M-codes. M16 and M17 may be used to cause quill movement forward and reverse while M28 and M29 may be used to cause tailstock body movement forward and reverse.

Some machine tool builders actually cause tailstock motion by engaging the turret to the tailstock and pulling it along with the Z-axis. If your machine has a programmable tailstock, you must reference your machine tool builder’s programming manual to determine how it is programmed.

Programmable steady rest

Like a tailstock, this programmable function can be equipped with turning centers that perform shaft work. For extremely long workpieces, the tailstock by itself may not provide enough support. The chuck may provide ample support for one end and the tailstock for the other, but the workpiece will be unsupported in the middle. A programmable steady rest will provide support anywhere it is required over the length of the workpiece. This accessory is discussed in the Appendix (after lesson 6.7).

Bar feeders and chuck activation

In bar applications, the raw material comes in the form of a long bar (as long as fifteen feet). The bar is usually relatively small in diameter (under 2.0 inches or so). Most bar feeders do not require anything in the way of special programming commands since they apply a constant pressure to drive the bar through the spindle. Most bar fed turning centers use the collet chuck (or chuck jaw) activation to control the bar advance. When the chuck is opened, the bar automatically advances. Closing the chuck will secure the bar in place for machining. Two M-codes control the chuck open and close process, but machine tool builders vary when it comes to *which* M-code numbers they use. M15 might be used to open the chuck while M14 may be used to close it. Again, you’ll have to reference your machine tool builder’s programming manual to determine the M-code numbers for chuck open/close.

Most machine tool builders recommend using a turret station to hold a bar-stop that will restrain the bar during its advance. To cause a bar advance, first the bar-stop is brought to within a small distance of the existing bar end (within about 0.100 inch). Next the chuck is opened (with M15 for instance) and the bar advances a small amount to contact the bar stop. The bar stop is then moved in the Z-plus direction to advance the bar the appropriate amount (part length plus cut off amount plus facing stock amount). Finally, the chuck is closed (with M14 for example) and the bar stop is retracted. We explain the programming of bar feed applications in detail in the Appendix (after lesson 6.7).

Part catcher

In bar applications, a workpiece is cut off after every cycle. During the cut-of machining operation, a part catcher is commonly used to keep the workpiece from falling into the chip pan after being parted from the bar. A part catcher is commanded to swing into position just prior to cut-off operation. As the workpiece is cut off, it falls gently into the part catcher and exits the machining area. The part catcher is then retracted. Two M-codes are commonly used to activate the part catcher, but again, machine tool builders vary with regard to *which* M-codes they use. M18 may be used to advance the part catcher, M19 to retract it. If your turning center has a part catcher, reference your machine tool builder’s programming manual to determine the related M-codes.

Tool touch off probe

While this device may not be programmable, it is an extremely helpful accessory – one that is being equipped with more and more turning centers. While we won’t be assuming that your turning center has one, we will include detailed information about how it is used. Once you understand how it can help CNC setup people and operators, you’ll want to have one for all of your turning centers.

A tool touch off probe will help during the initial setup of a job – and during long production runs. During initial setup, it can dramatically facilitate the task of program zero assignment (discussed in lessons 1.6 and 1.7). The setup person will first (manually) swing the tool touch off probe into its active position. They will then bring each cutting tool into contact with a stylus on the probe (actually twice). This will automatically align the cutting tool with the program zero point (again, program zero is discussed in lessons 1.5, 1.6. and 1.7).

During the production run, the tool touch off probe will help during dull tool replacement. The same techniques used during setup will be repeated when a dull tool is replaced.

Unfortunately, many existing turning centers are not equipped with a tool touch off probe, meaning you will have to learn other methods of performing the tasks they accomplish. Rest assured that we'll show alternatives to using a tool touch off probe in this text. Once you see both ways, you'll probably want all of your *future* turning centers equipped with tool touch off probes.

Automatic tool changing systems

As stated earlier, the turret of most turning centers can hold a maximum of twelve tools. This may not be enough tool stations for some applications – especially when the CNC user wants to keep all cutting tools in the machine – or when the machine is equipped with live tooling. For this reason, more and more machine tool builders are equipping their turning centers with an automatic tool changing device (similar to one found on a CNC *machining* center). This device automatically loads cutting tools into the turret of the turning center, minimizing tool loading time during setups while increasing the number of cutting tools that can be used by a program.

Exceptions to X-axis

We mentioned this earlier, but we want to elaborate. Almost all turning centers manufacturers orient the X-axis as we have shown – the X-minus direction is turret motion toward the center of the spindle (getting smaller in diameter). However, there are at least two turning center manufacturers we know of (Mori Seiki and Nakamura Tome) that sometimes reverse X-axis polarity. With these rather unique machines, X-plus is motion as the turret moves *toward* the spindle center.

A quick fix

If you must work on a turning center that has the X-axis reversed, it can be quite confusing, especially if your company also owns other turning centers with the X-axis as we have shown. You can easily reverse the direction of the X-axis by turning on a feature called *X-axis mirror image* (a standard feature with most turning centers). By turning on X-axis mirror image for those machines with the X-axis reversed, you will make all machines in your shop consistent with one another – at least when it comes to X-axis polarity.

If you do not turn on X-axis mirror image for turning centers with which the X-axis is reversed, almost all X coordinates must be designated with a minus sign (-). Also, circular motion words (G02 and G03 introduced in lesson 3.1) must be reversed – as must tool nose radius compensation commands (G41 and G42 discussed in lesson 4.4). Additionally, tool offset direction in the X-axis will also be reversed. These inconsistencies make it quite difficult for people to work with both types of machines (with and without the X-axis reversed).

Gang style turning centers with cutting tools on both sides of the spindle centerline

As discussed, a tooling table is used to hold cutting tools with this kind of machine. And cutting tools can be placed on the front or back side of the spindle centerline. For tools on the front side, X motion toward the spindle centerline is reversed (again, X-plus is toward the spindle center).

This causes the same inconsistency just described. That is, the X-axis polarity for any tool on the front side of the spindle centerline is reversed. Again, this causes the features circular motion (G02 and G03), tool nose radius compensation (G41 and G42), and all tool offsets to also be reversed. Additionally, most X coordinates for front-side tools must be programmed with X-minus coordinates.

Again, if the feature *programmable mirror image for the X-axis* is used (a standard feature with most gang type turning centers), the back turret and the front turret can be programmed in exactly the same manner. Simply turn on mirror image in the X-axis before programming any motions for cutting tools on the front side of the spindle and turn it off before commanding motions for cutting tools on the back side of the spindle center.

Two G-codes control turn programmable X-axis mirror image on an off. Unfortunately, the specific G-codes vary from one control model to another. With one popular control, G68 turns it on and G69 turns it off. You must reference your control manufacturer's programming manual to find the related G-codes.

Center cutting axis

Some turning centers are equipped with an axis of motion that can only move along the spindle's centerline (parallel to the Z-axis). The machine tool builder *Miyano* calls this axis the *B axis*. The purpose for this axis is to perform hole machining operations that are right on the spindle centerline. Operations like drilling, reaming, and tapping can be performed. Also, this motion can even be programmed during the activation of the other two axes (X and Z), meaning simultaneous machining can be accomplished which will minimize cycle time.

What else might be programmable?

While this text will acquaint you with the most common programmable functions of turning centers, you must be prepared for more. Other programmable devices that may be equipped on your turning center include chip conveyer, automatic door open and close, automatic loading devices, and a variety of other application-based accessories. If you have any of these functions, you must reference your machine tool builder's programming manual to learn how these special features are programmed.

> Check with an experienced person in your company or school to find out what other programmable features are available on the turning center you will be working with.

Key points for lesson 1.1:

- ✓ There are several types of CNC turning centers.
- ✓ With all types of CNC turning centers, X is the diameter controlling axis and X-minus is the toward spindle center, Z is the length controlling axis and Z-minus is toward the workholding device.
- ✓ X-axis positions are specified in diameter.
- ✓ Some turning centers – those that have live tooling – can additionally perform machining-center-like machining operations).
- ✓ You must understand the functions of your turning center that can be programmed.
- ✓ Spindle can be controlled in at least three ways (activation, direction, and speed). Additionally, many turning centers have more than one spindle range.
- ✓ Spindle speed can be specified in rpm or in surface feet/meters per minute.
- ✓ Feedrate specifies the motion rate for machining operations.
- ✓ Feedrate can be specified in per revolution fashion or per minute fashion.
- ✓ Coolant can be activated to allow cooling and lubricating of the machining operation.
- ✓ Most turning centers have a turret in which cutting tools are placed.
- ✓ Machines in the United States allow the use of inch or metric mode.
- ✓ You must determine what else is programmable on your turning center/s.

Quiz

1) Which style of turning center is the most popular?

A) Universal slant bed style C) Twin spindle

B) Chucker style D) Gang style

2) Specify the correct axis letter for each moving component below (universal style slant bed turning center).

a) ____Diameter controlling axis

b) ____Length controlling axis

c) ____Rotation of the spindle (live tooling machines only)

3) Specify the correct axis letter and polarity for each tool motion below.

a) _____Tool motion bigger in diameter

b) _____Tool motion toward the chuck

Answers: 1: A, 2a: X, 2b: Z, 2c: C, 3a: X+, 3b: Z-, 4a: G20 G99 F0.015, 4b: G21 G98 F50.0, 4c: M05, 4d: M08, 4e: M09, 4f: T0707, 4g: G20

4) Provide the CNC word or command needed to activate the following:

a) Feedrate of 0.015 ipr: ____________

b) Feedrate of 50.0 mmpm: ____________

c) Stop spindle: ____________

d) Turn on the flood coolant: ____________

e) Turn off the coolant: _____________

f) Index to station number seven: ____________

g) Select inch mode: ____________

Talk with experienced people in your company to learn more: Do your turning centers have more than one spindle range? If so, what are the speeds for each range? At what rpm does the spindle achieve maximum horsepower in each range? Do any of your machines have automatic doors? If so, what are the related M-codes? How many cutting tools can your turning centers hold?

