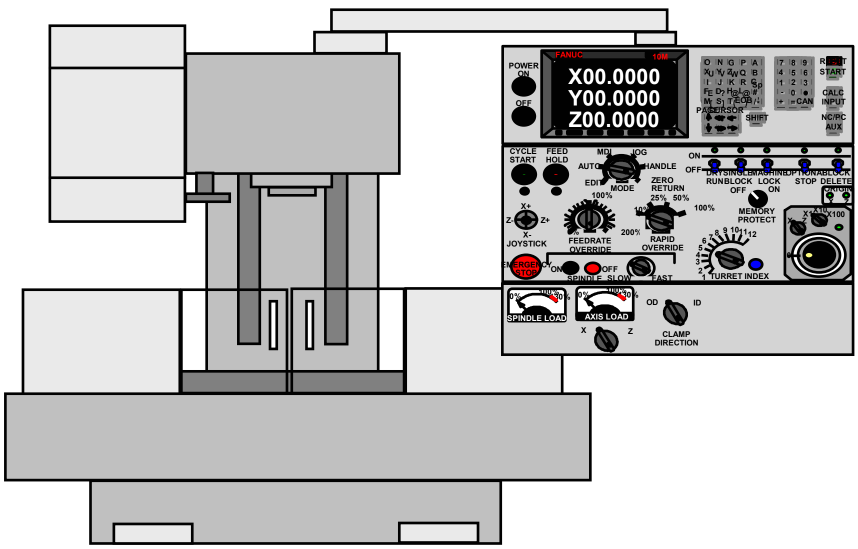
# Lesson plans

*for*

# Machining Center Programming, Setup, and Operation

Mastering CNC Machining Center Utilization

MIKE LYNCH




Concise information to help you get ready to teach – one double-sided sheet per lesson:

- Lesson name
- Lesson number
- Approximate time needed to present the lesson
- Presentation links slide from the slide show (list of topics)
- Lesson objective
- Key points to make for each topic
- Suggestions for if you are teaching only setup and/or operation (not programming)
- Suggestions for what to do at the machine to stress the points made in the lesson
- Suggestions for a lab exercise (if appropriate)
- Suggested homework assignment
- Suggestions for exercises to be done

These lesson plans are intended to be used in conjunction with our instructor materials (slide show presentations) and student materials (student manual and workbook)

CNC CONCEPTS, INC.


44 Little Cahill Road • Cary, IL 60013
Phone 847.639.8847 • Fax 847.639.8857
Internet: www.cncci.com

# Front and back of each page in this manual

| Lesson Plan | Machine Configurations | Lesson 1 |
|---|---|---|
| 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
Machine components
- C frame style vertical
- Gantry style vertical
- Horizontal

Directions of motion
- C frame style vertical
- Gantry style vertical
- Horizontal

Programmable functions
- Spindle
- Feedrate
- Coolant
- Tool changing
  - Double arm
  - Single arm
- Others

**Pages 19-30** in the student manual

**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.

**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 introductions 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 basis 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:**



## Lesson name and number
- At the top of each page we provide the lesson name and number so you can quickly find the lesson you need.

## Lesson intention and approximate presentation time
- In the second vertical row, we briefly state what you'll be covering in the lesson. We also provide the approximate time it will take to go through the material in the lesson. These times are approximate and will vary, of course, based upon student aptitude and the number of questions students have.

## Where you are in the class
- In the third vertical row, we provide a portion of the course outline – and highlight (bold) the current lesson – providing context relative to the rest of the class. We also provide some suggestions for introducing the lesson.

## Main topics for this lesson
- In the fourth vertical row, we show the topics you'll be covering. This is a copy of the presentation links slide in the slide show presentation for the lesson. This slide provides links to the main topic in the slide show, making it easy to get around – and making it especially easy to review.

## Pages in the student manual
- In the presentation links slide, we show the student manual pages related to the lesson. It will help you tell students where related material is covered in the book.

## Lesson objective
- To the right of the presentation links slide, we provide the main objective for the lesson. We also provide more suggestions for introducing the lesson.

## Key points to make for each topic
- In the fifth vertical row we provide suggestions for teaching each of the lesson's main topics. These suggestions parallel the main topics listed on the presentation links slide. In many cases, we make reference to PowerPoint presentation slides.
- This section is truly at the heart of the of each lesson plan. Here we provide suggestions for teaching the class.

## If you are teaching setup and operation
- Our curriculums are designed to help you teach all three skills related to mastering CNC machining center use. However, you may be teaching setup people and/or operators that don't have a need to know programming. We still recommend that you go through the course in the same order – lesson by lesson (though some lessons can be skipped – we'll point them out as we go along).
- In the next vertical column (possibly on the next page), we provide suggestions aimed at helping you teach setup people and operators – and programmers needing to know how to setup and operate machining centers.

## At the machine
- Here we provide suggestions for emphasizing what you've taught – in your lab and on an actual machine.

## Lab exercise
- When appropriate, we provide suggestions for practice right on the machine in your lab.

## Homework and exercise
- In the next vertical row, we provide suggestions for homework and exercises. In most cases, this includes reading and exercises in the student manual, as well as exercises from the workbook.

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:**


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:

| Lesson Plan | Machine Configurations | Lesson 1 |
| --- | --- | --- |
| 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
- Machine components
  - C frame style vertical
  - Gantry style vertical
  - Horizontal
- Directions of motion
  - C frame style vertical
  - Gantry style vertical
  - Horizontal
- Programmable functions
  - Spindle
  - Feedrate
  - Coolant
  - Tool changing
    - Double arm
    - Single arm
  - Others

**Pages 19-30** in the student manual

**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.

**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 introductions 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 basis 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:

# General Flow Of The CNC Process

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

**Pages 31-34** in the student manual

**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.

**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:**

Lesson Plan | **Visualizing Program Execution** | **Lesson 3**

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

**Pages 35-40** in the student manual

**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.

**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:

Lesson Plan | **Program Zero And The Rectangular Coordinate System** | **Lesson 4**

Explain how programmed positions are determined. | 30 minutes lecture time



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



**Pages 41-50** in the student manual

**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.

**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
- Zero return position
  - Vertical machining centers
  - 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
- Calculating PZA values
- Retaining PZA values
- Using a spindle probe

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

**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.

**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 on the workpiece (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
- In XY
- In Z
- The G92 command
- Limitation of G92
- Using G28 for safety

Assigning with fixture offsets
- In XY
- In Z
- Advantage of fixture offsets

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

**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.

**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



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
- 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

Word types (continued)
- 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

**Pages 67-76** in the student manual

**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.

**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 an 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?

Lesson Plan | **You Must Prepare To Write Programs** | **Key Concept 2**

Introduce Key Concept number two. | 10 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

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.

Key Concept number two is a short, one-lesson key concept. Though it is short, it is among the most important Key Concepts.

**If you are only presenting setup and/or operation**
▪ The preparation steps we show in Key Concept number two are related only to programming – so you can skip the material in this Key Concept. However, you'll still want to stress the importance of being properly prepared to perform any CNC task. The better prepared the setup person or operator, the easier it will be to perform the task. For example, gathering all components necessary to make a setup up-front will ensure that the setup person can complete the setup without repeated trips to the tool crib.

**Introduction slide for this Key Concept:**

Key Concept Number Two: Begins on **page 77** in the student manual

Preparation for programming

- Programming
- Documentation
- Program loading
- Work holding setup
- Program zero location
- Cutting tool assembly
- Loading tools
- Offset entries
- Program verification
- Sizing workpieces
- Inspection
- Program saving

The better prepared you are to do any task, the better you will be able to do it!

**Key Concept objective:** Help students understand the steps that must be taken prior to writing a program.

Key Concept number two is a one-lesson Key Concept. Lesson eight: Preparation for programming.

The presentation links slide (shown in the lesson plan for lesson 8) provides links to slides that help you explain the topics shown below (preparation and time, preparation and safety, and typical mistakes).

**Key points to make while introducing to Key Concept number two:**

**Preparation and time**
▪ Point out that some programmers skip the preparation steps in an attempt to save time. In reality, inadequate preparation usually adds to the time it takes to complete a job. And wasted time at the machine is a costly penalty to pay for lack of preparation.

**Preparation and safety**
▪ Inadequate preparation leads to mistakes. Mistakes in the program can lead to dangerous situations.

**Typical mistakes**
▪ Point out the mistakes a beginner is prone making including syntax mistakes, motion mistakes, processing mistakes, and mistakes of omission.

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

For the most part, you can skip Key Concept number two since it applies only to programming. But again, point out that setup people and operators must be properly prepared to perform their tasks.

### At the machine (about 10 minutes)

If you lab is well organized, you can use it to show students what they can expect from well organized companies – all hand tools, cutting tools, and work holding tools have a place and are put back in their place when not being used, drawers are well labeled and organized, and in general, the working area around the CNC machine/s is not cluttered.

But frankly speaking, many companies are ***not*** very well organized. Be sure students have a way to remember how your lab is arranged (pictures work nicely) for the time when they must organize the work area in a company they're working for.

### Lab exercise

We have no recommendations for lab exercises for this segment of the course.

### Homework

- Read the introduction to Key Concept number two that begins on page 77 in the student manual.

### Exercise

None for this segment of the course.

## Notes:

Explain and demonstrate the preparation steps required for programming. | 20 minutes lecture time

Lessons in Key Concept #2 (you are here):
- 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

Solicit questions about previous topics. If you haven't already, review the topics in Key Concept number one.

Explain that though Key Concept number two has little or nothing to do with programming words and commands, it is among the most important Key Concepts. Programmers must be prepared to write programs. With preparation done, writing a program is simply a matter of translating the plan into a language the CNC machining center can understand.

**Main topics for this lesson:**

**Lesson 8**

## Presentation links

- Key concept #2
- Divide and conquer
- Typical mistakes
  - Syntax mistakes
  - Mistakes of omission
  - Motion mistakes
- Preparation steps
  - Study & mark up print
  - Develop the machining process
  - Do the math
    - Number all points
    - Develop a coordinate sheet
  - Plan the setup
- Conclusion

**Pages 81-94** in the student manual

**Lesson objective:** Ensure that students understand and can perform the four steps required to prepare to write CNC programs.

This class, of course, is presenting G code level, manual programming. However, the preparation steps we show in this lesson are necessary regardless of how programs are prepared. If, for example, students will eventually be using a computer aided manufacturing (CAM) system to prepare programs, all of the preparation steps we show (except doing the math) will be required.

- Remind students that adequate preparation will make programming much simpler, reducing the potential for mistakes. Frankly speaking, the quality of most programs is directly related to the quality of the preparation done before the program is written.
- With a few programs under their belts, students will be tempted to skip these preparation steps. Doing so can be a terrible mistake.

**Key points to make for each topic:**

**Preparation steps**

- Point out that any complex task can be simplified by breaking it into small pieces. In a sense, we're providing a way to *divide and conquer*.
- In the slide show, there is an analogy for making a speech. Just as an ill-prepared speaker will be likely to make mistakes during the presentation, so will the ill-prepared programmer be prone to making mistakes.

**Study and mark up the print**

- Explain that in most companies, the programmer is given a working copy of the workpiece drawing (print). They can mark up this print in any way that helps them understand the job.
- The programmer should mark up the location of program zero, they should mark the surfaces that get machined, they should draw in any clamps or other obstructions, and in general, they should mark up anything that will help them during programming.

**Develop the machining process**

- The series of slides begins by explaining a process planning form that is provided in the student manual.
- Then we provide slides to help you explain the benefits of this form. Before the program is written, the programmer is forced think through: the process and all cutting tools used in the job, possible tooling interference problems, and cutting conditions fore each cutting tools.
- Point out that this completed form is the English-version of the program. Writing the program will be a simple matter of translating this form into a language the CNC machining center can understand.
- Explain that this form also makes great documentation for anyone who must work on the program in the future.

**Do the math**

- Point out that doing the math up-front will keep the programmer from breaking out of their train of thought when programming to come up with coordinates needed in the program.
- The slide show helps you demonstrate our recommended method of calculating coordinates – numbering each point on the print through which cutting tools will move and making a *coordinate sheet* that has all coordinates for these points. (This should be familiar to students if they have been doing the exercises.)
- The slide show also helps you point out that often the coordinates needed in the program are not specified right on the print. Our example shows the milling of a circular pocket.
- We also recommend that all Z coordinates be calculated prior to programming – and often several Z coordinates are required for one XY position (consider center-drilling, drilling, and tapping a hole). We provide a way to easily document these Z coordinates in the slide show.

**Plan the setup**

- Point out that there are many things about the setup that affect the way a program must be written. For example, clamps and other obstructions must be avoided by cutting tools. For this reason, the programmer must understand how the setup will be made before they can write a program.
- In the slide show, we describe a setup sheet (that is also in the student manual), helping students understand the things that must be documented for the setup person.

**Conclusion**

- We provide a series of slides to help you quickly review the preparation steps.

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

Once again, everything in lesson eight is related to preparation steps for *programming* – so you can skip this lesson if you're teaching only setup and/or operation. But as stated, you may want to explain the preparation steps a setup person or operator must perform.

### At the machine

We have no recommendations for things to do at the machine that are related to this lesson's content.

### Lab exercise

We have no recommendations for lab exercises related to this lesson.

### Homework

- Read all of lesson eight in the student manual.
- Have students fill in the coordinate sheet on page 94.

### Exercise (about 15 minutes if done in class)

Have students do exercise number eight in the workbook.

## Notes:

### Review questions for Key Concept number two

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.

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 Eight – Preparation steps for programming:***

- Why must you prepare to write programs?.
- Which is more valuable, the programmer's time or machine time?.
- Name the for general types of mistakes a beginning programmer is prone to making.
- Name the six steps that should be taken to get ready to write a program. (Mark up the print, develop the machining process, do the coordinate calculations, check the cutting tools, plan the setup, and create all documentation.)

Lesson Plan | **Understand The Motion Types** | **Key Concept 3**

Introduce Key Concept number three. | 5 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

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.

Key Concept number three is another one-lesson key concept. You'll be describing the three most common motion types during this key concept.

**If you are only presenting setup and/or operation**

▪ This is another Key Concept that is devoted to programming. This being the case, it doesn't hurt setup people and operators to know the ways a CNC machining center can move. You might want to introduce the motion types, but not show any of the related motion commands.

**Introduction slide for this Key Concept:**

Key concept number three:

You must understand the motion types available to you

Begins on **page 94** in the student manual

There are three (of four) basic motions that are used in almost all CNC programs

**Key Concept objective:** Help students understand and master the motion types for CNC machining centers.

Key Concept number three is a another one-lesson Key Concept. Lesson nine: Programming the three most basic motion types.

The presentation links slide (shown in the lesson plan for lesson 9) provides links to slides that help you explain the topic shown below (interpolation, ).

**Key points to make while introducing to Key Concept number three:**

**Interpolation**

▪ Remind students that they currently know how to determine coordinates (positions) through which cutting tools will move (this is presented in lesson four). But they must also know what it takes to command *how a cutting tool will move from point to point.*

▪ The slide show helps you describe *interpolation*. While students may not need to know the details of how interpolation works, it helps to understand what the machine is doing for them.

▪ We also briefly introduce the three most common motion types in this session – rapid motion, straight line (linear) motion, and circular motion.

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

You can skip Key Concept number three since it applies only to programming. But again, you may want to simply introduce motion types to setup people and operators so they can recognize the related motions when they see them taking place on the machine.

Do keep in mind that some companies require their setup people to be able to modify programs (to correct minor mistakes). While these modifications are usually limited to cutting conditions, if a setup person must make modifications to motion commands, they must of course, understand how motion types are programmed.

### At the machine (about 10 minutes)

With a job up and running on your lab machine, shows students the three basic motion types: rapid, straight-line, and circular. Point out that one command per motion is required.

### Lab exercise

We have no recommendations for lab exercises for this segment of the course.

### Homework

- Read the introduction to Key Concept number three that begins on page 95 in the student manual.

### Exercise

None for this segment of the course.

## Notes:

Present the three most common ways a CNC machining center can move. | 35 minutes lecture time

Lessons in Key Concept #3 (you are here):
- 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

Solicit questions about previous topics. Review the topics in Key Concepts one and two.

Explain that though students can calculate coordinates (from lesson four), they have only solved half the puzzle. *They must also know how to command the method by which a cutting tool will move from one point to the next.* This is the focus of lesson nine.

**Main topics for this lesson:**

**Lesson 9** Presentation links

- Key concept #3
- Interpolation
- Three motion types
- Motion commonalities
- Programmed point
  - For hole-making tools
  - For milling cutters
- Rapid motion
  - Examples
- Linear motion
  - Examples
- Circular motion
  - G02 or G03?
  - Specifying arc size
    - With R word
      - Example
      - Warning about R word
    - With Directional vectors
  - Arc limitations
  - Full circle in one command
    - Example
- Helical motion

**Pages 99-116** in the student manual

**Lesson objective:** Bring students to a level that they understand and can command the three most common motion types.

Explain that programming the motion types is remarkably simple. Students must specify the kind of motion they want, the end point for the motion, and possibly a feedrate. Circular motions additionally require the arc size to be specified.

- Motion types share several things in common. When students understand one motion type, similar techniques will be used for all.
- With the preparation steps described in lesson eight completed (especially the math), commanding motion should be relatively simple.

**Key points to make for each topic:**

**Interpolation and the three motion types**

- You present these topics in the introduction to Key Concept number three.

**Motion commonalities**

- The slide show helps you present the five things that all motion types share in common: all are modal, the end point is commanded, only moving axes are specified, all are affected by positioning mode (absolute or incremental), and the leading zero for each can be suppressed (G0 is the same as G00, etc.).

**Point programmed**

- Beginning programmers often have a problem with this. They must understand the actual point of the cutting tool that they are programming.
- In X and Y, hole-machining cutting tools (drills, taps, reamers, etc.) are simple. The hole's center line coordinates are programmed (center of the cutting tool).
- In Z, hole-machining cutting tools usually require the programmer to compensate for the cutting tool's *lead*. That is, it is the very tip of the cutting tool in Z that is being programmed. The slide show helps you present this.
- In X and Y for milling cutters, point out that sometimes it is the center of the cutting tool that is programmed – especially when face milling. But when side-milling, and especially when milling a contour, there is a feature called *cutter radius compensation* (presented in Key Concept number four), that allows the programmer to specify work-surface coordinates in the program – which makes programming much simpler. But point out that for now, the centerline positions for milling cutters must be used for all examples (until Key Concept number four is shown.

**Rapid motion – Students must understand that:**

- rapid motion is commanded by G00.
- motion occurs at the machine's fastest possible rate.
- a straight motion may not occur when two or more axes are specified (one axis will probably reach its destination first).
- rapid motion is used to reduce program execution time (whenever the cutting tool is not cutting, rapid motion should probably be used).
- An example in the slide show helps you stress these points.

**Linear motion – Students must understand that:**

- linear motion (also called straight-line cutting motion) is commanded by G01.
- motion will occur along a straight line, even if more than one axis is specified.
- a feedrate (F word) must be specified in (at least) the first linear motion command. Feedrate is modal – if a series of motions must occur at the same feedrate, only the first motion command requires a feedrate.
- this command is used when machining must occur along a straight line.
- Several examples in the slide show help you stress these points.

**Circular motion – Students must understand that:**

- G02 specifies clockwise circular motion – G03 specifies counter clockwise circular motion.
- motion will occur along a circular path.
- the arc size must be specified (with an R word).
- the current feedrate will be used.
- these commands are used when machining circular surfaces.
- Several examples are shown in the side show to help you stress these points.

**Specifying arc size**

▪ The slide shows help you present the two ways to specify arc size – with an R word to specify arc size directly and with directional vectors (I, J, and K). We recommend that students use the R word. But for the sake of completeness, we do help you show how directional vectors are used.

**Arc limitations**

▪ We help you present the limitations of circular motion commands. For example, point out that it is possible to cross only one quadrant line (arc centerline) per command. This means that it is not possible to generate an arc greater than 180 degrees per command. Larger arcs must be broken into two commands.

**Full circle in one command**

▪ Point out that one exception to the "crossing one quadrant line" limitation is commanding a full circle in one command. We show the technique for doing so in the slide presentation and student manual

**Helical interpolation**

▪ The slide show helps you introduce a fourth motion type called *helical interpolation*. This motion type is used for thread milling.

▪ Point out that if a company does not perform thread milling operations, there will be no need for helical motion. For this reason, helical motion is an option that must be purchased for an additional price from Fanuc.

▪ Thread milling (including helical motion) is presented in Key Concept number six.

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

Once again, everything in lesson nine is related to *programming* – so you can skip this lesson if you're teaching only setup and/or operation. But as stated, you may want to simply introduce the three most common motion types.

## At the machine (20-30 minutes)

Students actually work on their first two programs in the activities related to this lesson (in the homework activity on page 116 of the student manual and in the workbook exercise for lesson nine). You can use either of these program to help them get some meaningful practice at the machine. Or, if you have developed your own practice program (that actually machines a workpiece), you can use it instead.

Have them type the program into the control (meaning you'll need a step-by-step procedure that shows them how to enter new programs). Be sure to double check this program for mistakes since you're not going to be teaching program verification techniques at this point. For this reason, you may elect to let them practice typing the program into the control – but use your own proven program (that *you* have verified) when you actually run the program.

This also makes a great time to quiz them on some of the tasks that must be done prior to running a program. From what students should know so far, they should quickly point out that program zero must be assigned. Review the techniques for measuring program zero assignment values and entering them into a fixture offset.

While you haven't presented this yet, point out that cutting tools must be assembled and measured – and that tool lengths must be entered into tool length compensation offsets (these techniques are shown in Key Concept number four.

When you're ready, run the program for them. Again, it might be wise to simply air cut – without a workpiece in position. Students can still nicely see the program's execution, including the three motion types introduced in lesson nine.

## Lab exercise

We have no recommendations for lab exercises related to this lesson. Since students commonly work on their own when doing lab exercises, we don't recommend letting them type and run their own programs (yet).

## Homework

▪ Read all of lesson nine in the student manual.

▪ Have students fill in the blanks for the program on page 116 of the student manual.

## Exercise (about 35 minutes if done in class)

Have students do exercise number nine in the workbook. This requires them to calculate coordinates and fill in the blanks for an actual program – will take some time to do. You may not want to take so much time in class – instead you may elect to make this a homework assignment.

## Notes:

## Review questions for Key Concept number three

***Lesson nine – Motion types:***

▪ What is interpolation?

▪ For these cutting tools (drill, tap, reamer, boring bar, face mill, and end mill), name the point on the tool that is being programmed.

▪ What are the three most common motion types and what are their related G codes?

▪ Name the five things that all motion types share in common. (All are modal, all require that you specify the end point, only the moving axes need be specified, all are affected by absolute and incremental modes, all allow you to suppress the leading zero.)

▪ When should rapid motion be used? Linear? Circular?

Introduce Key Concept number four. | 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

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.

Key Concept number four contains four lessons that are related to certain unpredictable variables of tooling (work holding tools and cutting tools).

Start by pointing out that a programmer won't know every detail about a setup is made as the program is being written. These compensation types allow the programmer to ignore certain tooling-related information while they write the program.

Later (commonly at the machine during setup), the setup person (or someone) will determine and enter this information into the machine separate from the program.

**If you are only presenting setup and/or operation**
▪ This Key Concept contains much information that will be of great interest to setup people and operators. We'll be introducing principles related to trial machining and sizing, which setup people and operators must understand. While you'll want to stop short of getting too detailed with programming words and commands, you'll be presenting at least the very beginning of each lesson in Key Concept number four.

**Lessons related to this Key Concept:**


Begins on **page 117** in the student manual

Four Lessons

- 10 - What is compensation?
- 11 - Tool length compensation
- 12 - Cutter radius compensation
- 13 - Fixture offsets

**Key Concept objective:** Help students understand and master the three compensation types.

Remind students that they have now worked on their first complete CNC programs, filling in the blanks for motion commands. While doing so, they may have been wondering about the meaning of certain words in these practice programs. In this Key Concept, you'll be presenting the words and commands related to compensation types.

**Key points to make while introducing to Key Concept number four:**

**Introduction to compensation**
▪ Since lesson ten introduces your students to compensation, there is not much to say at this time. Move on to lesson ten.

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

Again, much of this Key Concept will be of great interest to setup people and operators. You'll elaborate in each lesson.

### At the machine

We have no suggestions yet.

### Lab exercise

We have no recommendations for lab exercises for this segment of the course.

### Homework

- Read the introduction to Key Concept number four that begins on page 117 in the student manual.

### Exercise

None for this segment of the course.

## Notes:

Explain why compensation is required with CNC machining centers. | 20 minutes lecture time

Lessons in Key Concept #4 (you are here):

4: Know the compensation types
- **10: Introduction to compensation**
- 11: Tool length compensation
- 12: Cutter radius compensation
- 13: Fixture offsets

Solicit questions about previous topics. If you haven't already, review topics in Key Concepts one, two, and three.

Explain that there are certain things that a programmer won't know about the machine's setup while a program is written. Compensation lets the programmer write the program without knowing these things. For example, the programmer won't know the physical position of program zero in the setup – but with fixture offsets, they can still write the program.

**Main topics for this lesson:**

**Lesson 10** Presentation links

- Key concept #4
- Analogies
  - Marksman analogy
    - Related to CNC compensation
- Understanding offsets
- Offset tables
  - One value per offset
  - Two values per offset
  - Geometry and wear offsets
  - Fixture offsets
- Trial machining

**Pages 119-124** in the student manual

**Lesson objective:** To make sure that students understand why compensation types are required.

During lesson six, you actually introduced one of the compensation types (fixture offsets). This was necessary as part of explaining program zero – and how program zero is assigned at the machine. In lesson ten, you'll be discussing why compensation is required for other tooling-related reasons.

- The compensation types will draw upon the students' basic machining practice skills.
- Tooling related information will be entered at the machine into some form of *offset*.

**Key points to make for each topic:**

**Analogies**

- The marksman analogy shown in the student manual and slide presentation is remarkably similar to how the compensation types are used on CNC machining centers. Use it to help students understand that an initial compensation setting may not be perfect. The tighter the tolerance, the more likely it will be that a second adjustment will be necessary after a cutting tool machines.
- If tolerances are small, it may be necessary to make an initial adjustment that forces the cutting tool to leave excess stock. After machining, another adjustment will be necessary.

**More on tolerances**

- Though not included in the slide show, the student manual contains a presentation about tolerance interpretation.
- Use this to ensure that students understand tolerance bands, whether a measured dimension is within the tolerance band, the target value for an adjustment, and how much adjustment (the deviation) is required.

**Understanding offsets**

- From the marksman analogy, point out that offset settings are like the amount of sight adjustment needed for the rifle.
- We also compare CNC offsets to the memories of an electronic calculator – they are referenced by a number and they have no meaning until they are invoked. But unlike calculator memories, CNC offsets are more permanent. They are retained after the machine's power is turned off.
- It is in offset registers that students will be entering certain tooling related information (like tool lengths, milling cutter radii, and fixture placement).
- The slide show helps you describe the various offset pages on a typical CNC machining center.

**Trial machining**

- While more detailed descriptions of trial machining are shown during the lessons for each compensation type, we introduce trial machining in lesson ten.
- The slide show helps you nicely illustrate a workpiece that has close enough tolerances that trial machining is required.

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

You'll want to do a thorough review of measuring devices and tolerance interpretation. With every workpiece attribute a setup person or operator measures, they will be making critical decisions:

What is the measured dimension? (Has the student measured the attribute correctly?) What is the tolerance band for the attribute? Is the measured dimension within the tolerance band? (Is the attribute acceptable?) What is the target value for the attribute? Is the measured attribute getting close to a tolerance limit? If so, how much adjustment will be required to bring the attribute back to the target value (after adjustment)? Is the cutting tool that machines this attribute getting dull? If so, what will have to be done to the adjustment after the tool is replaced?

These decisions are related to *sizing* – keeping workpiece attributes acceptable during a production run. Also explain *trial machining* – a technique that can be used when an attribute's tolerance is so small that the setup person is worried that the initial adjustment is not accurate enough.

**At the machine (10 minutes)**

Show students the various offset pages for the machine/s they will be working on. Demonstrate how offsets are entered (again provide a step-by-step procedure for entering offsets).

Reiterate the importance of knowing how to determine offset adjustments. Anyone can follow the procedure to enter an offset value. It takes more of an understanding to know *the value* to enter.

**Lab exercise**

We have no recommendations for lab exercises for this segment of the course.

**Homework**

▪ Read all of lesson ten, beginning on page 119 in the student manual.

**Exercise (about 10 minutes if done in class)**

Do exercise number ten in the workbook

## Notes:

Present tool length compensation, which is used in every tool of every program. | 25 minutes lecture time

Lessons in Key Concept #4 (you are here):
- 4: Know the compensation types
  - 10: Introduction to compensation
  - **11: Tool length compensation**
  - 12: Cutter radius compensation
  - 13: Fixture offsets

Solicit questions about previous topics. Review lesson ten.

Point out that cutting tools vary in length. And it is difficult (if not impossible) to know the precise length of each cutting tool as a program is written. Indeed, programs are often prepared long before they are needed at the CNC machine.

The feature tool length compensation will allow the programmer to write programs without knowing the lengths of cutting tools.

**Main topics for this lesson:**

**Lesson 11** Presentation links

- Reasons for TLC
  - Tools are different lengths
  - Sizing and tool wear
  - Trial machining
- Two ways to use
  - Offset is tool length
    - Measuring on machine
    - Measuring off line
    - Entering offsets from program
    - Review of alternatives
  - Offset is tool tip to work surface
- Programming TLC
  - Words involved
  - Example program
- Typical mistakes
  - Forgetting to instate
  - Forgetting to enter offset
- Sizing and trial machining

**Pages 125-138** in the student manual

**Lesson objective:** To help students understand tool length compensation, including how it is programmed as well as how it is used at the machining center.

When working on practice programs, students have seen the command to instate tool length compensation (the G43 command). They may have questioned you about this command already. In lesson 11, you'll be presenting the details of how tool length compensation is used.

- You'll be describing the full use of tool length compensation in this lesson, including programming, setup, and operation related topics. For this reason, much of this lesson will apply to anyone that works with a CNC machining center.
- Remind students that even though some topics are more related to setup and operation, a programmer must know enough about making setups and running the machine to direct setup people and operator.

**Key points to make for each topic:**

**Reasons for tool length compensation**

- The slide show helps you present the reasons why tool length compensation is required: cutting tools can be of any length, sizing and tool wear adjustments for depth attributes (like pocket depths), and trial machining for depth attributes.
- Point out that tool length compensation is required for every tool in every program, meaning that students *must* thoroughly understand and master this feature.

**Two ways to use**

- We include in the slide show and in the student manual the two most popular methods to use tool length compensation.
- To avoid confusion, we recommend that you show only one of these methods.
- Generally speaking, companies that have a lot of repeated jobs (like product-producing companies) often have several people involved with the tasks related to making setups. These companies tend to prefer the first method (tool length is the offset value).
- Companies that have but one person performing all of the tasks related to making setups (like many contract shops) tend to use the second method (distance from tool tip to program zero is the offset value).
- Note that the first method shown has several advantages over the second method (tool length compensation values can be determined off line, tool length compensation values for tools used in consecutive jobs need not be re-measured or re-entered, cutting tools can be used on different machines with the same tool length compensation values, and multiple identical cutting tools can be kept ready for action). We strongly recommend that you teach the first method – the tool length compensation value is the tool's length.

**Offset is the tool's length**

- The slide show helps you illustrate that with this method, the program zero assignment value in Z (fixture offset Z value) is the distance from the spindle nose to the Z axis program zero point (a negative value). The slide show also shows how to measure this value right on the machine after the workholding setup is made.
- We also help you illustrate that the tool length compensation value (the tool length compensation offset value for the tool) is the distance from the tool tip to the spindle nose (a positive value). You will also show how this value can be measured right on the machine and then entered into the tool length compensation offset.
- Next, you'll show how tool length compensation values can be measured off line, in preparation for an up-coming job. We even show how a special program can be created that enters tool length compensation values.

**Offset is distance from tool tip to program zero in Z**

- Again, we recommend showing but one method. This is not our recommended method.
- The slide show helps you illustrate that the program zero assignment value in Z (the fixture offset Z) will be zero when using this method.
- We also help you illustrate what the offset value represents if this method is used – as well as how to measure and enter it.

**Programming tool length compensation**

- Point out that programming remains exactly the same regardless of which method (shown above) is used. The only differences are related to the fixture offset Z value and the value stored in the offset – both of which are unrelated to programming. (More on the next page.)

▪ The slide show helps you explain the words involved with tool length compensation, including G43 and the H word. Also, explain that tool length compensation is instated during each tool's first Z axis movement (its approach movement).

▪ The offset number used with each tool should in some way be made to correspond to the tool station number. We recommend using the same offset number as tool station number (tool one – offset one, tool two – offset two, and so on).

▪ An example program helps you illustrate the programming of tool length compensation.

▪ Explain that since tool length compensation remains in effect until the next tool's instating command, there is no need to cancel it (though there is a G code to do so – G49).

**Typical mistakes**

▪ The slide show helps you warn students about mistakes they'll be prone to making – forgetting to instate it and forgetting to enter offset values. We also show the consequences of these mistakes.

**Sizing and trial machining**

▪ This is of special importance to setup people and operators. Point out that some depth dimensions are critical (having a very small tolerance).

▪ When a setup person is worried that an initial tool length compensation setting isn't accurate enough (this is commonly the case when a depth tolerance is less than about 0.001 inch or so), they can use trial machining to ensure that the cutting tool will not remove too much material (scrapping the workpiece) the first time it machines.

▪ The student manual and slide show help you present the five steps to trial machining.

▪ Point out that even with more open tolerances, it is commonly necessary to make a tool length compensation offset adjustment after machining the first workpiece. While the machined attribute will be within the tolerance band on the first workpiece, it may not be right at the target value. An adjustment can be made to ensure that the attribute will be right at the target value for the *next* workpiece to be machined.

▪ Students must understand that as cutting tool's wear, they tend to leave more material on the surfaces they machine. If tolerances are tight, a machined surface may shrink or grow out of its tolerance band before the cutting tool gets dull. If it does, a sizing adjustment (possibly several of them) will be necessary during the cutting tool's life.

▪ Eventually all cutting tools will wear out and must be replaced. If trial machining was needed for a cutting tool during setup, it will be necessary every time the cutting tool is replaced.

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

As stated, much of this lesson is related to setup and operation – especially setup. About the only part of this lesson you can skip are the topics *Programming tool length compensation* and *Typical mistakes*.

You'll want to place a strong emphasis on how tool length compensation values are measured – on the machine and/or off line.

You'll also want to emphasize the sizing and trial machining discussions – providing practice to confirm that students understand how to make trial machining and sizing adjustments after the initial tool length compensation offset is determined.

## At the machine (15 minutes)

There are several things you can do at the machine to demonstrate the principles presented in this lesson.

▪ Using a step-by-step procedure made for your machine, demonstrate tool length compensation offset measurement right on the machining center/s in your lab. Be sure to include how tool length compensation values can be transferred directly into offsets right from the relative display screen page.

▪ If you have an off-line tool length measuring device, demonstrate its use. Be sure to show how tool length compensation offsets are entered into the machine.

▪ Students will be working on two programs in this lesson, one in the homework assignment and one in the workbook. Using one of these programs – or one of your own – have them assemble the related cutting tools and take turns measuring them on the machine (using your step-by-step procedure for doing so). As always, watch them carefully to avoid mistakes. When they're finished, run the program for them. Point out variations in the Z axis positioning based upon the length of each tool being used (shorter tools will cause the Z axis to move further down on a vertical machining center).

## Lab exercise

While you must continue to watch students carefully, you may want to have them practice measuring the lengths of a few cutting tools. Have them write down each tool length compensation value and submit them for grading.

### Homework

▪ Read all of lesson eleven, beginning on page 125 in the student manual.

▪ Have students fill in the blanks for the program on page 137 of the student manual.

### Exercise (about 45 minutes if done in class)

Do exercise number eleven in the workbook. Since they must work on an actual program, this exercise will take some time to complete. You may elect to have them do this exercise as a homework assignment.

## Notes:

Present cutter radius compensation, which is only used with milling cutters. | 30 minutes lecture time

Lessons in Key Concept #4 (you are here):
- 4: Know the compensation types
  - 10: Introduction to compensation
  - 11: Tool length compensation
  - **12: Cutter radius compensation**
  - 13: Fixture offsets

Solicit questions about previous topics. Review lessons ten & eleven.

Point out that just as tool length compensation allows the programmer to write programs without knowing each cutting tool's length, so does cutter radius compensation let them do so without knowing each milling cutter's precise radius.

**Main topics for this lesson:**

**Lesson 12** Presentation links

- Reasons for CRC
  - When to use
  - Simplifies calculations
  - Range of cutter sizes
  - Sizing & trial machining
  - Roughing with finish coord.
- Two ways to use
  - Offset is cutter radius
  - Offset is deviation
- Three steps to using
  - Instate
  - Machine surfaces
  - Cancel
- Examples
  - Mill left side of part
  - Mill outside contour
  - Mill counter-bored hole
- Sizing and trial machining
  - Sizing
  - Trial machining
- Roughing with finish coord.

**Pages 139-164** in the student manual

**Lesson objective:** To help students understand cutter radius compensation, including how it is programmed as well as how it is used at the machining center.

Back in lesson nine (motion commands), students worked on two programs that involved milling cutters (one milling cutter in each program). Remind them that at that time they had to calculate the cutter's centerline tool path – and this complicated the task of calculating coordinates. Every coordinate required them to consider the precise size (radius) of the milling cutter being used.

- You'll be describing the full use of cutter radius compensation in this lesson, including programming, setup, and operation related topics.
- Remind students that even though some topics are more related to setup and operation, a programmer must know enough about making setups and running the machine to direct setup people and operator.

**Key points to make for each topic:**

**Reasons for cutter radius compensation**

▪ Point out that cutter radius compensation is only used for milling cutters – and only when the cutter is machining on its periphery (side milling). This feature is never used for drills, taps, or reamers. Mention that some companies don't do much side milling, meaning they don't have much of a need for cutter radius compensation.

▪ The slide show helps you present the reasons why cutter radius compensation is required: it simplifies the calculations needed for the program, it allows a range of cutter sizes, it allows sizing and trial machining for milled surfaces, and the same set of coordinates used to finish the surfaces being machine can be used for roughing.

▪ Point out that two of these reasons (simplified calculations and rough and finish milling with the same coordinates) apply only to manual programmers.

**Two ways to use**

▪ We include in the slide show and in the student manual the two most popular methods to use cutter radius compensation. To avoid confusion, we recommend that you show only one of these methods.

▪ Generally speaking, manual programmers (those not using a CAM system) prefer to program the work surface path and the offset value used with cutter radius compensation will be the radius of the milling cutter being used. We recommend that you teach this method.

▪ Computer aided manufacturing (CAM) system programmers often prefer to let the CAM system generate a tool path based upon the milling cutter's centerline path. The offset is the radial difference between the planned cutter size (the size the programmer expects the setup person to use) and the milling cutter actually being used.

**Three steps to using**

▪ This series of slides lets you illustrate the three steps to programming cutter radius compensation – instate it, machine the required surfaces, and cancel it. The most difficult step is instating. We provide several clarifying slides to help you explain.

▪ For instating, explain the two related G codes – G41 and G42. One easy way to explain (if students have basic machining practice experience) is that climb milling requires G41 to instate and conventional milling requires G42. We also provide some slides to help you explain G41 and G42 even if students don't know the difference between climb and conventional milling.

▪Still part of instating, an offset must be invoked (with a D word) by the instating command. The slide show helps you explain the two most common offset pages of Fanuc controls. With one of them, there is only one offset register per offset. With this style of offset table, the offset corresponding to the tool station number is already being used with tool length compensation. Explain that another offset must be chosen – we recommend adding a constant number (like 30) to the tool station number to come up with the offset number used with cutter radius compensation. With the other offset table, each offset has two registers – one for the tool's length and the other for its radius. With this offset table, have students simply use the offset number corresponding to the tool station number.

▪ Still for instating, explain the "prior position" – the XY location just before cutter radius compensation is instated. The slide show helps you describe this position.

▪ For step two, machining surfaces, the slide show provides several examples – and has some limitations and warnings.

▪ For canceling (step three), the slide show includes our recommendation to cancel cutter radius compensation and lets you show some examples.

**Examples**

▪ The slide show helps you show three complete examples of cutter radius compensation.

**Sizing and trial machining**

▪ This is of special importance to setup people and operators. Point out that just like tool length compensation allows the setup person or operator adjust depth attributes, so does cutter radius compensation allow them to adjust XY milled surfaces. Many of the same principles presented in lesson eleven apply to cutter radius compensation.

▪ Let students know that making offset adjustments with cutter radius compensation offsets can be confusing. The offset contains the milling cutter's *radius*. Often a milling cutter will mill completely around a workpiece, meaning the measurement being taken is over two milled surfaces. This requires the offset to be modified by *half* the deviation.

▪ The slide show helps you present trial machining and sizing for cutter radius compensation.

▪ As you did for tool length compensation, be sure to quiz students with examples of making offset adjustments.

▪ Also as with tool length compensation, the size of the tolerance determines whether trial machining is necessary whenever a new milling cutter is used – and whether any sizing adjustments will be necessary during the tool's life.

**Roughing with finishing coordinates**

▪ The slide show helps you illustrate how one set of coordinates (the finishing coordinates) can be used for both roughing and finishing. Point out that the setup person will "lie" to the control about the size of the rough milling cutter's radius, making it larger than the cutter's actual radius. The amount larger will be how much finishing stock the roughing cutter leaves. For example, if using a 1.0 inch diameter cutter (0.5 radius) and if you want to leave 0.03 for finishing, the rough milling cutter's offset must be set to 0.53.

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

As stated, much of this lesson is related to setup and operation – especially setup. Be sure to present the reasons for cutter radius compensation as well as the discussion for sizing and trial machining.

You'll want to place a strong emphasis on how milling cutters are measured – commonly with a micrometer.

You'll want to emphasize the sizing and trial machining discussions – providing practice to confirm that students understand how to make trial machining and sizing adjustments after the initial cutter radius compensation offset is determined.

## At the machine (15 minutes)

There are several things you can do at the machine to demonstrate the principles presented in this lesson.

▪ Using a step-by-step procedure made for your machine, demonstrate the entry of cutter radius compensation offsets.

▪ Students will be working on two programs in this lesson, one in the homework assignment and one in the workbook. Using one of these programs – or one of your own – have them assemble the related cutting tools and take turns measuring their lengths on the machine (using your step-by-step procedure for doing so). Have them measure the radius of each milling cutter and enter them into the appropriate offset register. As always, watch them carefully to avoid mistakes. When they're finished, run the program for them. Point out the variations in the XY axis motions based upon the size of milling cuter being used (the value in the offset).

## Lab exercise

You can students practice measuring the radius of a few milling cutters. Have them write down each cutter radius and submit them for grading.

### Homework

▪ Read all of lesson eleven, beginning on page 139 in the student manual.

▪ Have students fill in the blanks for the program on page 163 of the student manual.

### Exercise (about 45 minutes if done in class)

Do exercise number twelve in the workbook. Since they must work on an actual program, this exercise will take some time to complete. You may elect to have them do this exercise as a homework assignment.

## Notes:

Extend what students know about fixture offsets from lesson six. | 30 minutes lecture time

Lessons in Key Concept #4 (you are here):

- 4: Know the compensation types
  - 10: Introduction to compensation
  - 11: Tool length compensation
  - 12: Cutter radius compensation
  - **13: Fixture offsets**

Solicit questions about previous topics. Review lessons ten, eleven, and twelve.

Most of what students need to know about fixture offsets is presented in lesson six. Indeed, if only one coordinate system is required per program (which is commonly the case), it is possible to skip this lesson if you're running short on time.

**Main topics for this lesson:**

**Lesson 13**

## Presentation links

- Review of fixture offset usage
- Multiple program zero points
- Shifting point of reference
- Working with sub-plates
- Repeated setups
  - Using G10 to enter offsets
- Common scenarios
- Example program
- How many fixture offsets do you have
  - 48 fixture offset option

**Pages 165-174** in the student manual

**Lesson objective:** To complete your students' understanding of fixture offsets.

Students know that program zero must be assigned before a program can be run – and fixture offsets are used to assign program zero. To this point, only one program zero point (coordinate system) has been required per program, and students know how to assign it. In this lesson, you'll be showing how to assign multiple program zero points – and you'll show some time-saving techniques that will apply if setups are qualified and/or predictable.

▪ These topics may not be of immediate concern to students – if time is limited, you may wish to skip this lesson. If time permits later in the class, you can come back and present it.

### Key points to make for each topic:

**Review of fixture offset use**

▪ The slide show helps you review the main points from lesson six – showing how program zero is assigned when only one coordinate system is required in the program.

**Multiple program zero points**

▪ We include two ways of handling multiple program zero points in the slide show. The first is used when there is no relationship from one program zero point to another, as would be the case when two or more vises are used to hold workpieces machined by one program. With this method, assigning multiple program zero points is simply an extension of assigning one program zero point. Each set of program zero assignment values is measured and entered just as for one program zero point – there's simply more work for the setup person to do.

▪ The second method – which is also nicely illustrated in the slid show – is used when the setup person knows the distances from one program zero point to the next. This method involves using the *common fixture offset*, which shifts the point of reference from the zero return position to a more logical location within the setup (commonly one of the program zero points used for one of the programs.

**Shifting the point of reference**

▪ This discussion is actually part of the second method for using fixture offsets just mentioned. Point out that the zero return position doesn't always make a very logical point of reference for fixture offset entries. The method described above is one such time – is the next topic – working with sub-plates.

**Working with sub-plates**

▪ The slide show lets you present another excellent application for shifting the point of reference for fixture offset entries. Sub-plates are commonly used on vertical machining centers to minimize workholding setup time. But with a little ingenuity, they can also eliminate all of the tasks related to assigning program zero. The slide show helps you illustrate how.

**Repeated setups**

▪ Point out that companies that have a lot of repeated jobs strive to qualify their setups. This ensures that the workholding device will be placed on the machine table in exactly the same location every time the setup is made – which also means the program zero location is in the same location every time the setup is made. If this is done, the program zero assignment values used the last time the setup was made will work the next time the setup is made.

▪ Explain that one way to retain program zero assignment values from one time a job is run to the next is to write them down.

▪ A more efficient way is to include program zero assignment values in the program. That is, commands to assign program zero (enter fixture offsets) can be included in the program with G10 commands. The slide show helps you illustrate how G10 is used.

**Common scenarios**

▪ Admittedly, these discussions may be confusing to students. Which method should they use? We provide some scenarios to help you illustrate which method of program zero assignment is best for a given application.

**How many fixture offsets do you have?**

▪ Point out that Fanuc controls come standard with six fixture offsets (which is usually enough).

**How many fixture offsets do you have?**

▪ Point out that Fanuc controls come standard with six fixture offsets (which is usually enough). But more can be purchased as an option. The slide show helps you show how this fixture offset option is programmed.

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

As stated, this lesson may not be of immediate importance – even to setup people and operators. If there is ample time, it does help if setup people and operators understand the various ways that fixture offsets can be used. But if you know they will be commonly working with but one program zero point program, we recommend skipping this lesson.

## At the machine (10 minutes)

Call up the fixture offset page and point out the common offsets, which normally has all registers set to zero. Again, point out that the values in this fixture offset specify how far from the machine's zero return position is the point of reference for fixture offset entries – when set to zero, the zero return position *is* the point of reference for fixture offset entries.

You might want to plug in some values and run an example program (or make some manual data input commands) to demonstrate what will happen when the point of reference for fixture offset entries is shifted.

## Lab exercise

We have no suggestions for a lab exercise related to the material in this lesson.

## Homework

▪ Read all of lesson eleven, beginning on page 165 in the student manual.

## Exercise (about 40 minutes if done in class)

Do exercise number thirteen in the workbook. There is a programming activity in this exercise – which will take some time to complete.

## Notes:

## Review questions for Key Concept number four

***Lesson ten – Introduction to compensation:***

▪ Why are compensation types required on CNC machining centers? (You can ignore certain tooling-related concerns while writing the program.)
▪ What is trial machining? What is sizing?
▪ What is a target value?
▪ How do you determine whether trial machining is required?
▪ How do you know if sizing is required during a tool's life?
▪ Where are compensation values placed?

***Lesson eleven – Tool length compensation:***

▪ Why is tool length compensation required?
▪ How often is tool length compensation used?
▪ When is tool length compensation instated? What words are used in the instating command?
▪ How do you determine the offset number (H word) used with tool length compensation?
▪What value is actually stored in the tool length compensation offset?
▪ What value is stored in the Z register of the fixture offset?

▪***Lesson twelve – Cutter radius compensation:***

▪ What kind of cutting tools require cutter radius compensation?
▪ Name the four reasons cutter radius compensation is required. (Simplifies programming, allows a range of cutter sizes, allows trial machining and sizing, and finishing coordinates can be used for roughing).
▪ Name the three steps to programming cutter radius compensation.
▪ How do you determine the offset number used with cutter radius compensation? What value is placed in this offset?
▪ If more material must be removed by the milling cutter, which way (plus or minus) must the offset value be adjusted?

***Lesson thirteen – Fixture offsets:***

▪ Why are fixture offsets required?
▪ How many fixture offsets are available?
▪ Name the two ways fixture offset values can be determined.
▪ Describe how fixture offsets should be used if you're using a sub-plate.
▪ How can fixture offset values be retained for the next time the job is run?

Introduce Key Concept number five. | 5 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

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.

To this point in the class, you have been presenting the building blocks needed to write CNC programs. Students have worked on several programs, filling in the blanks for important words and commands – but they have not written a program from scratch.

In Key Concept number five, you're going to show them how to become self-sufficient CNC programmers – able to write programs on their own.

Since this Key Concept requires an understanding of everything you've presented so far, now is a great time to do a lengthy review – make sure your students are truly ready for Key Concept number five.

**If you are only presenting setup and/or operation**

- Truly – this Key Concept is devoted to programmers. But as you will point out during this Key Concept, the method by which a CNC program is written has a big impact on how cutting tools must be re-run by setup people and operators. Indeed, if the program is not properly formatted, it may be impossible to rerun a given cutting tool by itself (without running the entire program).
- Setup people and operators must know enough about the program's structure to know the *restart block* from which cutting tools begin.

**Lessons related to this Key Concept:**


Begins on **page 175** in the student manual

Two Lessons

14 - Program formatting
15 - Four kinds of program format

**Key Concept objective:** Help students master the ability to write programs by themselves.

Again, students have worked on several programs to this point in the class. They may have noticed quite a bit of structure and consistency among these programs. Point out that the practice programs have used the same structure you're going to be presenting here in Key Concept number five – so things should look pretty familiar.

- Students may have been questioning you when working on practice programs about words and commands that you had not yet explained. If you've explained some of these commands, students will be even more familiar with the material in this Key Concept.

**Key points to make while introducing to Key Concept number four:**

- Name a few programming words: G00, G01, G02, G03, M03, the O word, F word, S word – and so on. See if students can describe them. It's likely that they can. Point out that they already know quite a bit about the structure related to writing CNC programs.
- Remind students that learning to write CNC programs is like learning a foreign language that has only about 50-60 words. They've already learned the majority of these words.

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

Again, the majority of this Key Concept is related to programming. Setup people and operators should be familiar enough with a program's structure to be able to re-run tools.

### At the machine

We have no suggestions.

### Lab exercise

We have no recommendations for lab exercises for this segment of the course.

### Homework

- Read the introduction to Key Concept number five that begins on page 175 in the student manual.

### Exercise

None for this segment of the course.

## Notes:

Explain why programs must be formatted using a strict and consistent structure. | 10 minutes lecture time

Lessons in Key Concept #5 (you are here):

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

Solicit questions about previous topics. If you haven't already, do a lengthy review of all programming topics presented to this point.

Point out that many things are dictated by the way programs are formatted. It's best if *we* control the objectives that programs achieve (safety, ease of use, efficiency, etc.).

**Main topics for this lesson:**

**Lesson 14** Presentation links

- Key concept number five
- Importance of formatting
  - Familiarization
  - Consistency
  - Rerunning tools
- Four types of format
- Machine Differences
  - Automatic tool changers
  - M codes
  - Accessories
- Efficiency improvements

**Pages 177-186** in the student manual

**Lesson objective:** Ensure that students understand why programs must be strictly formatted to achieve the desired objectives.

Point out that only the structure by which programs are formatted dictates how safe, easy to use, and efficient programs will be. Unfortunately, what is done to enhance one objective commonly detracts from the others. Our emphasis will be on safety and ease of use. So our recommended formats will not be as efficient as possible.

- We'll be reviewing certain format related topics as well as presenting a few new ones in this lesson.

**Key points to make for each topic:**

**Key Concept number five**

▪ You have already introduced Key Concept number five.

**Importance of formatting**

▪ Begin by presenting the three reasons why CNC programs must be strictly formatted: To allow familiarization with programming, to ensure consistency among programs, and most importantly, to allow cutting tools to be run by themselves (actually, to be re-run).

▪ For familiarization, the slide show helps you present a simple analogy related to driving an automobile. It is unlikely that any driver can recite from memory all of the road signs – but when they see a road sign, they quickly recognize its meaning. In similar fashion, few CNC programmer can recite from memory all CNC words used in programming. But when they see the word – especially when it is in the correct context of a CNC program, they will easily recognize its use.

▪ For consistency, point out that programmers must be able to repeat past successes. If a given format works properly, achieving all required objectives, using its format in future programs will ensure continued success. Also, setup people and operators (and anyone else working with programs) will quickly become familiar with programs if they are consistently formatted.

▪ For re-running tools, the slide show provides several examples of why strict formatting is important. Point out that each tool in the program should be treated as a mini-program. All words and commands necessary to get the machine running (the same words and commands as for the first tool) must be programmed at the beginning of each tool – making the tool independent from the rest of the program. Sometimes this means programming seemingly redundant words and commands.

**Four types of format**

▪ Introduce the four types of program format that will be presented in lesson fifteen: Program startup format, tool ending format, tool startup format, and tool ending format. Explain how these formats can be used as a crutch until they are memorized.

**Machine differences**

▪ Point out some of the machine differences that require different program formatting – like automatic tool changer differences and differences in M code numbering.

▪ All programs shown in this class stress the most popular kind of automatic tool changer – the double arm style. The slide show helps you describe it in detail.

▪ The slide show also helps you present the single arm automatic tool changer – but to avoid confusion – only present the style that is used by your lab machines.

**Efficiency improvements**

▪ As stated, our formats for programming emphasize safety and ease of use. This section of the slide show helps you present some efficiency related limitations of our given formats – and provides suggestions for improvement.

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

As mentioned, this lesson is most related to programming. However, setup people and operators must know enough about program structure to be able to pick out the restart block for each tool.

Rerunning tools, of course, is often required – point out when (when verifying programs and when trial machining, for example). With double arm automatic tool changers, the restart block will depend upon whether or not the cutting tool to be re-run is in the spindle or not. With our given formats, if it is, the restart block will be the command *after* the M06 command that places the tool in the spindle. If it is not, the restart block is the M06 command itself.

## At the machine (20 minutes)

Pick one of the practice programs students have done in class and load it into the machine. Point out the strict structure used. Show students the restart block for each tool. Run the program once to show students the motions made by each tool.

Using a written step-by-step procedure for re-running tools, demonstrate the task of re-running the tools in the program. If you have a double-arm automatic tool changer, be sure to point out the difference between rerunning a tool that is already in the spindle (restart block is the command after the M06) and rerunning a tool that is not currently in the spindle (restart block is the M06 command.

Have students take turns practicing rerunning tools (again, using your written procedure to do so.

## Lab exercise

We have no recommendations for lab exercises for this segment of the course.

### Homework

- Read all of lesson fourteen that begins on page 177 in the student manual.

### Exercise (about 40 minutes if done in class)

Do exercise number fourteen in the workbook. There is a programming activity in this exercise – which will take some time to complete.

## Notes:

Introduce the four types of program format and show students how to use them. | 20 minutes lecture time

Lessons in Key Concept #5 (you are here):
- 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

Solicit questions about previous topics. Review the reasons why programs must be strictly formatted.

Students now know why programs must be strictly formatted – in this lesson you're going to show *how*.

**Main topics for this lesson:**

**Lesson 15**

## Presentation links

- A few more notes
  - G92 vs fixture offsets
  - Safety commands
  - Documentation
  - How G28 works
  - Using G53
  - When to tool change
- Program formats
  - Vertical MC using G92
  - Vertical MC using fixture offsets
  - Horizontal MC using G92
  - Horizontal MC using fixture offsets
- Example program
  - Points about example program

**Pages 187-198** in the student manual

**Lesson objective:** Help students become self-sufficient programmers.

Again, students have worked on several programs – but they haven't written one on their own yet. They should know the majority of CNC words and commands used in programs, so you shouldn't have too many new ones to describe in this lesson. Concentrate on providing students with the formats – crutches they can use until they have the formats memorized.

▪ Point out that students should only be on their own to develop the commands that perform the machining operations in the program. The balance of any program is related to format.

**Key points to make for each topic:**

**A few more notes**

▪ The slide show helps you make a few more points about program structure before you show the actual formats.

▪ If students have been doing the exercises and programming activities, it's likely that you have already explained some of these points.

▪ The slide show does include a nice presentation about how G28 (the zero return command) works – including several examples and warnings. This command tends to be the most confusing and misunderstood G code from Fanuc.

**Program formats**

▪ We provide four sets of format in the slide show and student manual: for vertical and horizontal machining centers with and without fixture offsets. To avoid confusion, pick and present only the one that applies best to your lab machine/s (probably for vertical machining centers using fixture offsets).

▪ When you show the formats for the first time, point out that certain values (numbers) within CNC words will change from tool to tool and program to program. But the structure will remain the same. The slide show helps you nicely illustrate this with color coding. In the student manual, changing values are provided in bold faced fonts.

▪ As you come across any new word, of course, you must explain it (like M01, M30, and M19). This should finalize any concerns and questions that students have about the most common CNC words.

▪ With an understanding of these formats, writing a program will be a (relatively) simple matter of beginning with the program startup format. The student will be on their own to develop the commands to machine with the first tool. Then they follow the format to end the tool (tool ending format). Next they follow the format to start the next tool (tool startup format). They're on their own again to develop the commands to machine with the second tool. They'll toggle among tool ending, tool startup, and cutting commands until they're finished with the program – at which time they'll follow the program ending format. Again, they're only on their own for the cutting commands with each tool.

**Example program** Use the example program to make these points:

▪ Certain seemingly redundant words are required from tool to tool. If, for example two tools run in sequence that require the same spindle speed or feedrate, the related words (S and F) must be specified in both tools. The slide show helps you make this point.

▪ Much of a typical program is simply format that can be copied from an existing program. (There are only six cutting commands in the example program – the rest of the program is related to format.).

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

This lesson is devoted entirely to programming. Other than continuing your presentations and practice related to re-running tools, we don't have any suggestions for setup and operation related to this lesson's content.

## At the machine (20 minutes)

Students will be writing their first program entirely on their own. This will make a great program to work with at the machine.

**Warning:** To this point, programs that you have worked with on the machine have been previously verified (having no problems). This will be a student's program, and as such, may include mistakes. If your facility has some form of tool path verification system – either off line or part of the CNC machine – be sure to use it prior to actually running the program on the machine. Even then, be ready for anything when you do run the program.

Actually, it can be helpful if the program does include some mistakes. While you're not yet describing program verification (that's coming later, in lessons 20 and 24), it doesn't hurt to allow students to see what they'll be up against when running their own programs.

After the program has been verified – and using a step-by-step procedure for running programs, have students practice running the program. And using the procedure to re-run tools, have them practice running one tool at a time (using optional stop to stop the machine after each tool.

## Lab exercise

We have no recommendations for lab exercises for this lesson.

### Homework

- Read all of lesson fifteen that begins on page 187 in the student manual.
- Write the program for the job shown on page 197 of the student manual.

### Exercise (about 50 minutes if done in class)

Do exercise number fifteen in the workbook. There is a programming activity in this exercise – which will take some time to complete.

## Notes:

Introduce Key Concept number six. | 10 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

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. There are ten Key Concepts further divided into twenty-four lessons.

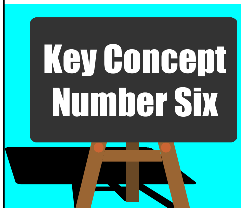
At this point, students should be able to write programs on their own. But they have just the rudimentary tools to do so. Point out that writing programs with only the tools you have seen so far will be quite tedious (I like to point out, for example, that drilling 50 holes using only G00 and G01 will take at least 150 commands).

In Key Concept number six, you'll be showing several features that shorten programs, make programming easier, and in general facilitate the programming process.

A good review of all material presented so far may be in order. Confirm that students are truly ready to learn about some rather advanced CNC programming features.

**If you are only presenting setup and/or operation**
- Like Key Concept number five, this Key Concept is devoted to programmers. But point out that certain programming features not only simplify programming, they make it easier to verify the program's correctness during setup – meaning setup people may have some interest in many of these features.

**Lessons related to this Key Concept:**




Begins on **page 199** in the student manual

Four Lessons

- 16 - Canned cycles
- 17 - Subprogramming
- 18 - Other special features
- 19 - Rotary device programming

**Key Concept objective:** Help students master those special programming features that are appropriate to their needs.

Point out that while it is important for students to understand the various special programming features that are available, not all will be of immediate importance. What may be quite helpful and often-used for one programmer may never be needed by another. You'll be placing an emphasis on the most popular features.

- Especially in the area of hole machining, students may have been questioning the difficulty related to drilling several holes – they may have asked if there's an easier way. In this Key Concept, you're going to be showing the easier ways to accomplish programming tasks.

**Key points to make while introducing to Key Concept number four:**

- Canned cycles will dramatically simplify the machining of holes. There will be but one command per hole – regardless of the kind of machining operation (drilling, tapping, peck drilling, etc.). And once a cycle is instated (for the first hole), machining other holes is a simple matter of listing the coordinates for each hole.
- Subprograms are used to keep from having to write a series of commands more than once.
- There are several other special features that can help with programming.
- You'll be showing the special considerations related to rotary device (rotary axes and indexers) in this Key Concept.

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

Again, the majority of this Key Concept is related to programming. If a company's setup people will be expected to modify programs during program verification, they should be familiar with the programming of some of these features.

### At the machine

We have no suggestions.

### Lab exercise

We have no recommendations for lab exercises for this segment of the course.

### Homework

▪ Read the introduction to Key Concept number six that begins on page 199 in the student manual.

### Exercise

None for this segment of the course.

## Notes:

Help students master these helpful hole-machining canned cycles. | 20 minutes lecture time



Solicit questions about previous topics. If you haven't already, do a lengthy review of topics presented to this point – especially the motion types.

Student know how to program the drilling of holes using G00 and G01. Now you'll show them how to easily program any hole machining operation – with but one command per hole.

**Main topics for this lesson:**



**Pages 201-222** in the student manual

**Lesson objective:** Help students master the programming of any hole machining operation.

Students have worked on several programs that machine holes (center drilling and drilling). But they've used G00 and G01 to do so. Now you'll help them master the programming of any hole machining operation – and show them how simple it can be with canned cycles (G73-G89).

- Point out that but one command per hole will be required.
- Once the first hole is programmed, programming other holes of the same type is as simple as listing hole coordinates.
- When finished programming a series of holes, the cycle is cancelled (with G80).
- Students may be wondering why you waited so long to show canned cycles. Don't let them minimize the importance of what they've learned so far.

**Key points to make for each topic:**

**Key concept #6**
- The previous two pages discuss how to introduce lesson six.

**Introduction to canned cycle commonalities**
- Explain what a "canned cycle" is (a series of pre-determined motions specified by one program command). I like to review the G28 command, saying that it is a kind of canned cycle. One command actually does two things (the motion to the intermediate position and then the motion to the zero return position). In fact, if the single block switch is on, the cycle start button must be pressed twice to complete a G28 command. In similar fashion, all hole-machining canned cycles will cause the machine to make several motions (at least three) from one command in the program.
- Point out that canned cycle share two things in common. First, they are all modal. Once a canned cycle is instated, it will remain in effect until it is cancelled (with G80).
- Second, all hole-machining canned cycles will cause four basic motions: 1) rapid to XY position. 2) rapid to R plane. 3) machine the hole. 4) retract from the hole. The slide show helps you illustrate these four motions.

**Description of each canned cycle**
- The slide show for this lesson provides you with a series of slides to show the exact motions for each of the canned cycle in order of popularity. That is, we show the most popular canned cycles first.
- During this slide show, you'll be able to visually introduce all of the words related to each canned cycle. Be sure to point out the consistency for word use from one canned cycle to another (especially X, Y, R, Z, and F)

**Simple example**
- Next, the slide show helps you present a simple example to help students understand the points you've made so far.

**Understanding G98 and G99**
- This can be a little difficult for you to present and for students to master. Point out that G98 and G99 (initial plane retract and R plane retract) allow the programmer to easily clear clamps and other obstructions (in Z) between holes.
- Explain that G98 specifies that, after the hole is machined, the tool will retract to the *initial plane*. The initial plane is defined as the tool's last Z position prior to the canned cycle command. G99 specifies that the tool will retract to the rapid plane (specified in the canned cycle command itself.
- The initial plane is usually specified right in the tool's first Z axis approach motion (the G43 command). If clamps are to be avoided, this Z position must be above the clamps (usually 2.0 inches above the work surface is sufficient.
- If no obstructions exist, the programmer can rapid the tool right to the R plane. In this case the rapid plane and the initial plane will be the same.
- We like to see students placing the G98 or G99 word at the end of the command to help remind them that these words control what happens *after* the hole is machined.
- G98 and G99 are modal. They are canceled when the G80 word (canned cycle cancellation) is executed. G98 is the initialized state (and reinstated at G80), meaning obstructions will be avoided if no G99 or G98 is specified.
- Several examples in the slide show and in the student manual illustrate the use of G98 and G99.

**Canned cycles and the Z axis**
- Point out that the R and Z word in the canned cycle command are absolute values (if working in the absolute mode – G90). We provide a series of slides to help you illustrate how to program holes into different Z surfaces.

### Using the incremental mode

- Let students know that if they have a series of equally spaced holes to machine (like a grid pattern of holes), it may be easier to program them using the incremental mode. Explain that the meanings of X, Y, R, and Z change in the incremental mode (the slide show helps you illustrate how.
- An L word in the canned cycle command specifies the number of holes to be machined by the command.

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

While setup people won't have to program canned cycles, they may have to edit canned cycle commands during program verification. Since they're relatively easy to understand, you may want to present this lesson to setup people.

### At the machine (15 minutes)

In the exercises for this lesson, students will be writing programs that use canned cycles. You can easily use them (or exercises of your own design) for practice at the machine. Be sure to illustrate how easy it is to modify canned cycle commands as the need arises (like changing G81 to G73 and adding a Q word to change from standard drilling to chip-break peck drilling.

Be sure to illustrate all of the most popular cycles (G73, G81, G82, G84, and G86).

### Lab exercise

Since the programs written during the exercises for this lesson contain several tools – and since canned cycles make the program so much shorter – you should be able to continue stressing how to re-run tools. Have students practice finding the restart block for each tool.

You may also want to throw in some "what if" scenarios. For example, what if a counter-bored hole has been machined by a tool in station number five and it is too shallow by 0.005 inch? Ask which offset is involved and how it must be modified to make the tool go 0.005 deeper.

### Homework

- Read all of lesson sixteen that begins on page 201 in the student manual.
- Write the program for the job shown on page 221 of the student manual.

### Exercise (about 30 minutes if done in class)

Do exercise number sixteen in the workbook. There is a programming activity in this exercise – which will take some time to complete.

## Notes:

Present the applications and use of subprograms. | 15 minutes lecture time

Lessons in Key Concept #6 (you are here):

- 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

Solicit questions about previous topics. Quickly review your canned cycles presentation.

Point out that there are times when a programmer must repeat a series of commands within a program. Have them consider, for example, center drilling, drilling, and tapping fifty holes. The list of coordinates needed to center drill will be repeated when drilling and tapping.

**Main topics for this lesson:**

**Lesson 17** Presentation links

- Applications
  - Repeated commands
  - Repeated operations
  - Control programs
  - Utility programs
- Related words
  - Picking program numbers
  - Nest levels
- Examples
  - Multiple operations on holes
    - Using L0
  - Rough & Finish contour milling
  - Identical pockets
    - Using incremental mode
    - Using G52
  - Control programs
- Intro to parametric programming

**Pages 223-236** in the student manual

**Lesson objective:** Help students recognize, understand, and master the applications for subprograms.

You've presented two times when commands in a program must be repeated: multiple operations on holes (center drill, drill, and tap, for example) and when rough and finish contour milling. Point out that any time commands must be repeated, it may be a good application for using subprograms.

- With subprograms, the programmer can cause the machine to exit the main program (temporarily) to execute a subprogram.
- When the machine finishes executing the subprogram, it will return to the main program to the command after the calling command and continue.
- Subprograms are named and loaded in the same way as *main programs* (every program shown to this point is a main program).

**Key points to make for each topic:**

**Applications**

▪ Start by describing the four application categories for subprograms – repeated commands and operations, control programs, and utility programs. The slide show helps you do so.

▪ The slide show starts by showing the application for multiple operations on holes – nicely illustrating how helpful a subprogram can be.

▪ The slide show then shows the application categories, but only lists example applications in each. Later in the slide show, we'll show complete examples.

**Related words**

▪ The slide show then helps you introduce the four words used with subprograms (M98, M99, P, and L).

▪Next, the slide show helps you present a simple example to help students understand the points you've made so far.

▪ Since using subprograms requires multiple programs to be loaded, we provide some suggestions for naming subprograms that will help avoid confusion.

▪ We then show the *nesting* limitation for subprograms (calling one subprogram from another).

**Examples**

▪ The slide show helps you present full examples in all application categories.

▪ One of the categories (repeated machining operations) requires that you introduce the G52 command, which allows temporary shifting of the program zero point. This is required to eliminate the need to program the operation in the incremental positioning mode. The slide show helps you introduce and explain G52.

▪ The *control programs* application example is related to machining centers that have pallet changers. But of course, not all machining centers have this accessory. However, this example should still help students understand control programs.

**Introduction to parametric programming**

▪ Point out that in order to use a subprogram, *all* of the commands in the subprogram must be totally redundant. If anything changes from one time the subprogram is needed to the next, the subprogram cannot be used.

▪ Explain that there is an optional feature called parametric programming (Fanuc's version of parametric programming is custom macro B). While parametric programming is beyond the scope of this class, students should at least be familiar with applications for parametric programming. The slide show helps you introduce the four application categories.

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

It's unlikely that setup people and operators will have much need to understand subprograms. Since setup people do load programs during setup, it may be important for them to know that subprograms exist – but you can limit your presentation of this lesson to explaining what a subprogram is.

### At the machine (10 minutes)

In the exercises for this lesson, students will be writing programs that use subprograms. You can easily use them (or exercises of your own design) for practice at the machine. When executing the program, monitor the PROGRAM display screen page. Be sure to point out that when a subprogram is executed, the main program appears to disappear – and only the subprogram is shown on the display..

### Lab exercise

We have no suggestions for lab exercises that pertain to the subject matter for this lesson.

### Homework

▪ Read all of lesson seventeen that begins on page 223 in the student manual.

### Exercise (about 40 minutes if done in class)

Do exercise number seventeen in the workbook. There is a programming activity in this exercise – which will take some time to complete.

## Notes:

Show a few more programming features that can facilitate programming. | 25 minutes lecture time

Lessons in Key Concept #6 (you are here):
- 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

Solicit questions about previous topics. Quickly review canned cycles and subprograms.

In this lesson, you'll be showing some other helpful programming features – but point out right away that some of these features may not be of immediate use. In fact, some may be very helpful to one programmer but never needed by another. But before your students can begin to apply a feature, of course, they must know it exists.

**Main topics for this lesson:**

**Lesson 18** Presentation links

Optional block skip
- Another optional stop
- Trial machining

N word techniques
- Statement labels
  - Changing machining order

Other G codes of interest
- G02-G03 helical motion
  - Example
- G04 dwell
  - Relieving tool pressure
  - Machine problems
- G09/G61exact stop check
- G10 data setting
- G15-G16 polar coordinates
  - Example

Other G codes (continued)
- G17/G18/G19 plane selection
- G20-G21 inch/metric
  - Metric advantage
- G30 second reference position
- G50-G51 scaling
- G60 single direction positioning
  - Example
- G64 normal cutting
- G68-G69 coordinate rotation
  - Examples
- G94-G95 feedrate mode selection
- G50.1-G51.1 mirror image

Other M codes of interest
- M00 – M02 – M13/M14/M15

Understanding parameters

**Pages 237-258** in the student manual

**Lesson objective:** Helps students recognize and master those special programming features that will be important to them.

At this point, you've presented the most popular programming features. Indeed, most of what you have presented will be of immediate need to the vast majority of CNC programmers. In this lesson, you'll be presenting some more important features, but some of them will not be of immediate need to everyone. For example:

- Optional block skip is a feature needed and used by most companies.
- Statement labels are rarely needed.
- Helical motion is only needed if the company performs thread milling operations.

You can, of course, skip – or minimize presentations for– topics you know students won't need.

**Key points to make for each topic:**

**Optional block skip (block delete)**
- The slide show begins by helping you explain how block delete works.
- You can then get as elaborate as you want showing applications.
- We feel one of the most important applications for block delete is with trial machining – and the slide show helps you describe how block delete can help.

**N word techniques**
- Though not often required, we help you show a technique that can be used when machining order (tool sequence) must be changed. This a rather advanced technique that uses sequence numbers as *statement labels* – and allow a kind of unconditional branch (GOTO) in a CNC program.

**Other G codes of interest**
- There are several G codes that have not been presented to this point in the class. Point out right away that if the G code has not been yet introduced, it is probably not needed on a regular basis. At this point in the class, you'll introduce the lesser needed G codes.
- G02-G03: The slide show helps you provide a good explanation of helical motion as it is used when thread milling.
- G04 (dwell): We recommend limiting applications for G04 to relieving tool pressure (some programmers are too quick to use G04 to program around machine problems – as is explained in the slide show.
- G09, G61: Though not often needed, the slide show does help you fully explain the *exact stop check function*.
- G10: This G code is introduced in Key Concepts one and four. Here the slide show helps you show some advanced techniques for the *data setting command*.

G15, G16: You then show how polar coordinates are programmed. The slide show also helps you introduce limitations for this feature.
- G17, G18, G19: The slide show helps you present the implications of plane selection. Examples include milling with a ball end mill and using right-angle heads.
- G20, G21: We've been mentioning the differences between inch and metric modes throughout the class. In this presentation, we recap – and we show the accuracy advantage of the metric mode.
- G30: Some, but very few, machining centers use the secondary reference point as the tool change position in the Z axis (instead of the zero return – G28 – position). This presentation helps you explain G30 should you need to.
- G50, G51: This presentation lets you introduce the feature *scaling*. Since it is so rarely used, we simply introduce it and show the related words.
- G60, G64: This presentation helps you explain the application and use for single-direction positioning (used when finish boring holes).
- G68, G69: This presentation helps you explain coordinate rotation. A good example is provided to help you illustrate its use.
- G94, G95: Some, but not many, CNC machining centers allow feedrate specification in feed per *revolution* as well as in feed per minute. For those that do, these two G codes are involved (G94 for per-minute and G95 for per-revolution). The slide show helps you illustrate.
- G50.1, G51.1: This presentation helps you explain the applications and limits of *mirror image*.

**Other M codes of interest**

- As with G codes, there are a few more M codes that your students must be exposed to.
- M00: The slide show helps you present the application and use of the *program stop* command.
- M13, M14: Some, but not many, machining centers allow the spindle and coolant to be activated at the same time. This section of the slide show explains them.
- If you know that the machine/s your students will be using have M codes not addressed by this class, you must introduce them here.

**Understanding parameters**

- While parameters have little to do with programming, there are some that affect the way the machine behaves when executing programs. All CNC people should at least know what parameters are. Better, they should know the kinds of functions that parameters control. This section helps you introduce parameters.

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

Some of these features (like block delete) require interaction from setup people and operators. Be sure setup people and operators understand features they will be working with.

## At the machine (10 minutes)

You may want to develop a special "demonstration program" that shows the use of features described in this lesson. But remember that some of these features are options, meaning you'll have to limit your program to showing only those features that are equipped on your machine.

With the machine well off its over-travel limits in each axis, you can easily demonstrate helical motion with the MDI command:

G91 G02 I-1.0 Z-1.0 F20.0

## Lab exercise

The programming activity of this lesson involves writing a program using helical motion to machine a thread. If you have students do this exercise, be sure to have them run the program on your lab machine.

**Homework**

- Read all of lesson eighteen that begins on page 237 in the student manual.

**Exercise (about 30 minutes if done in class)**

Do exercise number eighteen in the workbook. There is a programming activity in this exercise – which will take some time to complete.

## Notes:

Explain how the fourth and fifth axes are programmed. | 15 minutes lecture time



Solicit questions about previous topics.

Students now know how to handle the three most basic axes: X, Y, and Z. These three axes are linear axes. Explain that when a fourth (and possibly fifth) axis is added to a machining center, it is usually a *rotary* axis. And there are substantial differences related to the way these axes are handled.

**Main topics for this lesson:**



**Pages 259-280** in the student manual

**Lesson objective:** Master the programming of a rotary axis.

Point out that with the programming methods you've shown so far, only one side of the workpiece has been exposed to the spindle for machining – and probably only one program zero point has been required (per workpiece). When equipped with a rotary device, the machine can rotate the workpiece to expose several sides to the spindle for machining during the CNC cycle.

- Programs for machines with rotary devices tend to be longer – but no more difficult to write.
- While there are similarities to linear axes, there are also substantial differences.

You can, of course, skip – or minimize presentations for– topics in this lesson if your machining centers do not have rotary devices.

**Key points to make for each topic:**

**Introduction**
- We begin by introducing the two types of rotary devices – indexers and rotary axes.
- An indexer can only rotate to expose a surface for machining. No machining can occur during the rotation.
- A rotary axis can be used as an indexer, but it also allows machining to occur during rotation. That is, the feedrate for rotation can be precisely specified.

**Programming indexers**
- The slide show helps you explain the programming of three indexer types (90 degree indexers, 5 degree indexers and 1 degree indexers).

**Programming a rotary axis**
- The slide show also helps you explain the programming of a full rotary axis (limit your presentation, of course, to the kind/s of rotary devices equipped on your machine/s.
- You'll explain how the letter address (A, B, or C) is chosen based upon rotary axis orientation with the spindle.
- You'll explain polarity.
- You'll explain that a rotary axis, just like a linear axis, has a zero return position. The slide show helps you illustrate this position and how it is programmed with G28.
- As opposed to a linear axis, explain that a rotary axis has no over-travel limits. It can continue rotating an unlimited number of times.
- If a rotary axis is being used as an indexer (rotation *then* machining), we recommend programming it in the incremental mode (the slide show helps you show the somewhat unusual things that can happen in the absolute mode). But we do include a presentation to help you explain the assignment of program zero for a rotary axis.

**Programming a rotary axis**
- We do help you explain both absolute and incremental programming. But again, we recommend programming rotations in the incremental mode – this section helps you explain why.
- We help you explain the use of G00 for indexing (rotation at rapid) as well as G01 for cutting motions.
- Do explain the use of feedrate (F word) when G01 is used for rotation. Feedrate must be specified in *degrees* per minute (not inches per minute). The slide show helps you explain how to calculate degrees per minute feedrate.
- Though it is not described in detail, we do mention the feature *cylindrical interpolation* as well as its application.

**Approaching applications**
- The slide show helps you present the approach for handling rotary device programming. Included are processing considerations, helping students decide where to place a central program zero point, assigning multiple program zero points, and when to change tools.

**Example program**
- The slide show provides you with a sample program, but frankly speaking, the student manual includes a more realistic example. These programs should help students understand the programming of a rotary device.

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

Introduce rotary devices (not discussing programming). Make sure students understand the naming of the rotary device and its orientation on the machine (and polarity).

### At the machine (20 minutes)

Be sure to demonstrate the "unusual" things that can happen in the absolute mode. I like to give the command G90 G00 B0 first (in MDI). (Of course, use the letter address that applies to *your* rotary axis.) The rotary axis will rotate to the zero side. Then give the command G00 B90.0. The axis will rotate ninety degrees. Then G00 B180.0 – ninety more degrees. Then G00 B275.0 – ninety more degrees. Next type G00 B0, but before executing, ask students what will happen.

To their surprise, the axis will rotate 270.0 degrees in the opposite direction. This demonstration nicely illustrates that when going from a small B value to a larger one, the rotary axis will rotate in the positive direction (normally clockwise when viewed from above the axis). But when going from a large B to a smaller one, the axis will rotate in the negative (counter-clockwise direction).

You might also demonstrate the G01 mode – as well as how feedrate is specified in degrees per minute. In MDI, execute the command G91 G01 B90.0 F90.0. Have students note the large F word (F90.0). This motion will take a full minute to execute (ninety degrees of rotation at ninety degrees per minute.

### Lab exercise

In the exercise for this lesson, students write a program that can be used to demonstrate the rotary axis. Have students load and run it. Note that multiple program zero points must be assigned prior to running the program.

### Homework

- Read all of lesson nineteen that begins on page 259 in the student manual.

### Exercise (about 50 minutes if done in class)

Do exercise number nineteen in the workbook. There is a programming activity in this exercise – which will take some time to complete.

## Notes:

Introduce Key Concept number seven. | 10 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

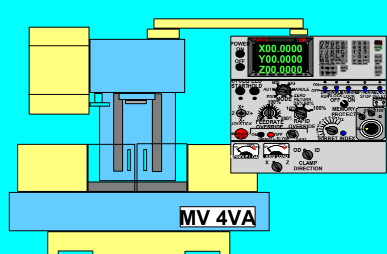
**10: You must know how to safely verify programs**
- 24: Program verification

## Machining Center Operation

Key concept number seven: Know your machine

- Lesson 20: Setup & operation tasks
- Lesson 21: Two operation panels

Begins on **page 281** in the student manual




**Key Concept objective:** Understand the machine from a setup person's or operator's viewpoint.

**If you are only presenting programming**

▪ Again, a programmer must understand enough about setup and operation to direct setup people and operators. While you might consider this class to be over if you're only teaching programming, we urge you to continue. While you can minimize detailed presentations, at least present enough to give programmers a working knowledge of setup and operation.

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. There are ten Key Concepts further divided into twenty-four lessons.

### Introduction to setup and operation

Key Concept number seven formally begins the setup and operation part of this course. However, you've done a great deal during the programming-related lessons to prepare students for setup and operation. Indeed, we've been giving suggestions in each lesson plan to help you stress setup and operation related topics.

We've done so for three reasons. First, and as stated, programmers *must* know enough about setup and operation to direct setup people and operators. Truly, the more a programmer knows about setup and operation, the better and more efficient the programs they will write.

Consider, for example, the technique shown in lesson eighteen that is related to trial machining using block delete. With a full understanding of what a setup person or operator must do in order to trial machine, a programmer can include commands right in the program that facilitate any trial machining application. Without this understanding, the setup person and operator must struggle through trial machining on their own.

Second, setup people and operators can truly benefit from having a working knowledge of certain programming features. When appropriate, we've provided suggestions in each lesson plan to help you explain certain programming functions to setup people and operators.

While you didn't go into programming details for setup people and operators, you explained enough to help them understand the setup- and operation-related implications of these programming features. During Key Concepts one and four, for example, you explained enough about program zero assignment, tool length compensation, and cutter radius compensation to help setup people and operators understand the reasons why certain things must be done at the machine.

Third, we've minimized the need for duplicating presentations. If you've followed our recommendations and presented the setup- and operation-related implications of certain programming features during programming, you won't have to repeat these presentations during the setup and operation part of this class – though reviewing key points never hurts.

### Are you teaching *only* setup and/or operation?

The second and third reasons given above apply only to setup people and operators. We have pointed out several times that this curriculum stresses all three tasks a student must master in order to be fully proficient with a CNC machining center: programming, setup, and operation.

However, companies commonly find that the most difficult position to keep filled is that of CNC operator. For this reason, you may find yourself teaching a class made up of only CNC operators – or setup people and operators. Again, please follow the recommendations for doing so that are included in each lesson plan. That is, ***don't try to begin a setup and operation class starting with Key Concept number seven***.

## Key points to make when introducing this key concept

▪ Assuming you're teaching all three tasks (programming setup, and operation, the slide show presentation helps you quickly review key topics you setup- and operation-related topics you presented during the programming-related lessons.
▪ Pages 283-288 in the student manual help you explain the machine's directions of motion from an operator's viewpoint. We also describe some of the most rudimentary procedures needed to run a machining center. If you have followed our suggestions for lab exercises at the machine, students should be familiar with these procedures.
▪ In key concept number one, students learned key things about the machine from a programmer's viewpoint. Now you'll be switching the viewpoint to that of a setup person or operator. Setup people and operators must be much more intimate with the machine – they must have a clear understanding of the tasks they must perform and know the function of all buttons and switches on the machine.

## If you're teaching only setup and/or operation

Again, don't try to begin teaching setup and operation starting at Key Concept number seven. Each lesson plan to this point has presented suggestions for teaching setup people and operators. Taken from the student manual, here's a recap of the programming-related lessons that include information on setup and/or operation:

***From Lesson One: Machine configurations***
▪ Present from the beginning to the topic "Programmable functions of machining centers"
***From Lesson Two: General flow of the programming process***
▪ Present the entire lesson
***From Lesson Four: Program zero and the rectangular coordinate system***
▪ Present from the beginning to the topic "Wisely choosing the program zero point location"
***From Lesson Five: Determining program zero point assignment values***
▪ Present the entire lesson
***From Lesson Six: Assigning program zero***
▪ Present from the beginning to the topic "What if my machining center doesn't have fixture offsets?
***From Lesson Ten: Introduction to compensation***
▪ Present the entire lesson
***From Lesson Eleven: Tool length compensation***
▪ Present from the beginning to the topic "Programming tool length compensation"
▪ Present from the topic "The setup person's responsibilities with tool length compensation" to the topic "Typical mistakes with tool length compensation"
▪ Present from the topic "Trial machining with tool length compensation" through the end of the lesson
***From Lesson Twelve: Cutter radius compensation***
▪ Present from the beginning to the topic "Steps to programming cutter radius compensation"
▪ Present from the topic "The setup persons responsibilities with cutter radius compensation" to the topic "Rough and finish milling with the same set of tool path coordinates"
▪ Present from the topic "Trial machining with cutter radius compensation" through the end of the lesson
***From Lesson Thirteen: Fixture offsets***
▪ Present from the beginning to the topic "Programming multiple program zero points"
▪ Present from the topic "Shifting the point of reference for fixture offset entries" to the topic "Programming fixture offset entries"
▪ Read from the heading "Some other applications for the common fixture offset" through the end of the lesson
***From Lesson Fourteen: Introduction to program structure***
▪ Present from the beginning to the topic "Machine variations that affect program structure"

## If you are teaching only operation

Again, most companies have more CNC operators than setup people or programmers. For this reason, there is commonly more turnover (replacement) of CNC operators than other CNC positions. Many companies have difficulty keeping this position fully staffed with qualified people.

Additionally, entry level CNC operators tend to have the least manufacturing experience. You may want to confirm that they possess prerequisite skills (like blueprint reading, shop math, an understanding of gauging tools, etc.) before continuing. For example, I've found that many people attending an entry level CNC operator's class don't know how to interpret tolerances – that is, they cannot come up with the correct adjustment amount (offset setting) in order to bring a measured dimension back to its target dimension.

## If you are teaching only programming

Again, programmers should be well enough versed with setup and operation to direct setup people and operators. Throughout the setup- and operation-related lessons we'll be recommending points that you should make to programmers.

## At the machine

We have no suggestions related to the material in this segment.

## Lab exercise

We have no recommendations for lab exercises for this segment of the course.

## Homework

▪ Read the introduction to Key Concept number seven that begins on page 281 in the student manual.

## Exercise

None for this segment of the course.

Explain the difference between setup tasks and production-running tasks. | 20 minutes lecture time



Solicit questions about previous topics. If you haven't already, review the setup and operation discussions presented during programming lessons.

Students must understand what they will be expected to do. Note that we present all tasks related to setting up and running CNC machining centers. If you know certain tasks will not be required of your students, be sure to point them out.

**Main topics for this lesson:**

**Lesson 20** Presentation links



**Pages 289-312** in the student manual

**Lesson objective:** Help students understand the tasks related to setting up and running a CNC machining center.

Determining the distinction between setup-related tasks and production running-related tasks is pretty simple. When the machine is down between production runs, it is in setup. It is, of course, the setup person the makes setups.

However, we look at *operation* two ways.

- First, there are certain things an operator must master to be confident with the machine. They must, for example, know all the buttons and switches, they must master certain operation procedures, and in general, they must be comfortable running the machine.
- Second, once a setup is completed, the operator must run workpieces. Any task related to completing a production run is the responsibility of the CNC operator.

**Key points to make for each topic:**

**Introduction to setup and operation**
- This topic is described in previous pages when introducing Key Concept number seven.

**The four Key Concepts of setup and operation**
- The slide show allows you to introduce all four setup and operation Key Concepts.

**Operator responsibilities**
- Explain that CNC-using companies vary with regard to what they expect of their CNC people. In this lesson, we're going to explain all of the tasks needed to setup and run a CNC machining center.
- Point out that most (especially product-producing) companies break up these tasks. Several people are involved. But in some companies (especially workpiece producing and tooling producing companies), one person may be expected to perform all of the tasks we show in this lesson.

**Tasks related to setup**
- These tasks, of course, get the machine ready to run production.
- The slide show helps you present these tasks in the approximate order that setups are actually made.
- We include verifying the program and running the first workpiece as part of setup. Until a part passes inspection, of course, the operator cannot start the production run.

**Tasks related to completing a production run**
- The slide show and student manual help you explain each task.
- First we show tasks that must be completed in every cycle (like workpiece load/unload, activating the cycle, and workpiece measurement).
- We also show tasks that don't take place in every cycle (sizing adjustments, preventive maintenance, replacing dull tools, etc.).

While it's not shown in the slide show presentation, the student manual provides an excellent "sample scenario" for how a job gets setup and run (starting on page 291). Using the same tasks described in the lesson, it walks students through the running of a sample job from start to finish.

**If you're teaching only programming**
- Again, programmers must be well acquainted with the tasks performed by setup people and operators. Without this understanding a programmer cannot provide adequate documentation.
- You must present this lesson in its entirely to entry-level programmers.

**If you're teaching only setup and/or operation**
Some of the tasks shown in this lesson have been introduced during the lessons related to programming. Though this is the case, present this lesson in its entirety.

### If you're teaching only operation

While you can skip the tasks related to setup, many of these tasks must be repeated during a production run. Also, it doesn't hurt a CNC operator to know what is involved with getting a CNC machining center ready to run production.

### At the machine (15 minutes)

Use one of the programs from the programming exercises to review the tasks related to setup and running production. While you don't have to perform every step, at least show students the results of every task (completed program zero assignment in fixture offsets, filled-in offset table, workholding device on the table, cutting tools in the automatic tool changer magazine, program in memory, etc.).

While it is unlikely that you have hundreds of workpieces to run, be sure students understand that in the real world, companies commonly run hundreds – if not thousands – of workpieces during a production run. During this time, tools show signs of wear and may require sizing adjustments. Eventually they dull completely and must be replaced. And if the production run lasts for days or weeks, it is likely that the machine will be turned off at some point. Many companies warm up their machines prior to starting a shift. It may be difficult, if not impossible, to illustrate all of this in class, but at least prepare students for what they'll face when they work for a CNC-using company.

### Lab exercise

Have students practice with pre-developed procedures for basic tasks like power-up, jogging the axes, using the handwheel, starting the spindle, and so on. Note that if students have been through the entire programming session, they've probably already done this exercise.

### Homework

- Read all of lesson twenty that begins on page 289 in the student manual.
- Do the exercise on page 312 in the student manual. (Note that this exercise helps test the student's understanding of some of the setup-related topics presented during the programming lessons – you may want to do this exercise as a group project.

### Exercise (about 10 minutes if done in class)

Do exercise number twenty in the workbook.

## Notes:

Explain all of the buttons and switches on your machine/s. | 20 minutes lecture time

Lessons in Key Concept #7 (you are here):



Solicit questions about previous topics. Review the tasks related to setup and running production.

You've discussed many important buttons and switches during programming-related lessons. And if you've been following our suggestions for things to do at the machine and having students do the lab exercises, it's likely that students have understand many of them. Now you'll be explaining all buttons and switches on the machine.

**Main topics for this lesson:**

**Lesson 21** Presentation links

- The two operation panels
- Buttons and switches on the control panel
- Buttons and switches on the machine panel

**Pages 313-322** in the student manual

**Lesson objective:** Help students understand all of the buttons and switches on your machining centers – and master the most often-used ones.

The presentation time for this lesson will vary based upon how much explaining you did during the programming-related lessons. Setup people and operators must understand the function of *all* buttons and switches on the machine. If a particular button or switch is never used, they should understand *why* it is never used.

- A setup person or operator should never give up until they know the function of all buttons and switches.
- When they start running actual machines in the shop, there will likely be unfamiliar buttons and switches. They'll be on their own to learn about them (from the control manufacturer's manual and the machine tool builder's manual.

**Key points to make for each topic:**

**The two operation panels**

- We break the operation panels on a CNC machining center into two categories – the control panel (made by the control manufacture – Fanuc in our case), and the machine panel/s (made by the machine tool builder.
- There could be several machine panels – the main one close to the display screen, as well as others located as needed (like near the automatic tool changer or pallet changer).

**Buttons and switches on the control panel**

- Again, these are buttons and switches located on the operation panel made by the control manufacturer.
- The slide show helps you describe each button and switch on a typical control panel.
- You must, of course, explain the function of buttons and switches that are on you machine's control panel that are not explained in the slide show or student manual. Show students where this information can be found in the related manuals.

**Buttons and switches on the machine panel**

- Again, these are buttons and switches located on the operation panel made by the machine tool builder.
- The slide show helps you describe each button and switch on a typical machine panel.
- Be sure to point out that machine panels vary dramatically from one machine tool builder to another. Builders can't seem to agree on what a CNC setup person or operator needs to run the machine.
- You must, of course, explain the function of buttons and switches that are on you machine's machine panel that are not explained in the slide show or student manual. Show students where this information can be found in the related manual/s.
- Note that you can minimize your presentation for the mode switch. We feel that it is such an important switch, we devote an entire lesson to discussing it (lesson 22).

**If you're teaching only programming**

- Again, programmers are expected to direct setup people and operators. How can a programmer answer questions about buttons and switches on the machine if they don't know the answers themselves?
- We recommend that you present this lesson in its entirety to programmers.

**If you're teaching only setup and/or operation**

Some (indeed, many) of the buttons and switches shown in this lesson have been introduced during the lessons related to programming. Though this is the case, present this lesson in its entirety. Have students consider any repeated information as review.

We cannot overstress the importance of setup people and operators understanding the buttons and switches on their machines. Running a machining center is difficult enough with a good understanding of its functions. Without this understanding, the setup person or operator will be in grave danger.

### If you're teaching only operation

While operators don't need to know as much as setup people, it is still important that they understand the machine functions they will be activating – as well as those they won't. I've been in companies that simply tell operators "not to worry" about a function they won't be using. This causes frustration with (motivated) operators.

Admittedly, entry-level operators may have trouble absorbing all it takes to run their machines. But when they have questions, they should be answered, especially as they gain experience.

### At the machine (30 minutes)

Indeed, this entire lesson can be presented at the machine. Go over each operation panel, thoroughly describing each function. When you're finished, ask students if there are any buttons and switches they don't recognize.

You'll also be describing every display screen page in this lesson. Most of these display screens have been discussed during the programming lessons.

It's unlikely that students will remember (memorize) every button and switch when you do this the first time. Be sure to review the buttons and switches as you begin upcoming lessons.

### Lab exercise

Have students continue practicing with pre-developed procedures for basic tasks like power-up, jogging the axes, using the handwheel, starting the spindle, and so on. Note that if students have been through the entire programming session, they've probably already done this exercise.

### Homework

▪ Read all of lesson twenty-one that begins on page 313 in the student manual.

### Exercise (about 10 minutes if done in class)

Do exercise number twenty-one in the workbook.

## Notes:

Introduce Key Concept number eight. | 5 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

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. There are ten Key Concepts further divided into twenty-four lessons.

The most important switch on any machining center is the mode switch. This switch must be placed in the appropriate position before the desired function can be activated. While there are more than three positions on the mode switch, this switch can be divided into three basic categories: manual, manual data input (MDI), and program operation. It is the focus of this one-lesson Key Concept to describe these modes.

Key concept number eight

You must understand the three modes of operation

Begins on **page 323** in the student manual

**Key Concept objective:** Ensure an understanding of the three basic modes of machine operation.

Point out that the mode switch is at the heart of any CNC machining center. No function can be activated unless this switch is in the appropriate position.

- The mode switch is always the first switch a setup person will set when performing any function on the machine.
- If the mode switch is improperly set, the machine will not respond to the desired action. (This is nice to know. The worst that can happen when the mode switch is not correctly set is that the machine won't respond – it simply won't do *anything*.)

**If you are teaching only programming**
- We recommend that you present this Key concept to programmers.

**At the machine**

No suggestions yet.

**Lab exercise**

No suggestions at this time.

**Homework**

- None.

**Exercise**

None for this segment of the class.

**Notes:**

Lesson Plan | **The Three Modes Of Operation** | **Lesson 22**

Explain every position on the mode switch. | 10 minutes lecture time

Lessons in Key Concept #8 (you are here):

- 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

Solicit questions about previous topics. Review setup and operation tasks – and review the buttons and switches on the machine.

The most important switch on the machine is the mode switch. You'll be explaining it in detail during this lesson.

**Main topics for this lesson:**

**Lesson 22** Presentation links

The importance of the mode switch
The three modes of operation
- Manual mode
- Manual data input (MDI) mode
- Program execution mode

**Pages 323-328** in the student manual

**Lesson objective:** Ensure an understanding of the three basic modes of machine operation.

We begin by explaining the importance of the mode switch. Then we present the three most basic modes.

**Key points to make for each topic:**

**The importance of the mode switch**

▪ The slide show begins by showing the two most popular types of mode switches – a rotary switch and a series of lighted buttons.
▪ Point out that if the mode switch is in the wrong position, the machine won't respond to your action.
▪ The mode switch is the first switch to be set when performing any function on the machine.

**The three modes of operation**

▪ **Manual mode:** Explain that manual mode, which includes (at least) jog, handwheel, and zero return, is used to get a quick response from the machine. In any of these modes, a button is pressed and the machine responds (starting the spindle, moving an axis, turning on the coolant, etc.). The slide show helps you show the details, presenting several examples of when manual mode is used.

**Manual data input (MDI) mode:** This mode includes the mode switch positions MDI and edit. Explain that the MDI mode switch position is used primarily to manually activate functions for which there are no manual controls. Most machining centers, for example, provide no manual means to activate the automatic tool changer. If an operator wants to cause a manual tool change they must use the MDI mode switch position to do so. The slide show helps you show some examples of using this function. The edit mode is used to modify CNC programs. If you've been running any practice programs on the machine, it's likely that you've demonstrated this function by now. Again, the slide show provides a good example.

▪ **Program activation mode:** This mode is used to run programs. With newer machines, there is only one mode switch position, labeled either auto or memory. Point out that older machines may also include a mode switch position labeled tape.

**At the machine (10 minutes)**

Using previously developed procedures to do so, demonstrate the various modes. If you've been following suggestions made in previous lessons, students have already practiced with some of these procedures.

Be sure to demonstrate what will happen (nothing) when the mode switch is in the wrong position.

**Lab exercise**

No suggestions at this time.

**Homework**

▪ Read all of lesson 22 that begins on page 323 in the student manual.

**Exercise (about 10 minutes if done in class)**

Do exercise number 22 in the workbook.

**Notes:**

Introduce Key Concept number nine. | 5 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

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. There are ten Key Concepts further divided into twenty-four lessons.

***We cannot overstress the importance of procedures.*** We urge you to develop a set of step-by-step procedures for each machine you have in your lab.

In the student manual, we show several (rather generic) sets of procedures for current controls in the Fanuc product line. You can use any one of them as a template for which to develop your own procedures. That is, develop procedures for each task presented (power-up, doing a zero return, jogging the axes, using the handwheel, starting the spindle, etc.).

Note that these procedures are listed in several categories (manual procedures, MDI procedures, setup procedures, program modification procedures, program running procedures, and program verification procedures). You may find the need to develop more procedures based upon the accessories equipped with your machine.

Whenever you see a student struggling with a machine function, it should be taken as a signal that you need to develop a procedure to help.

Eventually, setup people and operators will memorize the most often-needed procedures. But until then, they'll need some help.

Key concept number nine

You must understand the key operation procedures

Begins on **page 329**
in the student manual

**Key Concept objective:** Help students understand that having a procedure to perform any task will simplify the task – more importantly – that they will probably need to develop their own set of procedures when they start working for a CNC-using company.

I'm always amazed when I go into CNC using companies to find that they have not documented the procedures needed to run their CNC machines. While entry level people do eventually muddle through and memorize these procedures, they must put up with a great deal of frustration.

You must set a good example. Provide students with procedures for *everything*. And make students learn how to document the procedures needed to run a machine.

**If you are teaching only programming**

- Frankly speaking, we feel it is the programmer's responsibility to document the procedures needed to run a CNC machining center. How can they do so if they don't know which procedures to document?

**At the machine**

No suggestions yet.

**Lab exercise**

No suggestions at this time.

**Homework**

- None.

**Exercise**

None for this segment of the class.

**Notes:**

Explain every position on the mode switch. | 10 minutes lecture time

Lessons in Key Concept #9 (you are here):

- 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**

Solicit questions about previous topics.

Frankly speaking, running a CNC machining center is little more than following a series of procedures. The trick lies in knowing which procedure to perform and when to perform it.

**Main topics for this lesson:**

**Lesson 23** Presentation links

- Procedure importance
- Manual procedures
  - Start machine
  - Zero return
  - Start spindle
  - Jog axes
  - Use handwheel
  - Load tools
  - Activate coolant
  - Reset axis displays
  - Enter offsets
- MDI procedures
  - Change tools
  - Start spindle
- MDI procedures (continued)
  - Activate coolant
  - Zero return
- Setup procedures
- Program manipulation procedures
  - Load programs
  - Punch programs
  - Directory of programs
  - Delete programs
  - Call up a program
  - Search within a program
  - Alter, insert, and delete

**Pages 329-340** in the student manual

**Lesson objective:** Provide students with the procedures they need to run a CNC machining centers.

In many companies, entry level people really struggle when leaning how to run a machine. How do you power up the machine? A person may have to be shown this procedure several times before they remember how to do it. And this is but one (rather simple) procedure. This "show me" method of learning is frustrating for everyone involved.

In this lesson, you'll be showing students the most important procedures (procedures they should document for themselves for any machine the must run) as well as demonstrating their use.

**Key points to make for each topic:**

**Procedure importance**

▪ With an understanding of what must be done (which you've been showing throughout the class, running a machining center is little more than following a series of procedures.
▪Point out that step-by-step procedures will help newcomers perform any machine function – as long as they know why the function must be performed.
▪ The slide show provides a "road map" analogy to help you stress the importance of procedures.
▪ We divide the procedures into categories, beginning with manual procedures. In the slide show, we demonstrate every procedure. In the student manual (beginning on page 332), we document the procedures for several Fanuc control models. It might be best to actually demonstrate procedures (at least those that you haven't done to this point in the class) right on your lab machine.

**Manual procedures**

▪ These are procedures that will render an immediate response.

**Manual data input (MDI) procedures**

▪ These are procedures that use the MDI mode – usually required for functions that must be manually activated, but for which have no manual controls.

**Setup procedures**

▪ We've provided a few procedures needed during setup (like tool length measuring), but you may elect to develop more.

**Program manipulation procedures**

▪ These are procedures used to load, call up, and edit CNC programs.

**What about program running procedures?**

▪ Procedures needed to verify and run CNC programs are shown in lesson twenty-four.

**At the machine (10 minutes)**

If you haven't already, be sure to demonstrate the use of the procedures shown in this lesson.

**Lab exercise**

Have students develop a few procedures on their own. That is, make them document the step-by-step procedure for power-up, jogging the axes, using the handwheel, and other important procedures. You can provide them with a blank form – or simply have them write down the procedures in a notebook.

**Homework**

▪ Read all of lesson 23 that begins on page 329 in the student manual.

**Exercise (about 10 minutes if done in class)**

Do exercise number 23 in the workbook.

**Notes:**

Introduce Key Concept number ten. | 5 minutes lecture time



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. There are ten Key Concepts further divided into twenty-four lessons.

The final Key Concept draws together much of what has been presented in this class. Students must know how to verify new programs as well as programs that have been run before. They must also, of course, be able to machine acceptable workpieces. Though companies vary in this regard, we're assuming that it is quite important to make the *first* workpiece being machined a good one.

They must, of course, be able to find and correct mistakes as they are found. And mistakes can be related to the program or to the setup that has been made. This means they must be able to recognize the cause of problems being encountered – and again – this requires a good understanding of what has been presented so far.

We provide a series of procedures for verifying CNC programs (dry run, air cutting normal run, and cautiously running the first workpiece). These procedures are not overly specific – and are somewhat complex. And again, they require students to understand many of the points you've made so far.

## Key concept number ten

## You must know how to safely verify and run CNC programs

Begins on **page 341**
in the student manual

**Key Concept objective:** Help students understand how to safely verify new and previously run programs – and make the very first workpiece being machined a good one.

**If you are teaching only programming**

▪ Programmers are usually expected to assist the setup person when (especially new) programs are verified. For this reason, a programmer should understand all points made in this Key Concept.

### At the machine

No suggestions yet.

### Lab exercise

No suggestions at this time.

### Homework

- None.

### Exercise

None for this segment of the class.

## Notes:

Explain every position on the mode switch. | 10 minutes lecture time

Lessons in Key Concept #10 (you are here):

- 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**

Solicit questions about previous topics. Review lesson 20 (Tasks related to setup and operation). Review trial machining.

Setup people must be able to safely verify new and previously run programs.

**Main topics for this lesson:**

**Lesson 24** Presentation links

Safety priorities
Typical mistakes
New vs proven programs
Program verification functions
The most dangerous time
An example approach
Program verification procedures
Free flowing dry run
Normal air cutting run
Cautiously running first workpiece
Example
Rerunning tools
Completing a production run

**Pages 343-352** in the student manual

**Lesson objective:** Provide students with the procedures needed to safely verify programs.

While it's not shown in the slide show, the student manual provides an excellent example of verifying a program from start to finish – including the running of the first workpiece (page 346 begins this example). A similar example scenario is shown in lesson twenty, but in that example, there are no mistakes. With the example job shown here in lesson twenty-four, there are many mistakes. This should give students a realistic view of what they'll be in for when they begin working on a CNC machining center.

**Key points to make for each topic:**

**Safety priorities**

▪ We begin by relating the three levels of priority a setup person should adhere to – operator safety, machine safety, and workpiece safety.

**Typical mistakes**

▪ The slide show and student manual help you review the most common mistakes that are made when programming and making setups.

▪ Point out that when a mistake is found during program verification, the setup person will only see the "symptom" of the problem. Determining the problem requires a kind of backtracking from the symptom to potential causes. Knowing the typical mistakes that can be made helps limit the potential causes.

**New versus proven programs**

▪ The slide show and student manual helps you explain the differences.

**Review of program verification functions**

▪ The slide show helps you introduce and/or review machine functions like the program check page, feed hold, single block dry run, feedrate override and rapid override.

▪ When activating programs (by pressing cycle start), be sure students always keep a finger ready to press the feed hold button.

**Program verification procedures**

▪ Based upon whether the program being verified is a new or proven program, we show procedures to verify it.

▪ The student manual includes an excellent example scenario for helping you illustrate the program verification procedure – including trial machining for critical machining operations.

**At the machine (30 minutes)**

Use an example program that contains mistakes (possibly the one shown in the student manual on page 347) to demonstrate program verification – as well as how to correct mistakes.

If you have been running the programs students have written during class, it's likely that you've already done some of this.

Be sure to emphasize setup mistakes (like improper tool length compensation entries and program zero assignment mistakes.

**Lab exercise**

While you'll want to be very careful if the program contains mistakes, have students verify a program on their own.

**Homework**

▪ Read all of lesson 24 that begins on page 343 in the student manual.

**Exercise (about 10 minutes if done in class)**

Do exercise number 24 in the workbook.

### Final review

One last length review might be in order to recap all that has been done in class – and to prepare students for the final test.

### Final test

In your *For The Instructor* manual, we have provided a final test (with answers provided) that you can use to test the student's overall understanding of material presented in this class.

## Notes:

# Scheduling Time For Your Class

## Time for lectures

As you can imagine, the time needed to present course material will vary based upon several factors, including your selected topics, student aptitude, the number of questions asked, and whether or not you go off on any tangents.

Each Key Concept and lesson in this manual does include a suggested lecture time. But do keep in mind that we've tried to come up with an *average* time, based upon our experience and having students with pretty good aptitude and minimal questions. Here is a summary:

Introduce Key Concept one: 15 minutes
Present lesson one: 30 minutes
Present lesson two: 15 minutes
Present lesson three: 15 minutes
Present lesson four: 30 minutes
Present lesson five: 20 minutes
Present lesson six: 15 minutes
Present lesson seven: 15 minutes
Introduce Key Concept two: 10 minutes
Present lesson eight: 20 minutes
Introduce Key Concept three: 5 minutes
Present lesson nine: 35 minutes
Introduce Key Concept four: 15 minutes
Present lesson ten: 20 minutes
Present lesson eleven: 25 minutes
Present lesson twelve: 30 minutes
Present lesson thirteen: 30 minutes
Introduce Key Concept five: 5 minutes
Present lesson fourteen: 10 minutes
Present lesson fifteen: 20 minutes
Introduce Key Concept six: 10 minutes
Present lesson sixteen: 20 minutes
Present lesson seventeen: 15 minutes
Present lesson eighteen: 25 minutes
Present lesson nineteen: 15 minutes
Introduce Key Concept seven: 10 minutes
Present lesson twenty: 20 minutes
Present lesson twenty-one: 20 minutes
Introduce Key Concept eight: 5 minutes
Present lesson twenty-two: 10 minutes
Introduce Key Concept nine: 5 minutes
Present lesson twenty-three: 10 minutes
Introduce Key Concept ten: 5 minutes
Present lesson twenty-four: 10 minutes

**Approximate total lecture time: 10 hours**

Again, these times assume ideal conditions. If you have students with minimal previous machining experience and lower aptitude – and especially when questions start getting asked – it can be necessary to double the suggested times.

Also remember that these times are related to the presentation of new material and do not include time for review. If you follow our recommendation of spending *at least* ten percent of your lecture time reviewing previously presented material, you'll be adding more time to your lectures.

## Time you spend at the machine

Many of the lessons include suggestions for things you can demonstrate at a machine that reinforce your lectures. We provide recommended times, but again, they assume ideal conditions.

Key Concept one: 10 minutes
Lesson one: 15 minutes
Lesson two: 15 minutes
Lesson three: 20 minutes
Lesson four: 20 minutes per student
Lesson five: 20 minutes
Lesson six: 20 minutes
Lesson seven: 10 minutes per student
Key Concept two: 10 minutes
Lesson eight: none
Key Concept three: 10 minutes
Lesson nine: 30 minutes
Key Concept four: none
Lesson ten: 10 minutes
Lesson eleven: 15 minutes
Lesson twelve: 15 minutes
Lesson thirteen: 10 minutes
Key Concept five: none
Lesson fourteen: 20 minutes
Lesson fifteen: 20 minutes
Key Concept six: none
Lesson sixteen: 15 minutes
Lesson seventeen: 10 minutes
Lesson eighteen: 10 minutes
Lesson nineteen: 10 minutes
Key Concept seven: none
Lesson twenty: 15 minutes
Lesson twenty-one: 30 minutes
Key Concept eight: none
Lesson twenty-two: 10 minutes
Key Concept nine: none
Lesson twenty-three: 10 minutes
Key Concept ten: none
Lesson twenty-four: 30 minutes

**Approximate total time at the machine: 10 hours**

## Time with lab exercises

We provide some ideas for what you can have students do on the machine by themselves (with your supervision, of course). Suggested times are also provided. However, we encourage you to develop your own labs based upon your knowledge of what your students will need when they finish your class – and based upon the needs of local employers.

## Time for homework and assignments

These activities are intended to be done by students outside the classroom (though we do provide approximate times for completing assignments). Almost all of the homework involves reading assignments from the student manual. Some additionally involve completing exercises in the student manual.

Assignments are provided in the workbook. You can use them as tests to gauge students' understanding of presented material. We also provide a final test (in the *For The Instructor* manual) to help you gauge overall comprehension.

44 Little Cahill Road • Cary, IL 60013
Phone 847.639.8847 • Fax 847.639.8857
Internet: www.cncci.com