

Introduce course content, Key Concepts approach, and Key Concept number one. 15 minutes lecture time

1: Know your machine from a programmer's viewpoint

- 1: Machine configurations
- 2: General flow of the CNC process
- 3: Visualizing program execution
- 4: Program zero and the rectangular coordinate system
- 5: Determining program zero assignment values
- 6: Assigning program zero
- 7: Introduction to programming words

2: You must prepare to write programs

- 8: Preparation steps for programming

3: Understand the motion types

- 9: Programming the three most basic motion types

4: Know the compensation types

- 10: Introduction to compensation
- 11: Tool length compensation
- 12: Cutter radius compensation
- 13: Fixture offsets

5: You must provide structure to your CNC programs

- 14: Introduction to program structure
- 15: Four types of program format

6: Special features that help with programming

- 16: Hole-machining canned cycles
- 17: Working with subprograms
- 18: Other special programming features
- 19: Programming rotary devices

7: Know your machine from a setup person or operator's viewpoint

- 20: Tasks related to setup and running production
- 21: Buttons and switches on the operation panels

8: Know the three basic modes of operation

- 22: The three modes of operation

9: Understand the importance of procedures

- 23: The key operation procedures

10: You must know how to safely verify programs

- 24: Program verification

Need help getting prepared to present a lesson?

- The CD-ROM disks include two sets of presentations – one with **audio guidance** and the other without. Audio guidance is included on key slides (commonly transition slides) to help you understand what you're supposed to do. Slides that include audio guidance include a teacher icon in the lower-right corner. While running the slide show, click this icon to hear the guidance.

In the course outline to the left, the Key Concepts are shown in bold. The lessons included in each Key Concept are shown as well. As you can see, there are ten Key Concepts further divided into twenty-four lessons.

Start by describing the course content, introducing students to the material they will be learning. To help, this is also done in the PowerPoint presentation for Lesson 1.

Explain why you're using the Key Concepts approach

- It limits the number of main principles (to 10) a student must understand to become proficient with CNC machining centers.
- It lets students understand precisely where they stand as they go through the class.
- It provides a very good way to organize their thoughts about CNC.
- It provides a building-blocks approach to learning the material. You'll constantly be working from what student's know to what they don't.
- It puts a light at the end of the tunnel.

Programming is explained first

- The first six Key Concepts are related to programming. The last four are related to setup and operation.
- Many setup- and operation-related topics are discussed in detail during discussions about programming. A programmer must know enough about making setups and running production to direct setup people and operators.
- By the time you get to Key Concept number seven, students will have a very good understanding of the principles needed to setup and run CNC machining centers.







If you are only presenting setup and/or operation

- Again, many setup- and operation-related topics are presented during the programming-related Key Concepts.
- You'll still be going through the ten Key Concepts in order, but during the programming-related Key Concepts you'll only be presenting topics of interest to setup people and/or operators. (Note that each lesson plan provides you with guidance related to which topics to present).

Lessons for this Key Concept:**Key Concept
Number One**

Begins on **page 17** in
the student manual

Six Lessons

-  Machine configurations
-  General flow of the programming process
-  Visualizing the execution of a CNC program
-  Understanding program zero
-  Locating program zero
-  The two ways to assign program zero

Key Concept objective: Ensure that students understand the things a programmer must know about the CNC machine tool they will be working with.

Again, here are the lessons included in Key Concept one.

- Students must understand these early lessons. You'll be constantly building upon previously presented information.
- These are the things a *programmer* must know about the machine.
- In Key Concept number seven, you'll be presenting things a *setup person or operator* must know about the machine.

At the machine (about 10 minutes)

If any students have not seen a CNC machining center in action before, take them out to a machine in your lab and show them one! Have a program ready to run – cutting chips if possible. Let them see what they're going to be working with in the class. We've found that students show the most interest when they see a machine in action.

Even if they have seen CNC machines before, it will help to show them the machine/s they will be working with during the class. Also, be sure to give them a tour of the lab, showing them where workholding tools, cutting tools, hand tools, and gauging tools are kept.

Lab exercise

None for this class segment.

Homework

Have students read the Preface, pages 11-16 in the manual.

Exercise

None for this class segment.

Notes:

Explain machine components, directions of motion, and programmable functions. 30 minutes lecture time

Lessons in Key Concept #1 (you are here):

- 1: **Machine configurations**
- 2: General flow of the CNC process
- 3: Visualizing program execution
- 4: Program zero and the rectangular coordinate system
- 5: Determining program zero assignment values
- 6: Assigning program zero
- 7: Introduction to programming words

Introduce Key Concept number one. Briefly describe the related lessons. If you haven't already, you should also present some introductory information about the class itself (course times, course text, assignments, etc.).

Main topics for this lesson:

Lesson 1

Presentation links

- | | |
|--|--|
| Basic machining practice | Programmable functions |
| Machine components | Spindle |
| C frame style vertical | Feedrate |
| Gantry style vertical | Coolant |
| Horizontal | Tool changing |
| Directions of motion | Double arm |
| C frame style vertical | Single arm |
| Gantry style vertical | Others |
| Horizontal | |

Lesson objective: Introduce students to the kind/s of CNC machining center/s that they will be working with.

This is the presentation links slide for lesson one. When in the PowerPoint presentation, you can click any main topic to display the related slides (the back-arrow in the lower left corner will bring you back to this slide).

- Begin every lesson by briefly introducing the main topics.
- To skip topics, click the topic you want to present. When you're finished, click the return button. Then click the next topic you want to present.
- Use the same techniques to review topics.

Pages 19-30 in the student manual

Key points to make for each topic:

Basic machining practice

- Though beyond the scope for this class, basic machining practice is the key to mastering CNC usage.
- CNC people must understand the basic machining practices related to the CNC machine type being used.
- This understanding must include machining operations (hole machining operations like drilling, tapping, reaming, etc., as well as all forms of milling operations. They must also understand the processing (sequence of machining operations) used to machine a workpiece.

Machine components

- By showing the main components for those machine types you will be teaching, students will know what makes up a CNC machine tool.
- While students don't have to be machine designers, they should at least be able to properly reference key components by name.
- Show the difference between vertical and horizontal machining centers – as well as the differences among the various types of CNC machining centers.

Directions of motion

- Show the directions of motion (axes) for each kind of machining center you will be teaching.
- Point out that with many machines, the cutting tool does not move along with the axis (the table of a vertical machining center commonly moves in XY, while the cutting tool remains in a fixed position).
- Explain the polarity (plus versus minus) of each axis – be sure students understand. Again, since the cutting tool does not move along with every axis, polarity can be a bit confusing.

Programmable functions

- Explain that CNC programmers must know the functions of their CNC machine/s that are programmable. Presentations in the slide show include the four most common programmable functions – spindle, feedrate, coolant, and tool changing. If your machines have more programmable functions (like automatic doors), be sure to include them in your presentation.
- While this presentation includes an introduction to the related programming words, point out that students need not try to memorize them.

Spindle – be sure students understand that...

- most machining centers allow the spindle to be programmed in three ways, speed (with S), activation (with M03, M04, and M05), and range (also with S).
- speed is specified in revolutions per minute (rpm).
- M03 (forward) is used for right-hand tools and M04 (reverse) is used for left-hand tools – and that since right hand tools are much more popular than left-hand tools, M03 is more often used to activate the spindle.
- range selection is rather transparent – part of the S word.

Feedrate – be sure students understand that...

- an F word is used to specify feedrate and feedrate is specified only in per-minute fashion (inches per minute or millimeters per minute) with most machining centers.

Coolant – be sure students understand that...

- coolant is used to cool and lubricate the machining operation. M08 turns on flood coolant and M09 turns it off.

Tool changing – be sure students understand that...

- all true machining centers have automatic tool changers but they vary with regard to how they are programmed. Our slide shows help you present the two most popular types – single arm and double arm tool changers.

If you are teaching only setup and/or operation (with programming, or alone)**Main topics:**

Setup people and operators must also know about basic machining practices, machine components and directions of motion. You can eliminate the discussion of programmable functions.

Basic machining practice

- Though beyond the scope for this class, basic machining practice is the key to mastering CNC usage.
- CNC people must understand the basic machining practices related to the CNC machine type being used.
- This understanding must include machining operations (hole machining operations (drilling, tapping, reaming, etc.), as well as all forms of milling operations – as well as the processing (sequence of machining operations) used to machine a workpiece.

Machine components

- By showing the main components for those machine types you will be teaching, students will know what makes up a CNC machine tool.
- While students don't have to be machine designers, they should at least be able to properly reference key components by name.

- Show the difference between vertical and horizontal machining centers – as well as the differences among the various types of CNC machining centers.

Directions of motion

- Show the directions of motion (axes) for each kind of machining center you will be teaching.
- Point out that with many machines, the cutting tool does not move along with the axis (the table of a vertical machining center commonly moves in XY, while the cutting tool remains in a fixed position).
- Explain the polarity (plus versus minus) of each axis – be sure students understand. Again, since the cutting tool does not move along with every axis, polarity can be a bit confusing.
- Point out that a setup person or operator *must* know which way the machine will move when the plus or minus axis push-button is pressed – and when the handwheel is turned.

At the machine (about 15 minutes)

Once you've presented the lecture, demonstrate the points made at the machine in your lab. Show the main components and the directions of motion (axes) – be sure to show the polarity for each axis. Pay particular attention to any axis with which the cutting tool does not move (polarity for these axes can be confusing to students).

Be sure to demonstrate the programmable functions on the machine. Show how to start and stop the spindle, how to move the axes with jog and with the handwheel, show the activation of coolant and the automatic tool changer.

Lab exercise (about 5 minutes per student)

It's never too early to get students touching the machine – but be careful. At this early stage in the class, be sure to provide step-by-step procedures for anything you want them to do on the machine – and be sure to watch them carefully when they are practicing.

For example, provide step-by-step procedures for machine power-up, jogging axes, using the handwheel, starting the spindle, and making tool changes. Let them take turns practicing with these procedures.

Homework

- Read all of lesson one in the student manual.
- Take the quiz on page 30 of the student manual.

Exercise (about 15 minutes if done in class)

Have students do exercise number one in the workbook.

Notes:

Explain CNC-using company types and tasks related to using a machining center

15 minutes lecture time

Lessons in Key Concept #1 (you are here):

- 1: Machine configurations
- 2: **General flow of the CNC process**
- 3: Visualizing program execution
- 4: Program zero and the rectangular coordinate system
- 5: Determining program zero assignment values
- 6: Assigning program zero
- 7: Introduction to programming words

Whenever starting a new lesson, be sure to review what has been previously done. A good rule of thumb: Spend about 10% of each session reviewing. If students get confused, spend an entire session in review. Also, be sure to solicit questions about previously presented topics.

Remind students that you're still presenting Key Concept number one. Quickly review lesson one: the importance of basic machining practice, machine components, directions of motion, and programmable features.

Main topics for this lesson:**Lesson 2 Presentation links**[Understand the big picture](#)[Three company types](#)[What will you be doing?](#)[Flow of programming process](#)[Study the print](#)[Decide which machine](#)[Determine the process](#)[Choose tooling](#)[Write program](#)[Develop documentation](#)[Load program](#)[Make workholding setup](#)[Assign program zero](#)[Assemble cutting tools](#)[Measure cutting tools](#)[Load cutting tools](#)[Verify program](#)[Inspect first workpiece](#)[Run production](#)[Save corrected program](#)

Lesson objective: Introduce students to the tasks involved with getting a job up and running on a CNC machining center.

This is the presentation links slide for lesson two. Point out that it really helps to understand where CNC machine tools fit into the "bigger picture" of a company's manufacturing environment. CNC programming is but one small part of the picture.

- Introduce the topics being presented in the lesson.
- Point out that you'll simply introduce the tasks related to getting a job up and running on a CNC machine tool. Future lessons will elaborate on these tasks.



Pages 31-34 in the student manual

Key points to make for each topic:**Understand the big picture**

- Point out that different CNC-using companies expect different things from their CNC people.
- The most important factor contributing to personnel utilization is company type.
- The four most basic company types are product-producing companies, workpiece-producing companies, tooling-producing companies, and prototype-producing companies. This topic allows you to show the main differences related to how CNC people are utilized.

What will you be doing?

- Students must understand what will be expected of them once they go to work for a CNC-using company.

Flow of the programming process

- Show students the various tasks that must be completed in order to complete a job on a CNC machining center.
- While explaining each task, point out how many of these tasks require an understanding of basic machining practices.

If you are teaching setup and/or operation (with programming or alone)

This lesson must still be presented in its entirety.

At the machine (about 15 minutes)

If you have a job up and running on the machine, go out to the machine and point out what you can about the various things that must be done prior to running the job (the work holding setup, the assignment of program zero, the cutting tools, the offsets related to cutting tools, the program loading, the verification of the CNC program, etc.).

If you haven't already, run a workpiece to let students see the machine in action. Again, make sure students understand that developing a CNC program is but a small part of the entire CNC process.

Lab exercise (about 5 minutes per student)

Though we have no specific suggestions related to this lesson content, you can have your students continue practicing with procedures needed to run the machine. But again, you must be very careful to watch them as they run the machine.

Homework

Have students read all of lesson two in the student manual.

Exercise (about 10 minutes if done in class)

Have students do exercise number two in the workbook.

Notes:

Explain the importance of being able to visualize the movements of cutting tools. 15 minutes lecture time

Lessons in Key Concept #1 (you are here):

- 1: Machine configurations
- 2: General flow of the CNC process
- 3: Visualizing program execution**
- 4: Program zero and the rectangular coordinate system
- 5: Determining program zero assignment values
- 6: Assigning program zero
- 7: Introduction to programming words

Solicit questions about previous topics. Quickly review machine configurations, direction of motion, programmable features. Remind students that there are many tasks that must be completed in order to get a CNC machine up and running.

Point out that a programmer must be able to “see” the program’s execution in their mind.

Main topics for this lesson:

Lesson 3

Presentation links

[The importance of visualizing](#)

[Travel instructions analogy](#)

[Program make-up](#)

[Sequential order of execution](#)

[Machinist vs programmer](#)

[Advantage of machinist](#)

[Programmer’s disadvantage](#)

[A job handled by:](#)

[Manual machinist](#)

[CNC programmer](#)

[Program structure notes](#)

[Sequence numbers](#)

[Word order in a command](#)

[Decimal point usage](#)

[Modal words](#)

[Initialized words](#)

[Common mistakes](#)

Lesson objective: Get students to understand the importance of visualizing the program’s execution. Without this ability, they cannot write programs.

This is the presentation links slide for lesson three. Begin by reminding students about the importance of understanding basic machining practices. A machinist has seen many machining operations taking place. When writing a program, a CNC programmer must “see” the program’s execution in their mind while sitting at a bench or desk.

- Introduce the topics being presented in the lesson.
- Students will see their first complete program in this lesson.
- You’ll be introducing some points about program structure.

Pages 35-40 in the student manual

Key points to make for each topic:

The importance of visualizing – students must know that:

- without the ability to visualize a program’s execution, they cannot write CNC programs.
- even experienced machinists can have problems visualizing program execution.
- an understanding of how machining operations are performed is necessary to visualize.
- just as you cannot create a set of travel instructions without being able to visualize the path (a nice analogy is provided in the slide presentation), neither can you write a CNC program without being able to visualize how cutting tools will move through their paths.

Program make-up – students must know that:

- programs are made up of commands. Commands are made up of words. Words consist of a letter address and a numerical value. The letter address specifies the word type.
- programs are executed in sequential, step-by-step order from beginning to end.

Machinist versus programmer – students must know that:

- a machinist has everything needed to complete the job right in front of them (machine, work holding device, cutting tools, etc.). The programmer must write a program while sitting behind a desk, armed with only a print and a calculator.

An example job (machinist versus programmer)

- The slide show helps you present a simple example job – first done by a machinist, then by a CNC program. This series of slides shows the first complete program. While showing the program, you should explain each line, but be sure students understand that they don’t have to memorize the related commands.
- Stress that programs will be executed sequentially (just like a person following a set of travel instructions).
- Stress the general make-up of commands and words in the program.
- Most importantly, stress the importance of visualization – if the programmer cannot “see” the drill machining the hole in their mind, they cannot write the program.

Program structure notes – Explain:

- what sequence numbers are.
- that the word order within a command is unimportant.
- that certain word types allow a decimal point.
- the meaning of modal.
- the meaning of initialized.
- the most common beginner’s mistakes.

If you are teaching setup and/or operation (with programming or alone)

You can skip this lesson all together.

At the machine (about 20 minutes)

If students are at all weak in their basic machining practice skills, take them out to a machine and demonstrate the motions of the most common machining operations, including drilling, tapping, reaming, face milling, and side cutting.

Admittedly, this may not be enough to get them comfortable with the related operations, but at least they’ll know what each cutting tool is designed to do. Again, basic machining practice experiences is a prerequisite for this course.

Lab exercise (about 5 minutes per student)

Though we have no specific suggestions related to this lesson content, you can have your students continue practicing with procedures needed to run the machine. But again, you must be very careful to watch them as they run the machine.

Homework

- Read all of lesson three in the student manual.
- Take the quiz on page 40 of the student manual.

Exercise (about 15 minutes if done in class)

Have students do exercise number three in the workbook.

Notes:

Explain how programmed positions are determined.

30 minutes lecture time

Lessons in Key Concept #1 (you are here):

- 1: Machine configurations
- 2: General flow of the CNC process
- 3: Visualizing program execution
- 4: Program zero and the rectangular coordinate system**
- 5: Determining program zero assignment values
- 6: Assigning program zero
- 7: Introduction to programming words

Solicit questions about previous topics. Quickly review machine configurations, direction of motion, programmable features. Review the tasks related to getting a machine up and running. Review the importance of being able to visualize the execution of a CNC program.

Point out that a programmer must be able to determine positions through which cutting tools will move.


Main topics for this lesson:**Lesson 4****Presentation links**

[Rectangular coordinate system](#) [Absolute vs incremental](#)
[Axis drive make-up](#) [Absolute](#)
[Graph analogy](#) [Example](#)
[3d coordinate system](#) [Incremental](#)
[More on polarity](#) [Comparison](#)
[in XY](#) [Inch versus metric](#)
[in Z](#)
[Where to place program zero](#)
[in XY](#)
[in Z](#)

Lesson objective: Show students how positions (coordinates) are determined for use within a program. Get them to understand the absolute mode – that all positions used in a program are specified from a common location (the program zero point).

Begin by having students remember the program shown in lesson three. In this program, a drill is commanded to move through certain positions so that it could machine a hole. In this lesson, you'll be showing *how to determine tool path positions*.

- Introduce the topics being presented in the lesson.
- Students will be determining positions in a three dimensional coordinate system from a central origin.


Pages 41-50 in the student manual
Key points to make for each topic:**Rectangular coordinate system**

- You begin the slide show with an explanation of how an axis drive system works. While students don't need to know all of the inner workings of a machining center, the point we're making has to do with how positions are specified within a program. The question we eventually ask is: *How many rotations of a drive motor equate to 1" of linear motion?* Because of the rectangular coordinate system, programmers need not know the answer.
- Next, show an analogy related to making a graph. The graph in the slide show is for a company's productivity. You will relate each component of a graph to the related components of the rectangular coordinate system as it is used for CNC machining centers.
- Point out that, in CNC terms, the origin of the rectangular coordinate system is called the program zero point. All coordinates used within a CNC program will be specified from this point.
- The graph analogy shows a two-dimensional coordinate system (XY). Next, you'll show the three dimensional coordinate system for a CNC machining center – as well as how points are plotted in X, Y, and Z.

More on polarity

- While with the graph, all positions are plotted up and to the right of the origin (quadrant number one), with CNC coordinate systems, a tool must often move to a position that is to the left of or below the program zero point. Point out that every coordinate used in a CNC program has a polarity (plus or minus).
- With coordinates having a positive polarity, the polarity sign (plus) is assumed. Students must only include a polarity sign with negative coordinates (-).

Where to place the program zero point – students must know that:

- the wise placement of program zero will minimize the number of calculations needed to determine coordinates for the program.
- the program zero point is placed based upon print dimensioning. The datum surfaces for the drawing will be the program zero point surfaces for the program.
- these will be the same surfaces used for workpiece location in the work holding setup.

Absolute versus Incremental positioning

- When coordinates are specified from program zero, it is called the absolute mode of programming.
- G90 specifies absolute positioning mode.
- Students should concentrate on absolute positioning.
- Another positioning mode is available: the incremental positioning mode.
- With incremental positioning (specified by G91), positions are specified from the cutting tool's last position.
- Programs written incrementally are difficult to follow.
- If a mistake is made in a series of incremental positions, every movement from the point of the mistake will be incorrect.

Inch versus metric

- Point out that with most machines, programs can be developed in either measurement system mode.
- While most companies in the United States use the inch mode, there is an accuracy advantage to using the metric mode (shown during the slide show).

If you are teaching setup and/or operation (with programming or alone)

Setup people and operators should also understand the importance of the program zero point. When they look at CNC programs, it will help if they know the origin for the coordinates that are given in the program.

You don't have to present the entire lesson. Present from the beginning of the lesson to the main topic: *Where to place program zero.*

At the machine (about 20 minutes)

In you lab, run a program on the machine. You don't have to cut anything, but it might help hold attention if you do. As the program runs, monitor the **absolute** position display screen on the control. This screen, of course, constantly shows position relative to the program zero point.

Based upon watching this screen as the program executes, see if anyone can determine the program zero point position for the program.

You might also want to introduce the other display screen pages (relative, machine, and distance-to-go).

Lab exercise (about 3 minutes per student)

First, demonstrate how the relative position display can be used for taking measurements on the machine. Then have students practice:

Armed with a procedure for doing so, have students practice setting and resetting the relative position display screen. This, of course, is the display screen used for taking measurements on the machine. It will be used in lesson five when measuring the program zero point location at the machine.

Homework

- Read all of lesson four in the student manual.
- Have students fill in the coordinate sheet on page 48.

Exercise (about 20 minutes if done in class)

Have students do exercise number four in the workbook.

Notes:

Explain how the values needed to assign program zero are determined.

20 minutes lecture time

Lessons in Key Concept #1 (you are here):

- 1: Machine configurations
- 2: General flow of the CNC process
- 3: Visualizing program execution
- 4: Program zero and the rectangular coordinate system
- 5: Determining program zero assignment values**
- 6: Assigning program zero
- 7: Introduction to programming words

Solicit questions about previous topics. Review program zero point placement and how positions are determined for a CNC program.

Explain that just because a programmer wants the program zero point to be in a particular location, it doesn't mean the CNC machine is going to know where this location is. A conscious effort must be made to *assign* program zero.

Main topics for this lesson:


Lesson 5 Presentation links

- [Program zero must be assigned](#) [Calculating PZA values](#)
[Zero return position](#) [Retaining PZA values](#)
[Vertical machining centers](#) [Using a spindle probe](#)
[Horizontal machining centers](#)
[Program zero assignment values](#)
[In XY](#)
[In Z](#)
[Measuring program zero](#)
[In XY for rectangular workpiece](#)
[In Z](#)
[In XY for round workpiece](#)

Lesson objective: Ensure that students understand that program zero must be *assigned* – and that to assign program zero, certain values called *program zero assignment values* must be determined.

While this lesson is more related to setup and operation (especially setup), programmers must know enough about making setups to direct setup people (providing the appropriate documentation). For this reason, programmers should know how program zero is assigned at the machine.

- Introduce the topics being presented in the lesson.
- Students will be learning how to determine the location of program zero at the machine. You'll also be showing how these values can be retained for the next time the job must be run.
- You'll show how program zero is actually assigned in lesson six.

 **Pages 51-60** in the student manual

Key points to make for each topic:

Program zero must be assigned

- Point out that when a setup is made, it may be possible for the setup person to make the setup just about anywhere on the machine's table. If program zero is placed at datum surfaces (as it almost always is) the location of program zero within the setup will change based upon the placement of the workholding device.

Zero return position – Students must understand that:

- the zero return position is the point of reference for program zero assignment values.
- the zero return position is a reference position on the machine – commonly placed very close to the plus over-travel limit for each axis.
- the *machine position display screen* shows the machine position relative to the zero return position.
- three lights – called *axis origin lights* – will come on when the machine is at its zero return position.

Program zero assignment values

- Point out that one way to determine program zero assignment values is to measure them at the machine.
- For X and Y, program zero assignment values represent the distances between the spindle center while the machine is at the X and Y zero return position and the X and Y program zero point on the workpiece.
- For Z, the program zero assignment value is based upon how a feature called tool length compensation is used (discussed in Key Concept number four). If using our recommended methods, the Z axis program zero assignment value is the distance between the spindle nose and the Z axis program zero point on the workpiece.

Measuring program zero assignment values

- To measure program zero assignment values in X and Y, an edge finder or dial indicator is used.
- In the slide show, we provide to examples of program zero assignment value measurement for X and Y – with an edge finder for a rectangular workpiece and with a dial indicator for a round workpiece.
- In the slide show, we provide one way to measure the program zero assignment value in Z.

Calculating program zero assignment values

- Point out that with qualified setups (you'll probably have to explain what a qualified setup is – help for doing so is in the slide show), it may be possible to calculate the location of program zero in each axis (eliminating the need to measure program zero assignment values). The slide show helps you show how.

Retaining program zero assignment values

- Explain that with qualified setups, and even when you can't calculate program zero assignment values, it is possible to retain the program zero point location for future use. This will also eliminate the need to measure and enter program zero assignment values the next time the job is run.

Using a spindle probe

- Point out that a spindle probe facilitates the task of measuring and entering program zero assignment values. The slide show helps you show how a spindle probe is used.

If you are teaching setup and/or operation (with programming or alone)

Frankly speaking, this lesson is more related to setup than it is to programming. Again, we include it in a programming-related Key Concept because programmers must know enough about setups to direct setup people. So you must present this lesson in its entirety.

Admittedly, the procedures shown in lesson five are pretty general in nature. In the setup- and operation-portion of the course, you'll be presenting more specific information about the procedures used to actually measure program zero assignment values at the machine.

At the machine (about 20 minutes)

With a workholding setup made on your lab machine, demonstrate the techniques used to measure program zero assignment values. We recommend doing so with a rectangular workpiece using an edge finder.

This demonstration will require procedures to start the spindle (if you're using a "wiggler" style edge finder), jog the axes, use the handwheel, set and reset the relative position displays, and to do a zero return. Prior to this demonstration, we recommend creating specific written procedures for these tasks that can be used on your machine/s.

Lab exercise (about 20 minutes per student)

With the specific procedures shown above, have students practice measuring program zero assignment values in the same fashion just demonstrated. Again, be careful to monitor their progress.

If you're using this practice as a kind of assignment, have each student write down the values they come up with and submit them for grading.

Homework

- Read all of lesson five in the student manual.
- Have students fill in the coordinate sheet on page 60.

Exercise (about 20 minutes if done in class)

Have students do exercise number five in the workbook.

Notes:

Explain how program zero is actually assigned.

15 minutes lecture time

Lessons in Key Concept #1 (you are here):

- 1: Machine configurations
- 2: General flow of the CNC process
- 3: Visualizing program execution
- 4: Program zero and the rectangular coordinate system
- 5: Determining program zero assignment values
- 6: Assigning program zero**
- 7: Introduction to programming words

Solicit questions about previous topics. Review program zero point placement and how positions are determined for a CNC program. Review how program zero assignment values are determined.

Explain that there are actually two ways to assign program zero – and for all but the oldest machines, students will be using *fixture offsets* for doing so.

Main topics for this lesson:

Lesson 6 Presentation links

[Assigning in the program](#) [Assigning with fixture offsets](#)

[In XY](#)

[In XY](#)

[In Z](#)

[In Z](#)

[The G92 command](#)

[Advantage of fixture offsets](#)


[Limitation of G92](#)

[Using G28 for safety](#)

Lesson objective: Be sure students understand how program zero is assigned – with fixture offsets (most machines) or with G92 in a program (old machines).

Like lesson five, this this lesson is more related to setup than programming. But again, programmers must know enough about making setups to direct setup people (providing the appropriate documentation).

- Introduce the topics being presented in the lesson.
- For most machines, students will learn how to enter program zero assignment values into fixture offsets.
- For old machines (over 20 years old), students will learn how to assign program zero with the G92 command.
- Show only the program zero assignment method required for your lab machine/s (probably fixture offsets). The presentation links slide makes this easy.

 **Pages 61-66 in the student manual**

Key points to make for each topic:

Assigning program zero in the program (old machines)

- Regardless of how program zero is assigned, the program zero assignment values shown in lesson five will be used.
- The polarity for program zero assignment values is taken *from program zero to the zero return position* (almost always positive).
- A G92 command at the beginning of the program includes the program zero assignment values.
- G92 has many limitations and can be difficult – if not dangerous – to use (limitations are shown in the slide show). Fixture offsets eliminate these limitations.
- A zero return command (G28) should be included prior to the G92 command to ensure that the machine is in the proper position prior to executing the G92 command.

Assigning program zero with fixture offsets (most machines)

- Regardless of how program zero is assigned, the program zero assignment values shown in lesson five will be used.
- With fixture offsets, the polarity for program zero assignment values is taken *from the zero return position to program zero* (almost always negative).
- Fanuc-controlled machines come standard with six fixture offsets – more can be purchased as an option.
- The program zero assignment values are entered into the appropriate fixture offset registers.
- If assigning one program zero point in the program (a common scenario), use fixture offset number one (specified by G54 in the program).
- Point out the advantages of fixture offsets (over assigning program zero with G92 in the program).

If you are teaching setup and/or operation (with programming or alone)

As with lesson five, this lesson is more related to setup than it is to programming. Again, we include it in a programming-related Key Concept because programmers must know enough about setups to direct setup people.

So you must present this lesson in its entirety, especially if you are teaching the use of fixture offsets to assign program zero.

At the machine (about 20 minutes)

We're assuming you are using fixture offsets to assign program zero (again, you should use fixture offsets to assign program zero unless your machine does not have them). Use the program zero assignment values measured in lesson five and show how they are entered into fixture offset registers. Show the various fixture offset pages (commonly fixture offsets one through six).

If you have an old machine that doesn't have fixture offsets, then show how the G92 command in a program must be edited in order to assign program zero.

Lab exercise (about 5 minutes per student)

Again, we recommend that you provide a step-by-step procedure to enter fixture offsets, so students can practice with minimal help from you. Have them work with an unused fixture offset for practicing (like fixture offset number six so they cannot overwrite needed fixture offset values.

Using the program zero assignment values they measured in lesson five, have them enter them into fixture offset registers.

Homework

- Read all of lesson six in the student manual.
- Have students fill in the coordinate sheet on page 66.

Exercise (about 20 minutes if done in class)

Have students do exercise number six in the workbook.

Notes:

Introduce students to the word types used with CNC programs.

15 minutes lecture time

Lessons in Key Concept #1 (you are here):

- 1: Machine configurations
- 2: General flow of the CNC process
- 3: Visualizing program execution
- 4: Program zero and the rectangular coordinate system
- 5: Determining program zero assignment values
- 6: Assigning program zero
- 7: **Introduction to programming words**

Solicit questions about previous topics. Review program zero point placement and how positions are determined for a CNC program. Review how program zero assignment values are determined and how program zero is assigned.


Point out that in Key Concept number one, students have been exposed to several programming words. In this lesson you'll be showing all of the word types used in programming.

Main topics for this lesson:**Lesson 7 Presentation links**[Intro to word types](#)[Word types \(continued\)](#)[Word types](#)[O – program number](#)[\(\) – messages](#)[N – sequence number](#)[G – preparatory function](#)[X – X axis designator](#)[Y – Y axis designator](#)[Z – Z axis designator](#)[Decimal format \(XYZ\)](#)[Related words \(XYZ\)](#)[A B C – Rotary axis words](#)[R – rapid plane](#)[I J K – directional vectors](#)[Q – peck depth](#)[P – pause time](#)[L – number of executions](#)[F – feedrate](#)[S – spindle speed](#)[T – tool changing](#)[M – miscellaneous functions](#)[D H – offset number specification](#)[/ - block delete](#)

Lesson objective: Acquaint students with the word types (letter addresses) used in CNC machining center programs.

In this final lesson for Key Concept number one, you'll be explaining the meaning of all CNC word types. Though students need not try to memorize every work type, it helps if they can see the limited number of different words available to CNC programmers.

- Introduce the topics being presented in the lesson.
- Point out that this section in the student manual will make an excellent reference. If students forget the meaning of a given word type, they can come to this material.


Pages 67-76 in the student manual
Key points to make for each topic:**Introduction to word types**

- Point out that there are only about 50-60 different words used in CNC machining center programming. Have students look at learning programming as like learning a foreign language that has only 60 words.
- Many word types are easy to remember (like T for tool, S for speed, and F for feedrate. Others are not so easy to remember (like O for program number and N for sequence number).

Word types

- The slide presentation allows you to present the various word types in the order shown on the presentation links slide. In each case, you'll be telling students whether the word is a real number (allowing a decimal point) or an integer (whole number). You'll also specify the format for the word. Finally, you'll explain the word meaning, including any primary and secondary uses for the word.
- The student manual includes a full list of G and M words (starting on page 72). Point out that M words are determined by machine tool builders and can vary from machine to machine.

If you are teaching setup and/or operation (with programming or alone)

While it doesn't hurt setup people and operators to know the meaning of the various words used in programming, you can skip this lesson entirely – or just introduce the most common and memorable words.

Since setup people and operator may be called upon to call up programs and modify cutting conditions, for example, you may elect to describe words like O, F, and S). You may also wish to introduce G and M words.

At the machine (about 10 minutes)

While it doesn't have to be at the machine tool (the text editor of a computer will work), call up a program and point to the various words. See if students can remember any of the word meanings for words they see in the program.

Show the program check display screen page, which shows the currently active CNC words. Again, see if students can remember any of them.

Lab exercise

We have no suggestions for lab exercises that are related to this lesson. If you wish, you can have students continue practicing with machine operation procedures.

Homework

- Read all of lesson seven in the student manual

Exercise (about 15 minutes if done in class)

Have students do exercise number seven in the workbook.

Notes:**Review questions for Key Concept number one**

We cannot overstress the importance of reviewing material. As stated, you should start each session with a review. We recommend that you spend at least ten percent of each session reviewing what you've previously presented. And there may be times when you need to spend more time in review (if students are becoming confused). Reviews can do more than simple repeat or reiterate the same material. There are times when you can actually add to previous presentations as students catch on. What may be too much for them to handle the first time through the material may be easy to understand during a review.

One effective way to review is to treat the review as a kind of oral quiz, having students answering critical questions. If they answer quickly, and if everyone seems to understand, of course, continue. But when they can't answer questions, it should be taken as a signal that more presentation is necessary. Below we provide some example questions you can ask during your review of Key Concept number one. You may be able to add to this list.

Lesson One – Machine configurations:

- What are the two most basic types of machining centers? How can you tell the difference?
- What are some of the most basic components of a machining center?
- What are the three linear axes?
- What is the polarity (plus versus minus) for each axis?
- What are the three ways to control a machining center's spindle? What letter address is used to control spindle speed? What are the M codes used for spindle activation? Which spindle activation M code is used for right hand tools? How does the programmer specify a spindle range?
- What letter address is used to specify feedrate? How is feedrate specified?
- What other features on a machining center might be programmable?

Lesson Two – General flow of the CNC process:

- What are the four company types that use CNC machine tools?
- What are some of the tasks that must be completed in order to get a CNC machining center up and running?

Lesson Three – Visualizing program execution:

- Why must you be able to visualize a CNC program's execution?
- Name the basic components of a CNC program? How are CNC programs executed?
- What does modal mean? What does initialized mean? Which words allow a decimal point?
- What is the most common mistake a beginning programmer is prone to making?

Lesson Four – Program zero and the rectangular coordinate system:

- What do you call the origin for a CNC?
- How do you determine where to place the program zero point? What about Z?
- When you specify coordinates from program zero, what positioning mode is it called? What is the G code that specifies the absolute mode?
- What is the other positioning mode? What is the G code for incremental mode?

Lesson Five – Determining program zero assignment values:

- What must the CNC machine be "told" in order to assign program zero?
- What do the program zero assignment values represent?
- When will it be necessary to measure program zero assignment values at the machine during setup?
- How do you measure program zero assignment values at the machine during setup?
- When will it be possible to eliminate the need to measure program zero assignment values?

Lesson Six – Assigning program zero:

- What are the two ways to assign program zero? Which is better? Why?
- What is the polarity for fixture offset entries?

Lesson Seven – Introduction to CNC words:

- Approximately how many different word types are used in a CNC program?
- What is the meaning of the letter address O? N? G? X, Y and Z? F? S? T? R?
- How many G codes can be used per command? How many M codes can be used per command?