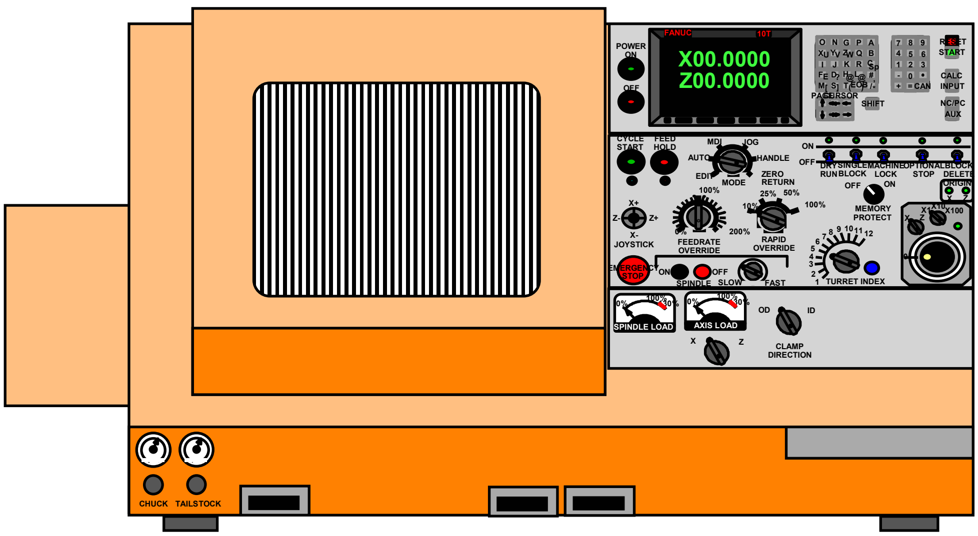
Lesson plans

for

Turning Center Programming, Setup, and Operation

Mastering CNC Turning 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)

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.

Machining Center Programming, Setup, and Operation

Lesson 1 (continued)

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 right 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 turning 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 turning 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.

Lesson Plan	Know Your Machine From a Programmer's Viewpoint	Key Concept 1
Introduce course content, Key Concepts approach, and Key Concept number one.		15 minutes lecture time

Programming:

1: Know your machine from a programmer's viewpoint
- 1: Machine configurations
- 2: Understanding turning center speeds and feeds
- 3: General flow of the CNC process
- 4: Visualizing the execution of a CNC program
- 5: Program zero and the rectangular coordinate system
- 6: Determining program zero assignment values
- 7: Four ways to assign program zero
- 8: Introduction to programming words

2: You must prepare to write programs
- 9: Preparation steps for programming

3: Understand the motion types
- 10: Programming the three most basic motion types

4: Know the compensation types
- 11: Introduction to compensation
- 12: Geometry offsets
- 13: Wear offsets
- 14: Tool nose radius compensation

5: You must provide structure to your CNC programs
- 15: Introduction to program structure
- 16: Four types of program format

6: Special features that help with programming
- 17: One-pass canned cycles
- 18: G71/G70 – rough turning and boring multiple repetitive cycles followed by finishing
- 19: G72-G75 – other multiple repetitive cycles
- 20: G76 – Threading multiple repetitive cycle
- 21: Working with subprograms
- 22: Special considerations for Fanuc 0T and 3T controls
- 23: Other special features of programming

---: Appendix – special machine types and accessories

Setup and operation

7: Know your machine from a setup person or operator's viewpoint
- 24: Tasks related to setup and running production
- 25: Buttons and switches on the operation panels

8: Know the three basic modes of operation
- 26: The three modes of operation

9: Understand the importance of procedures
- 27: The key operation procedures

10: You must know how to safely verify programs
- 28: 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-eight 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.

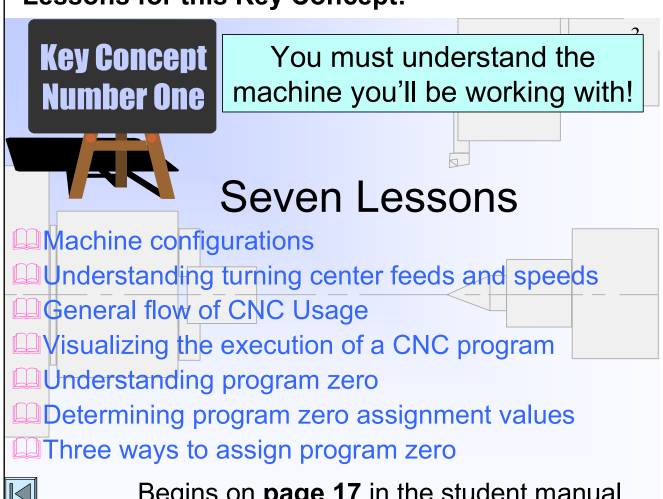
Explain why you're using the Key Concepts approach
- It limits the number of main principles (to 10) a student must understand to become proficient with CNC turning centers.
- It lets students understand precisely where they stand as they go through the class.
- It provides a very good way to organize 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 turning 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).

Need help getting ready to teach a lesson? - The CD-rom disk include two sets of presentations – one with **audio guidance** and the other without. Key slides have a *teacher icon*. When clicked, the audio guidance will play.

Lessons for this Key Concept:



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 turning center in action before, take them out to a machine in your lab and show them one! Have a program ready to run – cutting chips if possible. Let them see what they're going to be working with in the class. We've found that students show the most interest when they see a machine in action.

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

Lab exercise

None for this class segment.

Homework

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

Exercise

None for this class segment.

Notes:

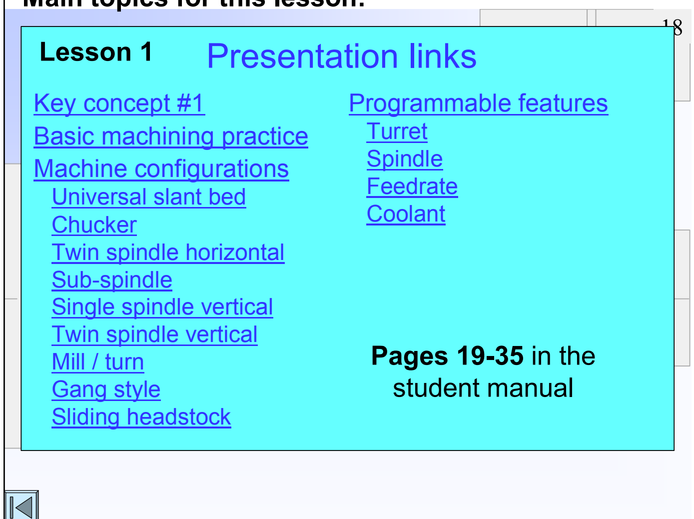
Explain machine components, directions of motion, and programmable functions. | 30 minutes lecture time

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

1: Machine configurations
2: Understanding turning center speeds and feeds
3: General flow of the CNC process
4: Visualizing the execution of a CNC program
5: Program zero and the rectangular coordinate system
6: Determining program zero assignment values
7: Four ways to assign program zero
8: 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 objective: Introduce students to the kind/s of CNC turning 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 (rough and finish turning, rough and finish boring, rough and finish facing, treading, necking, and drilling). 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.
- The slide show helps you show the difference among the various types of turning centers (universal style slant bed, chucker type, vertical type, twin spindle, gang type, etc.).

Directions of motion

- Show the directions of motion (axes) for each kind of turning center you will be teaching.
- Point out that with many machines, the cutting tool (and turret) move along with each axis. This makes it easy to determine and remember the polarity for each axis..
- Explain the polarity (plus versus minus) of each axis – X minus is toward spindle center in X, Z minus is toward the headstock in Z. Directions of motion and polarity are described during each machine type. This makes it easy to display only the machine/s you'll be teaching.

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 turret indexing. If your machines have more programmable functions (like a tailstock), 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 turning centers allow the spindle to be programmed in three ways, speed (with S), activation (with M03, M04, and M05), and range (with two M codes).
- speed can be specified in revolutions per minute (rpm) or in surface feet per minute (sfm) [which is meters per minute in metric mode].
- M03 (forward) is used for right-hand tools and M04 (reverse) is used for left-hand tools.
- range selection is done with M codes. While M codes vary, M41 is often used for low range and M42 is often used for high range. More about speed and feed control is presented in lesson two.

Feedrate – be sure students understand that…

- an F word is used to specify feedrate and feedrate. Feedrate is specified in either per-minute or per-revolution fashion – selected by two G codes (G98 and G99).

Coolant – be sure students understand that…

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

Turret indexing – be sure students understand that…

- most turning centers have a turret – a four-digit T word is used to specify a turret index.

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 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 only the styles of turning center/s you will be teaching.

Directions of motion

▪ Show the directions of motion (axes) for each kind of turning center you will be teaching.
▪ Point out that with many machines, the cutting tool (and turret) move along with each axis. This makes it easy to determine and remember the polarity for each axis..
▪ Explain the polarity (plus versus minus) of each axis – X minus is toward spindle center in X, Z minus is toward the headstock in Z. Directions of motion and polarity are described during each machine type. This makes it easy to display only the machine/s you'll be teaching.
▪ 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. 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 index the turret.

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 the turret index. Let them take turns practicing with these procedures.

Homework

▪ Read all of lesson one in the student manual.
▪ Take the quiz on page 35 in the student manual.

Exercise (about 15 minutes if done in class)

Have students do exercise number one in the workbook.

Notes:

Explain the two ways (each) that speed and feed can be selected. | 15 minutes lecture time

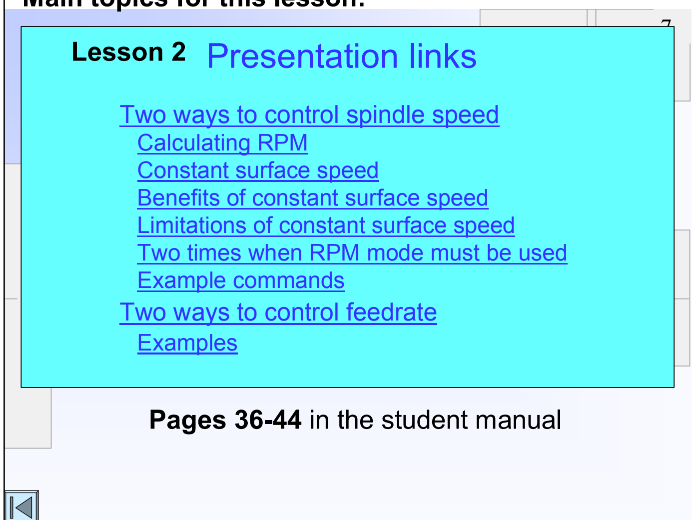
Lessons in Key Concept #1 (you are here):

1: Machine configurations
2: Understanding turning center speeds and feeds
3: General flow of the CNC process
4: Visualizing the execution of a CNC program
5: Program zero and the rectangular coordinate system
6: Determining program zero assignment values
7: Four ways to assign program zero
8: Introduction to programming words

Solicit questions about previously presented material. Review lesson 1.

Remind students that you're still presenting Key Concept number one. Part of understanding the machine is thoroughly understanding how spindle speed and feedrate are selected.

Main topics for this lesson:



Lesson objective: Help students understand speed and feed selection – as well as which speed and feed mode is most appropriate for a given machining operation.

This is the presentation links slide for lesson two. Use these links to get around in the presentation.

- Introduce tasks included in this lesson.
- Point out that a manual machinist has only one way to specify speed (rpm) and one way to specify feedrate (per revolution). With CNC turning centers there are *two ways* to specify each.

Key points to make for each topic:

Two ways to control spindle speed

- The slide show begins by helping you show how a manual engine lathe operator must select speed – in rpm. We show the formula for rpm calculation. This calculation is based upon the workpiece diameter being machined. Point out that for many turning operations, like facing and grooving, the diameter being machined constantly changes during the operation – making it difficult (and somewhat infeasible) to specify speed in rpm.
- We then help you introduce constant surface speed (css), a feature that lets the programmer specify speed directly in surface feet per minute (sfm) if working in the inch mode or meters per minute (mpm) if working in the metric mode.
- Point out that constant surface speed is used for machining operations that require different diameters to be machined (rough and finish turning, rough and finish boring, rough and finish facing, necking, etc.).
- Explain that there are at least two times when the rpm mode must be used – center cutting operations (like drilling) and threading. Also if the turning center is equipped with live tooling, it can only be programmed in rpm mode.
- While it is not of immediate importance, we do show the cycle time limitation of using the constant surface speed mode.
- Explain the two G codes used to specify the speed modes (G96 for css mode and G97 for rpm mode).
- The slide show helps you show these words in context with a few examples.

The two ways to specify feedrate

- Explain that as with speed, there are two ways to specify feedrate – in per-minute fashion (G98) and in per-revolution fashion (G99). The slide show helps you illustrate per-revolution mode.
- In each feedrate mode, either inches (ipr) or millimeters (mmpr) can be specified. (G20 specifies inch mode, G21 specifies metric mode.)
- Explain that for almost all machining operations, the per-revolution feedrate specification is desirable.
- About the only time that feed-per-minute mode should be used is when you want to make a feedrate motion with the spindle stopped. The slide show helps you illustrate one time this is required – when bar feeding.
- If the machine is equipped with live tooling, feedrate must be specified in feed-per-minute mode for any live tooling operation.

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

In many companies, setup people are expected to modify speeds and feeds in a program during the program's verification. If this is the case, of course, setup people must understand *how* cutting conditions are specified – meaning the material in this lesson will be important to them.

At the machine (about 15 minutes)

Run a program that demonstrates the use of constant surface speed mode (a facing operation is ideal). Let students see and hear the spindle accelerate as the cutting tool approaches the center of the spindle (X0). Point out that at center, the spindle will be running at its maximum speed (in rpm) in the selected spindle range.

While it may be a little advanced for this early point in the class, try to show the cycle time limitation of constant surface speed. With a multi-tool program, let students see what happens as the turret retracts in X to a large-diameter turret index position (the spindle slow down takes longer than the retract motion). The same will happen when the next tool approaches to a small diameter – there will be a slight delay while the spindle accelerates up to speed. (Much later in the class, you'll be showing how to counteract this limitation.)

Lab exercise (about 5 minutes per student)

Have students continue practicing with machine-operation procedures.

Homework

- Read all of lesson two in the student manual.
- Take the quiz on page 44 in the student manual.

Exercise (about 15 minutes if done in class)

Have students do exercise number two in the workbook.

Notes:

Explain CNC-using company types and tasks related to using a turning center | 15 minutes lecture time

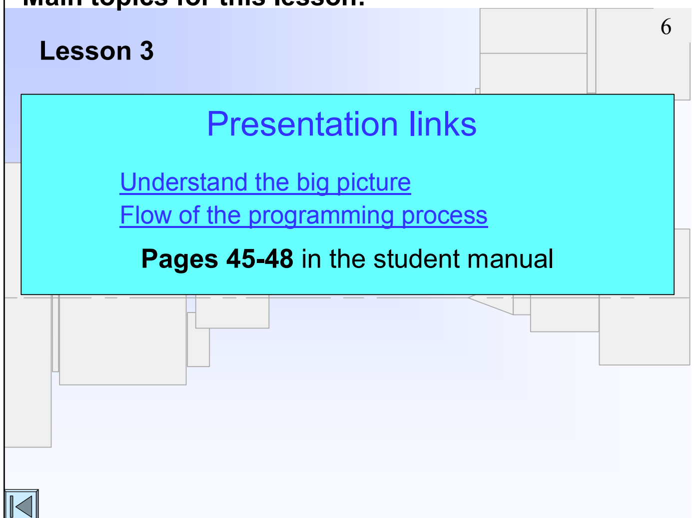
Lessons in Key Concept #1 (you are here):

1: Machine configurations
2: Understanding turning center speeds and feeds
3: General flow of the CNC process
4: Visualizing the execution of a CNC program
5: Program zero and the rectangular coordinate system
6: Determining program zero assignment values
7: Four ways to assign program zero
8: Introduction to programming words

Solicit questions about previously presented material. Review lessons 1 and 2.

Remind students that you're still presenting Key Concept number one. Students must understand where CNC turning centers fit into the bigger picture of the manufacturing environment.

Main topics for this lesson:



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

This is the presentation links slide for lesson three. 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 turning 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 three in the student manual.

Exercise (about 10 minutes if done in class)

Have students do exercise number three in the workbook.

Notes:

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

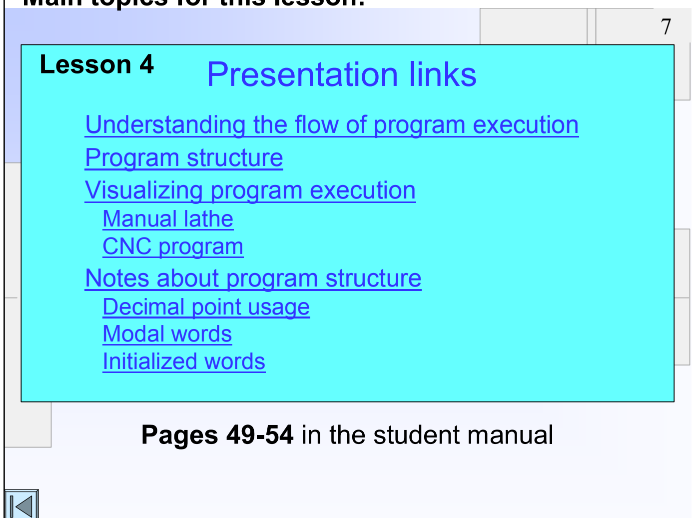
Lessons in Key Concept #1 (you are here):

1: Machine configurations
2: Understanding turning center speeds and feeds
3: General flow of the CNC process
4: Visualizing the execution of a CNC program
5: Program zero and the rectangular coordinate system
6: Determining program zero assignment values
7: Four ways to assign program zero
8: 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 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 four. 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 also be introducing some points about program structure.

Key points to make for each topic:

Understanding the flow of program execution – students must know that:

- writing a CNC program is like giving any other set of step-by-step instructions.
- programs are made up of commands – commands are made up of words – words are made up of a letter address (N, G, X, Z, etc.) and a numerical value.
- programs are executed sequentially. The machine will read, interpret the first command in the program. Then it will go on to the next command. Read, interpret, execute. It will continue in this fashion for the balance of the program.

Visualizing program executions – students must understand 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.
- 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.

Notes about 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 four in the student manual.
- Take the quiz on page 54 of the student manual.

Exercise (about 15 minutes if done in class)

Have students do exercise number four in the workbook.

Notes:

Explain how programmed positions are determined. | 20 minutes lecture time

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

1: Machine configurations
2: Understanding turning center speeds and feeds
3: General flow of the CNC process
4: Visualizing the execution of a CNC program
5: Program zero and the rectangular coordinate system
6: Determining program zero assignment values
7: Four ways to assign program zero
8: Introduction to programming words

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

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

Main topics for this lesson:

7

Lesson 5 Presentation links

Linear axis drive system
Graph analogy
Productivity for last year
XZ plane of turning center
How coordinates are calculated
Where to place the program zero point
Absolute versus incremental modes

Pages 55-64 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 four. In this program, a turning tool is commanded to move through certain positions so that it can turn a workpiece. 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 two dimensional coordinate system from a central origin.

Key points to make for each topic:

Linear axis drive 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 turning 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.

Graph analogy

- 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 turning 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 (XZ).

How coordinates are calculated

- 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 (-).
- Be sure students understand that X coordinates specify a diameter. Some dimensions (like chamfers are specified in radial fashion. The slide show helps you illustrate.

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.
- Program zero in the X axis is always the spindle/workpiece center. Program zero in Z is usually placed at the right end of the finished workpiece. (All examples in the slide show and student manual use this position as program zero.)

Absolute versus Incremental positioning

- When coordinates are specified from program zero, it is called the absolute mode of programming.
- With most turning centers, X and Z specify absolute positions.
- Students should concentrate on absolute positioning.
- Another positioning mode is available: the incremental positioning mode. With this positioning method, positions are commanded from the cutting tool's current position.
- With most turning centers, U and W are used to specify incremental departures. U specifies a diameter increase or decrease, W specifies an incremental departure distance in the Z axis.
- 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.

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 five in the student manual.
- Have students fill in the coordinate sheet on page 61.

Exercise (about 20 minutes if done in class)

Have students do exercise number five 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: Understanding turning center speeds and feeds
3: General flow of the CNC process
4: Visualizing the execution of a CNC program
5: Program zero and the rectangular coordinate system
6: Determining program zero assignment values
7: Four ways to assign program zero
8: 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 – right at the machine.

Main topics for this lesson:

8

Lesson 6 Presentation links

Why program zero must be assigned for each tool
How program zero assignment values are determined
The zero return position
Alternatives for program zero assignment
Program zero assignment values for center cutting tools
Program zero assignment values for boring bars

Pages 65-76 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. We show *four* popular methods to do so. Only show the method students will be using for your particular machine/s.
- You'll show how program zero is actually assigned in lesson seven.

Key points to make for each topic:

Why program zero must be assigned for each tool
▪ The slide show helps you explain that each cutting tool (especially different types of cutting tools) have a different position in the turret. The tip of a turning tool, for example, will not be in the same position as a drill or boring bar.

How program zero assignment values are determined
▪ The slide show help you briefly introduce the four most common ways to assign program zero. You'll be elaborating on these methods in lesson seven.

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.
▪ two lights – called *axis origin lights* – will come on when the machine is at its zero return position.

Alternatives for program zero assignment
▪ ***Assigning program zero in the program*** – This method is used with old machines that don't have geometry offsets.
Assigning program zero with geometry offsets and the measure function – This method is used if the machine has geometry offsets, but not the work shift function.
▪ ***Assigning program zero with geometry offsets and the work shift function*** – This method is the preferred method unless the machine has a tool touch-off probe.
▪ ***Assigning program zero with the tool touch-off probe*** – This is the preferred method if the machine has a tool touch-off probe.
▪ There is a bit of a discrepancy between the slide show and the student manual. The student manual doesn't show the alternative methods until lesson seven.

Program zero assignment values for center cutting tools
▪ The slide show helps you illustrate how to determine program zero assignment values for drill, taps, reamers, and other center cutting tools.
▪ The X axis program zero assignment value will be the same for any drill, tap, or reamer.

Program zero assignment values for boring bars
▪ The slide show helps you illustrate a simple way of determining the X axis program zero assignment value for any boring bar.

Other considerations about program zero assignment
▪ In the student manual beginning on page 72, we present a few more points related to trial machining, sizing, and replacing dull cutting tools

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.

The procedures shown in lesson six are pretty specific. In the setup- and operation-portion of the course, you'll be presenting more information about the procedures used to 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 using the method you're teaching.

This demonstration will require procedures to start the spindle, 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.

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 six in the student manual.
- Have students fill in the coordinate sheet on page 76.

Exercise (about 20 minutes if done in class)

Have students do exercise number six in the workbook.

Notes:

Lesson Plan | **For Ways To Assign Program Zero** | **Lesson 7**

Explain how program zero is actually assigned. | 20 minutes lecture time

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

1: Machine configurations
2: Understanding turning center speeds and feeds
3: General flow of the CNC process
4: Visualizing the execution of a CNC program
5: Program zero and the rectangular coordinate system
6: Determining program zero assignment values
7: Four ways to assign program zero
8: 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 four ways to assign program zero – and which one should be used depends upon the age of the machine and whether the machine is equipped with certain accessories.

Main topics for this lesson:

6

Lesson 7 Presentation links

Assigning program zero in the program with G50
Assigning program zero with fixture offsets (preferred)
A few more points about program zero assignment

Pages 77-100 in the student manual

Lesson objective: Be sure students understand how program zero is assigned using the method that is most appropriate for their machine).

Like lesson six, 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).

▪For (current) machines, students will be using geometry offsets to assign program zero (use the work shift and measure function).
▪ For old machines (over 20 years old), students will learn how to assign program zero with the G50 command.
▪ Show only the program zero assignment method required for your lab machine/s.
▪ Actually, the slide show for lesson six shows the four ways to assign program zero – the student manual does so here in lesson seven. The student manual provides a concise description of each method – as well as how to choose which method is best.

Key points to make for each topic:

Assigning program zero in the program with G50
▪ Regardless of how program zero is assigned, the program zero assignment values shown in lesson six will be used.
▪ The polarity for program zero assignment values with this method is taken *from program zero to the zero return position* (almost always positive.
▪ A G50 command at the beginning of the program includes the program zero assignment values.
▪ G50 has many limitations and can be difficult – if not dangerous – to use (limitations are shown in the slide show). Geometry offsets eliminate these limitations.
▪ A zero return command (G28) should be included prior to the G50 command to ensure that the machine is in the proper position prior to executing the G50 command.
▪ G50 should only be used to assign program zero if the machine does not have geometry offsets.

Assigning program zero with geometry offsets (most machines)
▪ Regardless of how program zero is assigned, the program zero assignment values shown in lesson six will be used.
▪ With geometry offsets, the polarity for program zero assignment values is taken *from the zero return position to program zero* (almost always negative).
▪ The program zero assignment values are entered into the appropriate geometry offset registers.
▪ If the *work shift value* is used as shown in lesson six, geometry offsets for tools used in the previous job will remain correct.
▪ The *measure function* can facilitate the measurement and entry of geometry offset values.
▪ A tool touch-off probe should be the method of program zero assignment if the machine has one.

A few more points about program zero assignment
▪ The slide show helps you make points about trial machining and sizing – as well as what must be done when dull tools must be replaced.

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

As with lesson six, 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.

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

At the machine (about 20 minutes)

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

If your machine has a tool touch-off probe, show its use. If not show how the measure function helps with the entry of geometry offset values. Show the geometry offset pages.

If you have an old machine that doesn't have geometry offsets, then show how the G50 commands 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 measure and enter geometry offsets, so students can practice with minimal help from you. Have them work with an unused geometry offsets for practicing (like geometry offsets numbered over twenty so they cannot overwrite needed geometry offset values.

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

Homework

- Read all of lesson seven in the student manual.
- Have students fill in the coordinate sheet on page 99.

Exercise (about 20 minutes if done in class)

Have students do exercise number seven in the workbook.

Notes:

Lesson Plan | **Introduction To Programming Words** | **Lesson 8**

Introduce students to the word types used with CNC programs. | 15 minutes lecture time

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

1: Machine configurations
2: Understanding turning center speeds and feeds
3: General flow of the CNC process
4: Visualizing the execution of a CNC program
5: Program zero and the rectangular coordinate system
6: Determining program zero assignment values
7: Four ways to assign program zero
8: Introduction to programming words

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

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

Main topics for this lesson:

Lesson 8 Presentation links

Intro to word types
Word types
O – program number
() – messages
N – sequence number
G – preparatory function
X – X axis designator
Z – Z axis designator
Decimal format (XYZ)
Related words (XYZ)
C – Rotary axis designator

Word types (continued)
R – radius designator
I J K – directional vectors
P – subprogram designator
L – number of executions
F – feedrate
S – spindle speed
T – Turret index
M – miscellaneous functions
/ - block delete
EOB - end of block

Pages 101-112 in the student manual

Lesson objective: Acquaint students with the word types (letter addresses) used in CNC turning 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 turning 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 turret, 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 106). Point out that M words are determined by machine tool builders and can vary from machine to machine – especially for special accessories.

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 eight in the student manual

Exercise (about 15 minutes if done in class)
Have students do exercise number eight 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 most basic types of turning centers? How can you tell the differences?
- What are some of the most basic components of a turning center?
- What are the two linear axes?
- What is the polarity (plus versus minus) for each axis?
- What are the three ways to control a turning 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 are the two related G codes?
- What other features on a turning 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 turning 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 program?
- How do you determine where to place the program zero point?
- When you specify coordinates from program zero, what positioning mode is it called? How do you specify positions in the absolute mode?
- What is the other positioning mode? How do you specify incremental positioning?

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?

Lesson Six – Assigning program zero:
- What are the four ways to assign program zero? Which is best? Why?
- What is the polarity for geometry 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, 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

Programming:

1: Know your machine from a programmer's viewpoint
- 1: Machine configurations
- 2: Understanding turning center speeds and feeds
- 3: General flow of the CNC process
- 4: Visualizing the execution of a CNC program
- 5: Program zero and the rectangular coordinate system
- 6: Determining program zero assignment values
- 7: Four ways to assign program zero
- 8: Introduction to programming words

2: You must prepare to write programs
- 9: Preparation steps for programming

3: Understand the motion types
- 10: Programming the three most basic motion types

4: Know the compensation types
- 11: Introduction to compensation
- 12: Geometry offsets
- 13: Wear offsets
- 14: Tool nose radius compensation

5: You must provide structure to your CNC programs
- 15: Introduction to program structure
- 16: Four types of program format

6: Special features that help with programming
- 17: One-pass canned cycles
- 18: G71/G70 – rough turning and boring multiple repetitive cycles followed by finishing
- 19: G72-G75 – other multiple repetitive cycles
- 20: G76 – Threading multiple repetitive cycle
- 21: Working with subprograms
- 22: Special considerations for Fanuc 0T and 3T controls
- 23: Other special features of programming

---: Appendix – special machine types and accessories

Setup and operation

7: Know your machine from a setup person or operator's viewpoint
- 24: Tasks related to setup and running production
- 25: Buttons and switches on the operation panels

8: Know the three basic modes of operation
- 26: The three modes of operation

9: Understand the importance of procedures
- 27: The key operation procedures

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

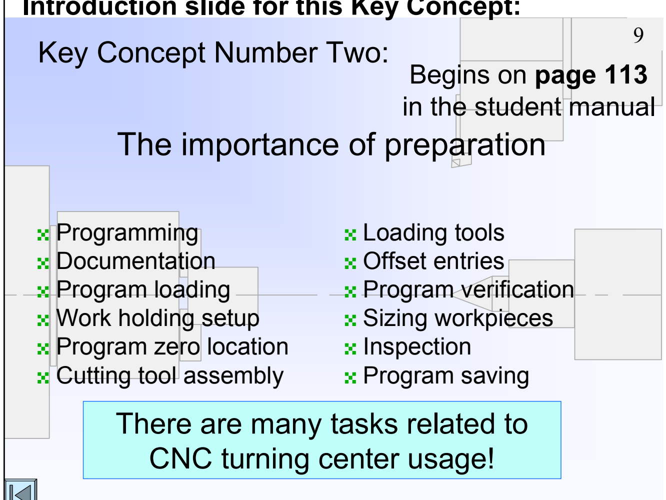
In the course outline to the left, the Key Concepts are shown in bold. The lessons included in each Key Concept are shown as well. As you can see, there are ten Key Concepts further divided into twenty-four lessons.

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 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 nine: Preparation for programming.

The presentation links slide (shown in the lesson plan for lesson 9) 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 113 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):
- 2: You must prepare to write programs
  - **9: Preparation steps for programming**
- 3: Understand the motion types
  - 10: Programming the three most basic motion types
- 4: Know the compensation types
  - 11: Introduction to compensation
  - 12: Geometry offsets
  - 13: Wear offsets

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 turning center can understand.

Main topics for this lesson:

Lesson 9 Presentation links

- The importance of preparation
- Divide and conquer
- Typical mistakes
- **Preparation steps**
  - Study & mark-up print
  - Develop the process
  - Plan the tool paths
  - Do the math
  - Plan the setup

Pages 117-134 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:

The importance of preparation

▪ 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 jaws 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 turning center can understand.

▪ Explain that this form also makes great documentation for anyone who must work on the program in the future.

Plan the tool paths

▪ The slide show and student manual provide you with a way of demonstrating the importance of tool paths. The programmer must know how each tool will move through its machining operation.

▪ Many of the points within each tool path is not actually on the workpiece. Consider roughing operations, for example, which leave finishing stock.

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.)

Plan the setup

▪ Point out that there are many things about the setup that affect the way a program must be written. For example, jaws, tailstock quill, 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.

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 nine in the student manual.
▪ Have students fill in the coordinate sheet on page 132.
▪ Have students plan the tool paths and fill in the coordinate sheet for the drawing on page 133.

Exercise (about 15 minutes if done in class)

Have students do exercise number nine 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.)

Introduce Key Concept number three. | 5 minutes lecture time

Programming:
1: Know your machine from a programmer's viewpoint
1: Machine configurations
2: Understanding turning center speeds and feeds
3: General flow of the CNC process
4: Visualizing the execution of a CNC program
5: Program zero and the rectangular coordinate system
6: Determining program zero assignment values
7: Four ways to assign program zero
8: Introduction to programming words
2: You must prepare to write programs
9: Preparation steps for programming
3: Understand the motion types
10: Programming the three most basic motion types
4: Know the compensation types
11: Introduction to compensation
12: Geometry offsets
13: Wear offsets
14: Tool nose radius compensation
5: You must provide structure to your CNC programs
15: Introduction to program structure
16: Four types of program format
6: Special features that help with programming
17: One-pass canned cycles
18: G71/G70 – rough turning and boring multiple repetitive cycles followed by finishing
19: G72-G75 – other multiple repetitive cycles
20: G76 – Threading multiple repetitive cycle
21: Working with subprograms
22: Special considerations for Fanuc 0T and 3T controls
23: Other special features of programming
---: Appendix – special machine types and accessories

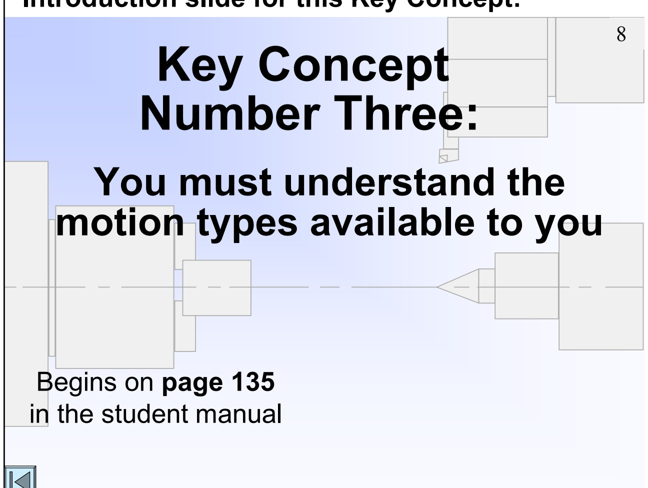
Setup and operation
7: Know your machine from a setup person or operator's viewpoint
24: Tasks related to setup and running production
25: Buttons and switches on the operation panels
8: Know the three basic modes of operation
26: The three modes of operation
9: Understand the importance of procedures
27: The key operation procedures
10: You must know how to safely verify programs
28: 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 turning 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 objective: Help students understand and master the motion types for CNC turning 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 10) 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 135 in the student manual.

Exercise

None for this segment of the course.

Notes:

Lesson Plan | **Programming The Three Most Basic Motion Types** | **Lesson 10**

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

Lessons in Key Concept #3 (you are here):
- 2: You must prepare to write programs
  - 9: Preparation steps for programming
- 3: Understand the motion types
  - **10: Programming the three most basic motion types**
- 4: Know the compensation types
  - 11: Introduction to compensation
  - 12: Geometry offsets
  - 13: Wear offsets

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

Explain that though students can calculate coordinates (from lesson five), 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 ten.

Main topics for this lesson:

Lesson 10 Presentation links

- Key concept #3
  - Understanding interpolation
  - Motion commonalities
  - Understanding the point you're programming
- Motion commands
  - Rapid motion
  - Straight line motion
  - Circular motion (with R word)
    - With directional vectors

Pages 139-156 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 9 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.).

Understanding the point you're programming
- Beginning programmers often have a problem with this. They must understand the actual point on the cutting tool that they are programming.
- For center-cutting hole-machining cutting tools (drills, taps, reamers, etc.) are simple. It is the center of the tool in X and the extreme tip of the tool that is being programmed.
- For single-point turning tools and boring bars, it is the extreme tip of the tool in each axis that is being programmed. The slide show helps you illustrate.
- While this may be a little too early, try to point out that with single point tools, there is a small radius on the cutting tool's tip. For angular and circular machined surfaces, this radius will not remain in contact with the surface being machined – causing a small discrepancy between the programmed path and the machined path. This is the reason why tool nose radius compensation (discussed in Key Concept number four) is required.
- For threading tools and grooving tools, it is the extreme tip of the tool in X and the leading edge of the tool in Z.

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 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 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.

Polar coordinate interpolation

▪ If your machine has live tooling, you may want to mention that control manufacturers like Fanuc do provide special interpolation types for applications that require them. The feature polar coordinate interpolation is used to mill a contour around the outside of a workpiece with a milling cutter held along the Z axis. This feature is described in detail during lesson twenty-three.

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

Once again, everything in lesson ten 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 154 of the student manual and in the workbook exercise for lesson ten). 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 typed and 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 geometry offsets.

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 ten.

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 ten in the student manual.
▪ Have students fill in the blanks for the program on page 154 of the student manual.

Exercise (about 35 minutes if done in class)

Have students do exercise number ten 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, turning tool, facing tool, boring bar, threading tool, grooving tool), 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

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).

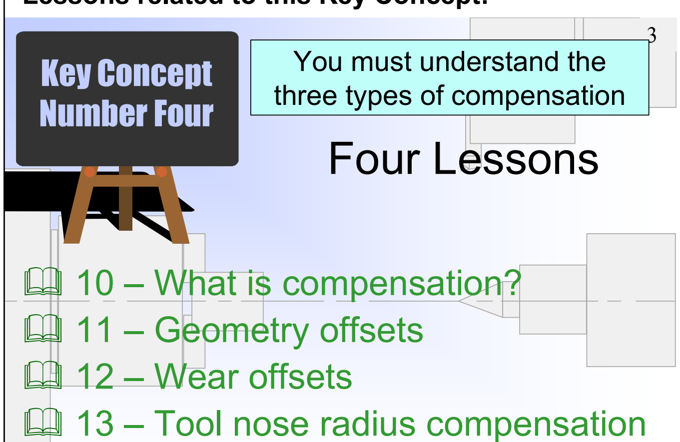
Start by pointing out that a programmer won't know every detail about a setup is made as the program is being written. These compensation types allow the programmer to ignore certain tooling-related information while they write the program.

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 157** in the student manual

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 the compensation types.

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

Introduction to compensation

▪ In the student manual beginning on page 157, you'll find a brief review of tolerance interpretation. Every compensation type requires tolerance-related decisions to be made. When measuring a workpiece, for example, the setup person or operator must determine if the measured attribute is within its tolerance band or not. Even if it is, is it approaching its tolerance limit? Is an adjustment necessary? If an adjustment is necessary, how much adjustment will bring the attribute back to its target value for the next workpiece being machined?

▪ There is a quiz on page 160 that helps you confirm whether or not students understand tolerance interpretation.

▪ This discussion also allows you to introduce trial machining and sizing – topics that will be elaborated upon during each compensation-related lesson.

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 157 in the student manual.
▪ Take the quiz on page 160 of the student manual.

Exercise

None for this segment of the course.

Notes:

Lesson Plan | **Introduction To Compensation** | **Lesson 11**

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

Lessons in Key Concept #4 (you are here):
- 4: Know the compensation types
  - **11: Introduction to compensation**
  - 12: Geometry offsets
  - 13: Wear offsets
  - 14: Tool nose radius compensation
- 5: You must provide structure to your CNC programs
  - 15: Introduction to program structure
  - 16: Four types of program format

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 geometry offsets, they can still write the program.

Main topics for this lesson:

Lesson 11 Presentation links

- Intro to key concept #4
- Marksman analogy
- Understanding offsets
  - Instating offsets
- Trial machining

Pages 163-168 in the student manual

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

During lesson seven, you actually introduced one of the compensation types (geometry offsets). This was necessary as part of explaining program zero – and how program zero is assigned at the machine. In lesson eleven, 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:

Marksman analogy

- The marksman analogy shown in the student manual and slide presentation is remarkably similar to how the compensation types are used on CNC turning 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 (program zero assignment values, wear adjustments, and tool nose radius).
- The slide show helps you describe the various offset pages on a typical CNC turning 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 eleven.
- 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 eleven, beginning on page 163 in the student manual.

Exercise (about 10 minutes if done in class)

Do exercise number eleven in the workbook

Notes:

Lesson Plan | **Geometry Offsets** | **Lesson 12**

Extend what students know about geometry offsets from lessons six and seven. | 10 minutes lecture time

Solicit questions about previous topics. Review lesson eleven.

Most of what students need to know about geometry offsets is presented in lessons six and seven. Indeed, if program zero assignment values are measured at the machine (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 12 Presentation links

Why geometry offsets are required

Review of program zero

Understanding work shift

How accurate are program zero assignment measurements?

Pages 169-177 in the student manual

Lesson objective: To complete students' understanding of geometry offsets.

▪ 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.

▪ This may be considered a review of program zero assignment from lessons six and seven.

Key points to make for each topic:

Why geometry offsets are required

▪ The slide show helps you show the benefits of geometry offsets over assigning program zero in the program with G50.

Review of program zero

▪ The slide show helps you review the assignment of program zero.

Understanding work shift

▪ Again, work shift is described in lesson seven. What is included in this lesson is review.

How accurate are program zero assignment measurements?

▪ This presentation helps you reiterate the importance of trial machining.

▪ The student manual includes a lengthy explanation of the actual workings of geometry offsets and work shift.

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

As stated, this lesson is related to setup and operation – especially setup.

At the machine (15 minutes)

If you haven't already, demonstrate geometry offsets at the machine. Here's the demo that I like to give. After a job has been set up.

▪ Right after power-up, send the machine to its zero return position. Call up the absolute position display page. The X and Z registers will show zero. Point out that right now, the machine thinks the program zero point is the zero return position!

▪ Next, be sure the work shift value is set to zero. By MDI, command a turret index to station one (T0101), including offset invoking. Now show the absolute position display. The values changed to the values in geometry offset number one's registers (plus or minus the wear offset settings. Explain that the T word invokes the geometry offset (first two digits).

▪ The student manual explains the way geometry offsets work in detail.

Lab exercise

We have no suggestions related to the content for this lesson.

Homework

▪ Read all of lesson twelve, beginning on page 169 in the student manual.

▪ Have students fill in the blanks for the program on page 175 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:

Lesson Plan | **Wear Offsets** | **Lesson 13**

Explain how wear offsets can be used for trial machining and sizing. | 25 minutes lecture time

Lessons in Key Concept #4 (you are here):
- 4: Know the compensation types
  - 11: Introduction to compensation
  - 12: Geometry offsets
  - **13: Wear offsets**
  - 14: Tool nose radius compensation
- 5: You must provide structure to your CNC programs
  - 15: Introduction to program structure
  - 16: Four types of program format

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

Point out that wear offsets allow the programmer to write programs without knowing how much tool wear will be experienced during each tool's life. These offsets can also be used when the setup person is worried about whether the initial setting of the geometry offset (program zero assignment) is accurate enough.

Main topics for this lesson:

Lesson 13 Presentation links

- Why wear offsets are required
  - Imperfections during setup
  - Program zero assignment mistakes
  - Tool pressure
  - Tool wear
  - Tool position after replacement
- How wear offsets work
- More on trial machining
- Transitioning from job to job

Pages 177-194 in the student manual

Lesson objective: To help students master the understanding and use of wear offsets.

Point out that the tolerances held by today's turning centers are very small. It is not uncommon for tuning center users to hold overall tolerances of under 0.001 inch on a regular basis. For diameter dimensions, if a tool's program zero assignment is off by 0.0005 inch, the resulting diameter deviation will be 0.001 inch.

- Even if geometry offsets are measured and entered *perfectly*, there can still be a substantial deviation when cutting the first workpiece (tool pressure causes this deviation).
- Wear offsets can be used for trial machining when there is a doubt about the accuracy of the initial geometry offset setting.
- As a tool dulls a small amount of material will be removed from its cutting edge. With small tolerances, this wear may cause the dimension to grow or shrink out of its tolerance band.

Key points to make for each topic:

Why wear offsets are required
- The slide show helps you present five reasons why wear offsets are required.

How wear offsets work
- This topic in the slide show begins by helping you explain how a "total" offset is determined. The control will total the wear offset, the geometry offset and the work shift value to determine the total amount of offset used for program zero assignment.
- We provide some example scenarios for machining with a tool to illustrate how wear offsets work.

More on trial machining
- Point out that while wear offsets can be used for trial machining, it might be better to use geometry offsets. If this is done, the wear offset for each tool can begin the production run at zero. When a dull tool is replaced, the operator will easily know how to reset the wear offset (to zero) after replacing the tool.
- This, of course, is only important with large production runs when tools will need replacing during the production run.

Transitioning from one job to the next
- Be sure students understand that when a tool is machining properly in one job, it will continue to do so in the next – especially if the work shift value is being used. A common mistake is clearing offsets when a job is finished.
- Offsets should only be cleared (set to zero) when the related tool is removed from the turret. In this way, the setup person and operator can rest assured that if a cutting tool is in the turret (from any previous job), the related offsets are still correct.

Special point: When an adjustment is necessary, students must understand, of course, the amount by which an offset must be changed. An explanation of determining the deviation from the measured value to the target value is provided in the introduction to Key Concept number four.

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

As stated, this lesson is related to setup and operation. Be sure all students understand the points made.

At the machine (15 minutes)

- With a simple program (we recommend simply turning one diameter), demonstrate wear offset adjustments. If actually cutting a workpiece, you can show how a small modification to the wear offset can make the cutting tool remove more material.
- Quiz students about which way wear (or geometry) offsets must be changed (plus or minus) in order to leave excess stock for trial machining.
- Show students how to modify the current values within offsets (commonly by using the +INPUT function). With this function, point out that students must only calculate the deviation – not the new offset value.

Lab exercise

With a few sample workpieces available for measuring, provide students with a dimension and tolerance for some "critical" workpiece attributes. Be sure students can come up with the target value (commonly the mean value). After taking physical measurements on the workpieces, have them specify the amount of deviation (as well as polarity) for offset adjustment.

Homework

- Read all of lesson thirteen, beginning on page 177 in the student manual.

Exercise (about 45 minutes if done in class)

Do exercise number thirteen 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. | 15 minutes lecture time

Lessons in Key Concept #4 (you are here):
- 4: Know the compensation types
  - 11: Introduction to compensation
  - 12: Geometry offsets
  - 13: Wear offsets
  - **14: Tool nose radius compensation**
- 5: You must provide structure to your CNC programs
  - 15: Introduction to program structure
  - 16: Four types of program format

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

You may have described how the radius of a single point turning tool or boring bar affects machining during lesson ten (the motion types). In this lesson, you'll be describing how tool nose radius compensation is used.

Main topics for this lesson:

Lesson 14 Presentation links

- Why tool nose radius compensation is required
- Steps to programming
  - Instate
  - Cut the work surface
  - Cancel
- Programming example
- TNR from the setup person's point of view
- Programming TNR offset entries

Pages 195-206 in the student manual

Lesson objective: To ensure that students understand why tool nose radius compensation is required as well as how it is used.

In lesson ten, you explained how there are times when a programmer must compensate for certain cutting tool related attributes when determining the coordinates to be programmed. With a twist drill, for example, the programmer must compensate for the drill lead. With a tap, they must compensate for the number of imperfect threads on the end of the tap. Explain that with any single point cutting tool, there is a small radius on the tip of the tool. And it is this tooling attribute that you'll be dealing with in this lesson.

▪ Compared to cutter radius compensation used with CNC machining centers, tool nose radius compensation for turning centers is quite easy to program.

Key points to make for each topic:

Why tool nose radius compensation is required

▪ The slide show helps you illustrate how the small radius on single point cutting tools is not always in contact with the workpiece when tool nose radius compensation is not used.
▪ Be sure to point out that this is only true with angular and circular surfaces.

Steps to programming

▪ Introduce the three simple steps needed to program tool nose radius compensation – instate, cut the work surface, and cancel.
▪ Explain that instating tool nose radius compensation involves including the appropriate instating word (G41 or G42) in the cutting tool's approach movement to the workpiece. When looking in the direction of the cutting tool's movement, G41 will be used if the cutting tool is on the left side of the work surface. G42 will be used if it is on the right. If machining occurs toward the chuck (as it normally does), G41 will be used for internal boring and G42 will be used for external turning.
▪ The slide show helps you demonstrate how to cut the work surface. This involves simply programming the work surface path with linear and circular motion commands. We also provide some warnings in the slide show about common mistakes made when programming the work surface path.
▪ Canceling tool nose radius compensation involves including a G40 word in the cutting tool's retract motion to the turret index position.

Programming example

▪ The slide show helps you illustrate the programming of tool nose radius compensation with a nice example.

Tool nose radius compensation from the setup person's point of view

▪ Point out that the R and T registers of the cutting tool's offset (we recommend using the geometry offset) must be correctly set in order for tool nose radius compensation to work.
▪ It is commonly the setup person's responsibility to enter these offset values.
▪ Explain that the R register must contain the tool nose radius.
▪ Explain that the T register must contain a code number that tells the machine the style of cutting tool being used (turning tool, boring bar, etc.).
▪ There is a chart showing the code numbers in the student manual as well as in the slide show. We recommend that students try to remember the two most common ones: 2 for a boring bar and 3 for a turning tool.

Programming tool nose radius compensation offset entries

▪ Point out that when a programmer chooses the style of cutting tool to be used for a given application, the radius of the cutting tool will remain fixed. That is, when an operator replaces a dull tool, the new tool will have the same radius as the dull tool. This, of course, means the programmer knows the tool nose radius as the program is written.
▪ The slide show helps you illustrate the use of the G10 command – a command that allows the programming of offset entries.
▪ Point out that programming tool nose radius compensation offset entries will keep the setup person from having to enter them – saving time and minimizing the potential for mistakes.

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

Though a setup person or operator doesn't need to understand how tool nose radius compensation is programmed, they should understand what it is doing for them. So explain the reasons for tool nose radius compensation. Also explain how the tool offsets must be entered (R and T registers) in order for tool nose radius compensation to work.

At the machine (15 minutes)

Run a program that uses tool nose radius compensation to machine angular and circular surfaces in single block mode. After a motion that machines a taper or radius, toggle between the program page (that shows the programmed coordinate) and the position page (that shows the tool's actual position). Point out that the difference between these coordinates is related to tool nose radius compensation.

The machine is automatically recalculating programmed coordinates in order to keep the cutting tool's radius tangent to each surface being machine.

Lab exercise

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

Homework

▪ Read all of lesson fourteen, beginning on page 195 in the student manual.

Exercise (about 40 minutes if done in class)

Do the exercise on page 206 in the student manual.

Do exercise number fourteen 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 eleven – Introduction to compensation:

▪ Quiz students about the terms related to tolerances (mean value, tolerance band, target value, etc.).
▪ Why is compensation required on CNC turning centers? (to deal with unpredictable tooling-related variables).
▪ What is trial machining?
▪ What is the target value for a given dimension?
▪ When must trial machining be done?
▪ Why must sizing be done for workpiece attributes having small tolerances?
▪ Where are trial machining and sizing adjustments made? (in offsets)

Lesson twelve – Geometry offsets

▪ Why are geometry offsets better than G50?
▪ What are program zero assignment values?
▪ What offset values must be changed when a given cutting tool is used from one job to another if our recommend methods are used? (none)
▪ When should offsets for a cutting tool be cleared? (when the tool is removed from the machine)
▪ What is the work shift value?

▪***Lesson twelve – Wear offsets:***

▪ Which kinds of cutting tools require wear offsets? (all)
▪ How are wear offsets specified in the program? (with the second two digits of the T word).
▪ How do you determine the wear offset number? (it should be the same as the tool station number)

Lesson thirteen – Tool nose radius compensation:

▪ What kinds of cutting tools require tool nose radius compensation? (single point cutting tools)
▪ Is tool nose radius compensation required if only straight surfaces are being machined (surfaces parallel to an axis)? (no)
▪ What are the three steps to using tool nose radius compensation?
▪ When is G41 used to instate? G42?
▪ What G code is used to cancel?
▪ What must the setup person do in order for tool nose radius compensation to work?
▪ What command can be used to enter tool nose radius compensation offset values?

Introduce Key Concept number five. | 5 minutes lecture time

Programming:

1: Know your machine from a programmer's viewpoint
- 1: Machine configurations
- 2: Understanding turning center speeds and feeds
- 3: General flow of the CNC process
- 4: Visualizing the execution of a CNC program
- 5: Program zero and the rectangular coordinate system
- 6: Determining program zero assignment values
- 7: Four ways to assign program zero
- 8: Introduction to programming words

2: You must prepare to write programs
- 9: Preparation steps for programming

3: Understand the motion types
- 10: Programming the three most basic motion types

4: Know the compensation types
- 11: Introduction to compensation
- 12: Geometry offsets
- 13: Wear offsets
- 14: Tool nose radius compensation

5: You must provide structure to your CNC programs
- 15: Introduction to program structure
- 16: Four types of program format

6: Special features that help with programming
- 17: One-pass canned cycles
- 18: G71/G70 – rough turning and boring multiple repetitive cycles followed by finishing
- 19: G72-G75 – other multiple repetitive cycles
- 20: G76 – Threading multiple repetitive cycle
- 21: Working with subprograms
- 22: Special considerations for Fanuc 0T and 3T controls
- 23: Other special features of programming

---: Appendix – special machine types and accessories

<u>Setup and operation</u>

7: Know your machine from a setup person or operator's viewpoint
- 24: Tasks related to setup and running production
- 25: Buttons and switches on the operation panels

8: Know the three basic modes of operation
- 26: The three modes of operation

9: Understand the importance of procedures
- 27: The key operation procedures

10: You must know how to safely verify programs
- 28: 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.

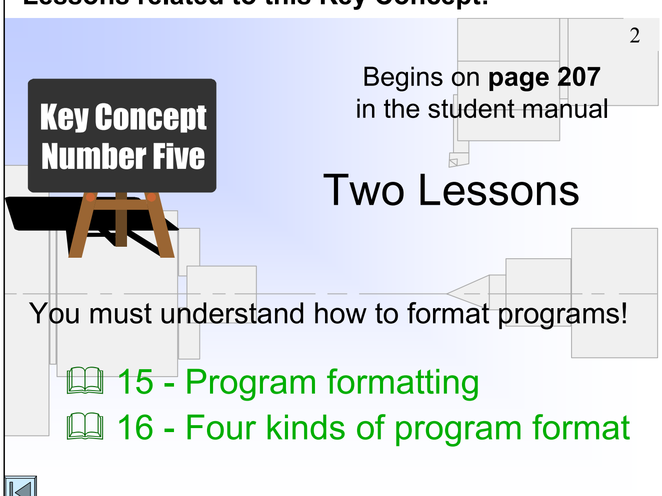
In Key Concept number five, you're going to show them how to become self-sufficient CNC programmers – able to write programs on their own.

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 find the *restart block* from which cutting tools begin.

Lessons related to this Key Concept:



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 207 in the student manual.

Exercise

None for this segment of the course.

Notes:

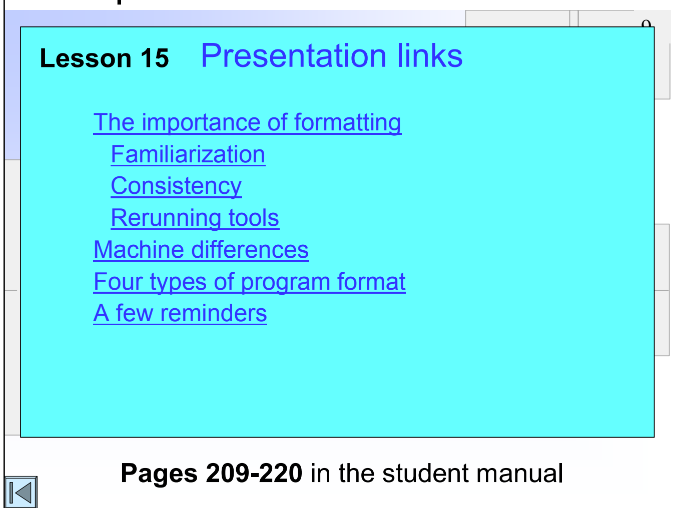
Explain why programs must be formatted using a strict and consistent structure. | 10 minutes lecture time

Lessons in Key Concept #4 (you are here):
- 4: Know the compensation types
  - 11: Introduction to compensation
  - 12: Geometry offsets
  - 13: Wear offsets
  - 14: Tool nose radius compensation
- 5: You must provide structure to your CNC programs
  - **15: Introduction to program structure**
  - 16: Four types of program format

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:



Pages 209-220 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:

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. Two common examples are related to starting the spindle (M03) and coolant (M08). Though the spindle and coolant are never turned off in the program (until the M30 at the end of the program), we include an M03 and M08 at the beginning of each tool. This ensures that if a tool is restarted, the spindle and coolant will come on.

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 turret and tailstock differences and differences in M code numbering.

A few reminders

▪ 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 our given format for using geometry offsets, for example, the restart block is the command that includes the turret index (like T0101).

At the machine (10 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 command 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.

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 209 in the student manual.

Exercise (about 40 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 the four types of program format and show students how to use them. | 20 minutes lecture time

Lessons in Key Concept #4 (you are here):
- 4: Know the compensation types
  - 11: Introduction to compensation
  - 12: Geometry offsets
  - 13: Wear offsets
  - 14: Tool nose radius compensation
- 5: You must provide structure to your CNC programs
  - 15: Introduction to program structure
  - **16: Four types of program format**

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 16 Presentation links

- Format using geometry offsets (preferred)
  - Program startup
  - Tool ending – a common index position
  - Tool startup
  - Program ending
- Format using G50
  - Program startup
  - Tool ending
  - Tool startup
  - Program ending
- Example showing the formats

Pages 221-238 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.
- Though we show two sets of format (using geometry offsets and using G50), remind students that geometry offsets should be their method of choice. The only reason to use G50 is if the machine doesn't have geometry offsets.

Key points to make for each topic:

Format using geometry offsets (preferred)

- We provide two sets of format in the slide show and student manual: for assigning program zero with geometry offsets and for assigning program zero with G50 commands. To avoid confusion, pick and present only the one that applies best to your lab machine/s (probably for using geometry 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 and M30). This should finalize any concerns and questions that students have about the most common CNC words.

Format using G50

- Once again, point out that this format should only be used if the machine doesn't have geometry offsets – as will be the case with old machines.
- 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.

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 sixteen that begins on page 221 in the student manual.
- Write the program for the job shown on page 237 of the student manual.

Exercise (about 50 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:

Introduce Key Concept number six. | 10 minutes lecture time

Programming:

1: Know your machine from a programmer's viewpoint
- 1: Machine configurations
- 2: Understanding turning center speeds and feeds
- 3: General flow of the CNC process
- 4: Visualizing the execution of a CNC program
- 5: Program zero and the rectangular coordinate system
- 6: Determining program zero assignment values
- 7: Four ways to assign program zero
- 8: Introduction to programming words

2: You must prepare to write programs
- 9: Preparation steps for programming

3: Understand the motion types
- 10: Programming the three most basic motion types

4: Know the compensation types
- 11: Introduction to compensation
- 12: Geometry offsets
- 13: Wear offsets
- 14: Tool nose radius compensation

5: You must provide structure to your CNC programs
- 15: Introduction to program structure
- 16: Four types of program format

6: Special features that help with programming
- 17: One-pass canned cycles
- 18: G71/G70 – rough turning and boring multiple repetitive cycles followed by finishing
- 19: G72-G75 – other multiple repetitive cycles
- 20: G76 – Threading multiple repetitive cycle
- 21: Working with subprograms
- 22: Special considerations for Fanuc 0T and 3T controls
- 23: Other special features of programming

---: Appendix – special machine types and accessories

__Setup and operation__

7: Know your machine from a setup person or operator's viewpoint
- 24: Tasks related to setup and running production
- 25: Buttons and switches on the operation panels

8: Know the three basic modes of operation
- 26: The three modes of operation

9: Understand the importance of procedures
- 27: The key operation procedures

10: You must know how to safely verify programs
- 28: 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).

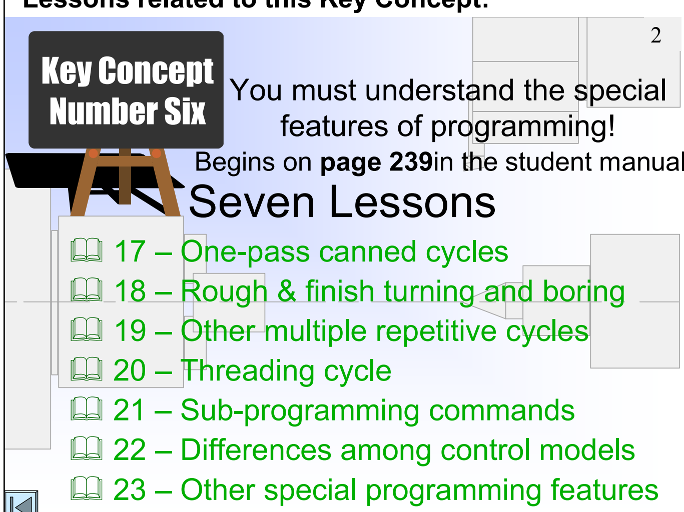
In Key Concept number six, you'll be showing several features that shorten programs, make programming easier, and in general facilitate the programming process.

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:



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.

- While doing exercise, students may have been questioning the difficulty of programming rough machining operations (rough turning and rough boring) – 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 *many* programming tasks, including rough turning, rough boring, and rough facing.

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

▪ One-pass canned cycles are not very helpful, and have been replaced with certain multiple repetitive cycles.
▪ Rough and finish turning, facing and boring are quite easy to program. It requires the programmer to simply provide a finish pass definition and based upon but one command the machine will do all the roughing.
▪ Other multiple repetitive cycles are available for peck drilling and grooving, but they are not very helpful.
▪ The multiple repetitive cycle for threading is very helpful. It allows the programming of an entire thread with one command.
▪ 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 accessories that are commonly equipped with CNC turning centers 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 239 in the student manual.

Exercise

None for this segment of the course.

Notes:

Help students understand the features and limitations of these cycles. | 10 minutes lecture time

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

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 machining operations using G00, G01, G02, and G03. Now you'll begin presenting how the programming of certain machining operations can be simplified. However, the one-pass canned cycles are not so helpful. We include them for the sake of completeness, but there are much more helpful cycles coming up in future lessons.

Main topics for this lesson:

Lesson 17 Presentation links

Key concept number six
Cycle consistencies
One pass turning or boring cycle
One pass facing cycle
One pass threading cycle

Pages 241-246 in the student manual

Lesson objective: Help students master the programming of one-pass canned cycles – and to know their limitations.

Students have worked on several programs that perform rough and finish operations. But they've used G00 and G01 to do so. Now you'll help them master the programming of these operations using special cycles. Again, one-pass canned cycles are not very helpful. They have been replaced by multiple repetitive cycles.

- As the name implies, each one-pass canned cycle will cause the machine to make one pass – a turning pass, a facing pass, or a threading pass.
- 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.

Cycle consistencies
- These consistencies apply to one-pass canned cycles as well as multiple repetitive cycles.
- Point out that all cycles can be used for internal work as well as external work.
- Point out that all canned cycles require the programmer to send the tool to a convenient starting position. This position will vary for each canned cycle, but it is from this position that the cycle will begin.
- Also make it clear that all canned cycles will leave the tool back at the convenient starting position when the cycle ends.

One pass turning and boring cycle
- The slide show helps you explain the G90 – one pass turning and boring cycle. An example is also provided in the student manual.

One pass facing cycle
- In similar fashion, the slide show helps you explain the G94 one pass facing cycle.

One pass threading cycle
- And finally, the slide show helps you explain the one-pass threading cycle, G92.
- Once again, point out that all three of these cycles are quite limited – and there are multiple repetitive cycles that are much more helpful.

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 (10 minutes)

You may want to demonstrate the use of each of these cycles with an example program at the machine. But again, we don't recommend spending too much time on them – the multiple repetitive cycles are much more helpful.

Lab exercise

We have no suggestions related to the material covered in this lesson.

Homework	Exercise (about 10 minutes if done in class)
▪ Read all of lesson seventeen that begins on page 241 in the student manual.	Do exercise number seventeen in the workbook.

Notes:

Help students understand how these very helpful cycles are used. | 20 minutes lecture time

Solicit questions about previous topics.

As stated in lesson seventeen, the one-pass canned cycles have been replaced with multiple repetitive cycles. In this lesson, you'll present two very popular multiple repetitive cycles. G71 is used for rough turning and rough boring. G70 is used for finishing.

Main topics for this lesson:

Lesson 18 Presentation links

- Introduction to multiple repetitive cycles
- How G71 works
- Words in the G71 command
- The finish pass definition
  - An example of G71 for turning
- Finishing with G70
- Using G71 and G70 for rough boring
  - An example program of G71 & G71 for boring
- What about tool nose radius compensation?

Pages 247-258 in the student manual

Lesson objective: Help students master the simple way to program rough turning and rough boring followed by finishing.

Students have worked on several programs that perform rough and finish operations. But they've used G00 and G01 to do so. Now you'll help them master the programming of these operations using special cycles.

- With G71, the programmer will define the finish pass and provide one simple command that specifies how roughing must be done.
- The machine will figure out how to rough turn or rough bore, regardless of how many passes is required.

Key points to make for each topic:

Introduction to multiple repetitive cycles
- The slide show helps you introduce the various multiple repetitive cycles and specifies which ones will be very helpful.

How G71 works
- The slide show helps you graphically illustrate the two phases of G71. In phase one, the machine will take a series of roughing passes, leaving steps on the workpiece. In phase two, the machine will take a semi-finish pass, leaving the specified amount of finishing stock.

Words in the G71 command
- The slide show helps you introduce the meaning of each word needed in the G71 command, including P and Q (used to specify the starting and ending block for the finish pass definition), D (used to specify depth of cut), U and W (used to specify finishing stock), and F (the feedrate used for the entire roughing cycle).

The finish pass definition
- The slide show helps you point out that right after the G71 command, the programmer specifies the finish pass.

Finishing cycle – G70
- To keep from having to repeat the finish pass definition for the finishing tool, G70 lets the programmer specify the starting and ending block for the finish pass.

Using G71 and G70 for rough and finish boring
- The slide show helps you illustrate the small differences for using these multiple repetitive cycles for internal work.

What about tool nose radius compensation?
- The slide show helps you illustrate that it is still possible (and easy) to use tool nose radius compensation for the finishing tool/s when using G71 and G70.

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

While setup people won't have to program multiple repetitive cycles, they may have to edit them 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)

Students will be writing two practice programs that incorporate G71 and G70. Load one into the machine and demonstrate its use. Be sure to modify the depth of cut to show how easy it is to generate a greater or smaller number of roughing passes by simply changing one word.

Lab exercise

With your supervision, have students load and verify the program written in the programming activity for this lesson

Homework

▪ Read all of lesson eighteen that begins on page 247 in the student manual.
▪ Write the program for the exercise shown on page 257.

Exercise (about 40 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:

Help students understand the rough facing cycle. | 10 minutes lecture time

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

Solicit questions about previous topics.

Students now know how to easily command rough turning and rough boring with G71. In this lesson you'll be presenting G72, which can be used in a similar fashion for rough facing (still followed by G70 for finish facing). The other three multiple repetitive cycles (G73, G74, and G75) are not so helpful. In this lesson, you'll be explaining why.

Main topics for this lesson:

Lesson 19 Presentation links

- G72 rough facing cycle
- G73 pattern repeating cycle
- G74 peck drilling cycle
- G75 grooving cycle

Pages 259-268 in the student manual

Lesson objective: Help students understand and master four other multiple repetitive cycles.

There are times when a workpiece to be rough machined is in the form of a large diameter yet short workpiece (like a flange). For this kind of workpiece, it makes more sense to rough face it than to rough turn it – which is the function of G72.

- G73 is used for "pattern repeating", but there are not many feasible applications for its use.
- G74 is used for peck drilling, but it only breaks the chip as the hole is drilled. It does not clear the chips from the hole.
- And G75 is called the grooving cycle, but it simply plunges the groove once. It will not form chamfers or radii on the groove's corners.

Key points to make for each topic:

G72 rough facing cycle
- The slide show helps you explain the minimal differences between G71 and G72. An example program is also provided in the slide show and in the student manual.

G73 pattern repeating cycle
- While we show a pretty good application for G73 in the slide show and in the student manual, frankly speaking, few programmers use G73. Some try to use it for castings and forgings with limited success.

G74 peck drilling cycle
- While G74 works nicely with gummy materials to break chips, it will not clear chips between pecks – as is necessary for deep holes. Note that we provide an example of deep hole peck drilling using G00 and G01.

G75 grooving cycle
- This cycle only makes one grooving pass – assuming that the grooving tool is the same width as the groove. Also, it will not make chamfers or radii on the groove's corners. For these reasons, most programmers program grooving operations long-hand – with G00 and G01.

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

While setup people won't have to program multiple repetitive cycles, again, they may have to edit them during program verification. You may want to explain the words in the G72 command.

At the machine (10 minutes)

Students will be writing a practice program that incorporates G72 and G70. Load it into the machine and demonstrate its use. As with your demonstration of G71, be sure to modify the depth of cut to show how easy it is to generate a greater or smaller number of roughing passes by simply changing one word.

Lab exercise

We have no suggestions related to the material contained in this lesson.

Homework

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

Exercise (about 40 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:

Lesson Plan	G76 Threading Multiple Repetitive Cycle	Lesson 20

Help students understand how to easily program threading operations. | 20 minutes lecture time

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

6: Special features that help with programming
- 17: One-pass canned cycles
- 18: G71/G70 – rough turning and boring multiple repetitive cycles followed by finishing
- 19: G72-G75 – other multiple repetitive cycles
- **20: G76 – Threading multiple repetitive cycle**
- 21: Working with subprograms
- 22: Special considerations for Fanuc 0T and 3T controls
- 23: Other special features of programming
- ---: Appendix – special machine types and accessories

Solicit questions about previous topics.

Students have seen the G92 on-pass canned cycle for threading. But as you have presented, G92 has several limitations – the number of passes and the depth of each pass must be calculated, it is difficult to program an in-feed angle, and if something is wrong, it is difficult to modify the threading operation. These limitations have been overcome with G76.

Main topics for this lesson:

Lesson 20 Presentation links

- How threading is done on a lathe
- G32 – great for tapping
- Point on the threading tool that is programmed
- G76 threading command
  - Words involved
  - Example program
- Other important points about threading
- Taper threading
- Multiple start threads

Pages 269-280 in the student manual

Lesson objective: Help students understand and master the G76 command.

G76 makes it easy to program threading operations, regardless of how many passes are needed to chase the thread.

- The threading tool is first positioned to the convenient starting position.
- The G76 command is then given, specifying the ending diameter (major or minor diameter of the thread), the ending point of the thread (Z), the thread depth (K), the depth of the first pass (D), the tool angle (A), and the pitch (F).

Key points to make for each topic:

How threading is done on a lathe
- We begin by introducing threading – which some of your students may be quite familiar with.

G32 – great for tapping
- G32 is like G01 in that one motion will be made per G32. But unlike G01, G32 will disable feedrate override and feed hold, making it quite useful for programming tapping commands.

Point on the threading tool that is programmed
- The slide show helps you point out that students should program from the leading edge of the threading tool, not the tip (point).

G76 threading command
- This is the heart of your presentation for this lesson. The slide show helps you explain the words involved with G76, as well as how to program threads.
- The slide show also helps you illustrate the programming of taper threads – as well as multiple start threads.

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

While setup people won't have to program multiple repetitive cycles, again, they may have to edit them during program verification. You may want to explain the words in the G76 command.

At the machine (15 minutes)

Students will be writing a practice program that incorporates G76. Load it into the machine and demonstrate its use. As with your demonstration of G71, be sure to modify the depth of the first pass to show how easy it is to generate a greater or smaller number of passes by simply changing one word.

Lab exercise

We have no suggestions related to the material presented in this lesson.

Homework	**Exercise (about 40 minutes if done in class)**
▪ Read all of lesson twenty that begins on page 269 in the student manual.	Do exercise number twenty 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
  - 17: One-pass canned cycles
  - 18: G71/G70 – rough turning and boring multiple repetitive cycles followed by finishing
  - 19: G72-G75 – other multiple repetitive cycles
  - 20: G76 – Threading multiple repetitive cycle
  - **21: Working with subprograms**
  - 22: Special considerations for Fanuc 0T and 3T controls
  - 23: Other special features of programming
  - ---: Appendix – special machine types and accessories

Solicit questions about previous topics. Quickly review your presentations for multiple repetitive cycles.

Point out that there are times when a programmer must repeat a series of commands within a program. Have them consider, for example, the machining of identical grooves on a workpiece.

Main topics for this lesson:

Lesson 21 Presentation links

- How sub-programming works
- Applications for sub-programming
- Related words
- Examples
  - Multiple identical machining operations (grooves)
  - Flip jobs
  - Utility application (bar feeding)
- Introduction to parametric programming

Pages 281-295 in the student manual

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

You've presented one time when commands in a program must be repeated: multiple identical grooves. 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 machining multiple identical grooves – nicely illustrating how helpful a subprogram can be. We also show how subprograms can help with flip jobs and with bar-feeding.
- 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 how to program in the incremental positioning mode (using U and W instead of X and Z).
- The *control programs* application example is related to turning centers that machine flip jobs (machining half then turning the part around in the middle of the cycles.
- We also show a subprogram application that is helpful with turning centers that have bar feeders. But of course, not all turning centers have this accessory.

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.

An exception to this statement is if you're using a subprogram for bar-feeding. This program must, of course, be modified from one job to the next. It is the setup person, of course, that must modify the subprogram.

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 twenty-one that begins on page 281 in the student manual.

Exercise (about 40 minutes if done in class)

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

Notes:

Lesson Plan | **Special 0T and 3T Considerations** | **Lesson 22**

Present the minor programming differences among Fanuc control models. | 10 minutes lecture time

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

6: Special features that help with programming
- 17: One-pass canned cycles
- 18: G71/G70 – rough turning and boring multiple repetitive cycles followed by finishing
- 19: G72-G75 – other multiple repetitive cycles
- 20: G76 – Threading multiple repetitive cycle
- 21: Working with subprograms
- **22: Special considerations for Fanuc 0T and 3T controls**
- 23: Other special features of programming
- ---: Appendix – special machine types and accessories

Solicit questions about previous topics.

Point out that there are some minor differences with regard to how certain Fanuc controls are programmed. Examples shown so far have been for the majority of Fanuc controls. Information in this lesson pertains only to 0T, 3T, and some 18T controls. You may, of course, skip this lesson if your students won't be working with these controls.

Main topics for this lesson:

Lesson 22 Presentation links

- Two styles of programming
- Multiple repetitive cycle differences
  - G71
  - G72
  - G73
  - G76
- Sub-programming differences

Pages 297-300 in the student manual

Lesson objective: Be sure students understand the minor programming differences among Fanuc controls.

All examples shown to this point have been for the majority of Fanuc control models. You'll now present the minor programming differences for 0T, 3T, and some 18T controls.

- You may want to expand this presentation to include other controls that you know your students will be working with. While it is beyond the scope of this curriculum, you may, for example, want to discuss some of the minor programming differences between Fanuc controls and others (like Okuma, Allen Bradly, and Acromatic controls).

Key points to make for each topic:

Two styles of programming
- The slide show helps you explain the second style of programming multiple repetitive cycles which commonly requires two command blocks for G71, G72, G73, and G76.

Multiple repetitive cycle differences
- The slide show then helps you show the actual differences for each multiple repetitive cycle.
- Be sure to point out that the second style of programming actually has some advantages – especially for G76 threading (allowing minimum depth of cut, final pass depth, and number of spring passes to be programmed).

Sub-programming differences
- Finally, the slide show helps you explain the minor format difference for the M98 command's P word and L word.

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 the material in this lesson – unless, of course, they'll be modifying programs at the machine. If they will, you should present this short lesson.

At the machine

We have no suggestions related to the material presented in this lesson.

Lab exercise

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

Homework
- Read all of lesson two that begins on page 297 in the student manual.

Exercise (about 15 minutes if done in class)

Do exercise number twenty-two in the workbook.

Notes:

Lesson Plan | **Other Special Programming Features** | **Lesson 23**

Show a few more programming features that can facilitate programming. | 20 minutes lecture time

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

6: Special features that help with programming
- 17: One-pass canned cycles
- 18: G71/G70 – rough turning and boring multiple repetitive cycles followed by finishing
- 19: G72-G75 – other multiple repetitive cycles
- 20: G76 – Threading multiple repetitive cycle
- 21: Working with subprograms
- 22: Special considerations for Fanuc 0T and 3T controls
- **23: Other special features of programming**
- ---: Appendix – special machine types and accessories

Solicit questions about previous topics. Quickly review multiple repetitive 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 23 Presentation links

- Block delete techniques
  - How block delete works
  - Another optional stop
  - Help with trial machining
- N word techniques
  - Changing machining order
- Documenting in the program
- Other G codes of importance
  - G04 - dwell
  - G10 – data setting
  - G20/G21 – inch metric
- Other M codes of interest
  - M00 – program stop
  - M02 – end of program
  - M13/M14 – Spindle & coolant
- Understanding parameters

Pages 301-312 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:

- Block delete is a feature needed and used by most companies.
- Statement labels are rarely needed.

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.
- 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*.
- 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.

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, turning 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 (15 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.

You may also want to demonstrate how find out whether a given Fanuc control has an optional G code. In MDI, enter and execute the G code by itself. If you receive no alarm – or if the alarm is related to the format for the G code, the machine has it. If you receive alarm 10 (unusable G code), the machine does not have it.

Lab exercise

We have no recommendations related to the material presented in this lesson.

Homework

▪ Read all of lesson twenty-three that begins on page 301 in the student manual.

Exercise (about 15 minutes if done in class)

Do exercise number twenty-three in the workbook.

The Appendix in the student manual

The Appendix immediately follows lesson twenty-three in the student manual. Since it is related primarily to programming, we place it right after the programming presentations. Here is a list of the materials covered in the appendix. Based upon your knowledge of what your students will need upon completing this course, you can cover the related material.

Ending a bar feed program 326
An example bar feeding program 326

Notes:

Introduce Key Concept number seven. | 15 minutes lecture time

Programming:

1: Know your machine from a programmer's viewpoint
- 1: Machine configurations
- 2: Understanding turning center speeds and feeds
- 3: General flow of the CNC process
- 4: Visualizing the execution of a CNC program
- 5: Program zero and the rectangular coordinate system
- 6: Determining program zero assignment values
- 7: Four ways to assign program zero
- 8: Introduction to programming words

2: You must prepare to write programs
- 9: Preparation steps for programming

3: Understand the motion types
- 10: Programming the three most basic motion types

4: Know the compensation types
- 11: Introduction to compensation
- 12: Geometry offsets
- 13: Wear offsets
- 14: Tool nose radius compensation

5: You must provide structure to your CNC programs
- 15: Introduction to program structure
- 16: Four types of program format

6: Special features that help with programming
- 17: One-pass canned cycles
- 18: G71/G70 – rough turning and boring multiple repetitive cycles followed by finishing
- 19: G72-G75 – other multiple repetitive cycles
- 20: G76 – Threading multiple repetitive cycle
- 21: Working with subprograms
- 22: Special considerations for Fanuc 0T and 3T controls
- 23: Other special features of programming

---: Appendix – special machine types and accessories

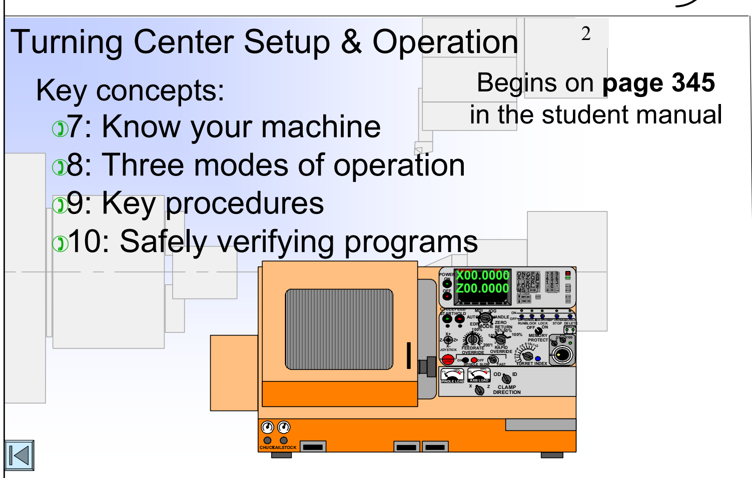
Setup and operation

7: Know your machine from a setup person or operator's viewpoint
- 24: Tasks related to setup and running production
- 25: Buttons and switches on the operation panels

8: Know the three basic modes of operation
- 26: The three modes of operation

9: Understand the importance of procedures
- 27: The key operation procedures

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



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

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 twenty-three 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, wear and geometry offsets, and tool nose 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 turning center: programming, setup, and operation.

However, because of the numbers involved, 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 and go through the curriculum in order. 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 345-349 in the student manual help you explain the most rudimentary procedures needed to run a turning 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 heading "Programmable functions of turning centers"

From Lesson Two: Understanding turning center speeds and feeds
▪ Present the entire lesson

From Lesson Three: General flow of the programming process
▪ Present the entire lesson

From Lesson Five: Program zero and the rectangular coordinate system
▪ Present from the beginning to the heading "Wisely choosing the program zero point location"

From Lesson Six: Determining program zero point assignment values
▪ Present the entire lesson

From Lesson Seven: Four ways to assign program zero
▪ Present the entire lesson

From Lesson Eleven: Introduction to compensation
▪ Present the entire lesson (also Present the introduction to Key Concept Four that precedes this lesson)

From Lesson Thirteen: Wear offsets
▪ Present the entire lesson

From Lesson Fourteen: Tool nose radius compensation
▪ Present from the beginning to the heading "Steps to programming tool nose radius compensation"
▪ Present from the heading "Tool nose radius compensation from the setup person's point of view" to the heading "Programming tool nose radius compensation values"

From Lesson Fifteen: 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 recommendations at this point.

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 345 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

Lessons in Key Concept #7 (you are here):
- 7: Know your machine from a setup person or operator's viewpoint
  - **24: Tasks related to setup and running production**
  - 25: Buttons and switches on the operation panels
- 8: Know the three basic modes of operation
  - 26: The three modes of operation
- 9: Understand the importance of procedures
  - 27: The key operation procedures

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 turning 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 24 Presentation links

- Introduction to setup & operation
  - Key concept number eight
- Operator responsibilities
- Setup tasks versus operation tasks
  - Tasks related to setup
  - Tasks related to maintaining production

Pages 350-374 in the student manual

Lesson objective: Help students understand the tasks related to setting up and running a CNC turning 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 turning 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 is 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 then 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 352). 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 versed with the tasks performed by setup people and operators. Without this understanding a programmer cannot provide adequate documentation and direction.
▪ 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 turning 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-four that begins on page 350 in the student manual.
- Do the exercise on page 374 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-four 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):
- 7: Know your machine from a setup person or operator's viewpoint
  - 24: Tasks related to setup and running production
  - **25: Buttons and switches on the operation panels**
- 8: Know the three basic modes of operation
  - 26: The three modes of operation
- 9: Understand the importance of procedures
  - 27: The key operation procedures

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 25 Presentation links

- The two operation panels
- The control panel
  - Keyboard
  - Display screen functions
    - Position
    - Program
    - Offset
    - Program check
- Machine panel
  - Mode switch
  - Cycle start & feed hold
  - Emergency stop
  - Feedrate override
  - Jog feedrate
  - Rapid override
  - Handwheel
  - Conditional switches
    - (Single block, dry run, optional stop, etc.)

Pages 375-384 in the student manual

Lesson objective: Help students understand all of the buttons and switches on your turning 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 turning 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 a tailstock or bar feeder).

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 26).

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 turning 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 (25 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-five that begins on page 375 in the student manual.

Exercise (about 10 minutes if done in class)

Do exercise number twenty-five in the workbook.

Notes:

Introduce Key Concept number eight. | 5 minutes lecture time

Programming:

1: Know your machine from a programmer's viewpoint
- 1: Machine configurations
- 2: Understanding turning center speeds and feeds
- 3: General flow of the CNC process
- 4: Visualizing the execution of a CNC program
- 5: Program zero and the rectangular coordinate system
- 6: Determining program zero assignment values
- 7: Four ways to assign program zero
- 8: Introduction to programming words

2: You must prepare to write programs
- 9: Preparation steps for programming

3: Understand the motion types
- 10: Programming the three most basic motion types

4: Know the compensation types
- 11: Introduction to compensation
- 12: Geometry offsets
- 13: Wear offsets
- 14: Tool nose radius compensation

5: You must provide structure to your CNC programs
- 15: Introduction to program structure
- 16: Four types of program format

6: Special features that help with programming
- 17: One-pass canned cycles
- 18: G71/G70 – rough turning and boring multiple repetitive cycles followed by finishing
- 19: G72-G75 – other multiple repetitive cycles
- 20: G76 – Threading multiple repetitive cycle
- 21: Working with subprograms
- 22: Special considerations for Fanuc 0T and 3T controls
- 23: Other special features of programming

---: Appendix – special machine types and accessories

Setup and operation

7: Know your machine from a setup person or operator's viewpoint
- 24: Tasks related to setup and running production
- 25: Buttons and switches on the operation panels

8: Know the three basic modes of operation
- 26: The three modes of operation

9: Understand the importance of procedures
- 27: The key operation procedures

10: You must know how to safely verify programs
- 28: 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 turning 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 385** 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 turning 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	**Exercise**
▪ None.	None for this segment of the class.

Notes:

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

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

7: Know your machine from a setup person or operator's viewpoint
- 24: Tasks related to setup and running production
- 25: Buttons and switches on the operation panels

8: Know the three basic modes of operation
- **26: The three modes of operation**

9: Understand the importance of procedures
- 27: 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 26 Presentation links

The importance of the mode switch

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

Pages 385-390 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. Some turning centers, for example, provide no manual means to cause the turret to index. 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 26 that begins on page 385 in the student manual.

Exercise (about 10 minutes if done in class)

Do exercise number 26 in the workbook.

Notes:

Introduce Key Concept number nine. | 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.

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 391**
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, I feel it is the programmer's responsibility to document the procedures needed to run a CNC turning center. How can they do so if they don't know which procedures to document?

At the machine

No suggestions.

Lab exercise

No suggestions at this time.

Homework	Exercise
▪ None.	None for this segment of the class.

Notes:

Lesson Plan	**The Key Operation Procedures**	**Lesson 27**

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

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

- 7: Know your machine from a setup person or operator's viewpoint
  - 24: Tasks related to setup and running production
  - 25: Buttons and switches on the operation panels
- 8: Know the three basic modes of operation
  - 26: The three modes of operation
- 9: Understand the importance of procedures
  - **27: The key operation procedures**

Solicit questions about previous topics.

Frankly speaking, running a CNC turning 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 27 Presentation links

The importance of procedures
Manual procedures
MDI procedures
Setup procedures
Program manipulation procedures
Program running procedures

Pages 391-402 in the student manual

Lesson objective: Provide students with the procedures they need to run a CNC turning 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 turning 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 394), 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 program zero 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-eight.

At the machine (15 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 twenty-seven that begins on page 391 in the student manual.

Exercise (about 10 minutes if done in class)

Do exercise number twenty-seven 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 403**
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
  - 26: The three modes of operation
- 9: Understand the importance of procedures
  - 27: The key operation procedures
- 10: You must know how to safely verify programs
  - **28: Program verification**

Solicit questions about previous topics. Review lesson 24 (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 28 Presentation links

- Safety priorities
- New versus proven programs
- Program verification procedures
  - Free flowing dry run
  - Normal air cutting run
  - Cautiously running first workpieces
- Rerunning tools
- Completing a production run

Pages 405-413 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 408 begins this example). A similar example scenario is shown in lesson twenty-four, but there are no mistakes. With the example job shown here in lesson twenty-eight, 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 turning 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 408) 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 geometry offset entries).

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 twenty-eight that begins on page 405 in the student manual.

Exercise (about 10 minutes if done in class)

Do exercise number twenty-eight 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: 15 minutes
  - Present lesson five: 20 minutes
  - Present lesson six: 20 minutes
  - Present lesson seven: 20 minutes
  - Present lesson eight: 15 minutes
- Introduce Key Concept two: 10 minutes
  - Present lesson nine: 20 minutes
- Introduce Key Concept three: 5 minutes
  - Present lesson ten: 35 minutes
- Introduce Key Concept four: 15 minutes
  - Present lesson eleven: 20 minutes
  - Present lesson twelve: 10 minutes
  - Present lesson thirteen: 25 minutes
  - Present lesson fourteen: 15 minutes
- Introduce Key Concept five: 5 minutes
  - Present lesson fifteen: 10 minutes
  - Present lesson sixteen: 20 minutes
- Introduce Key Concept six: 10 minutes
  - Present lesson seventeen: 10 minutes
  - Present lesson eighteen: 20 minutes
  - Present lesson nineteen: 10 minutes
  - Present lesson twenty: 20 minutes
  - Present lesson twenty-one: 15 minutes
  - Present lesson twenty-two: 10 minutes
  - Present lesson twenty-three: 20 minutes
- Introduce Key Concept seven: 15 minutes
  - Present lesson twenty-four: 20 minutes
  - Present lesson twenty-five: 20 minutes
- Introduce Key Concept eight: 5 minutes
  - Present lesson twenty-six: 10 minutes
- Introduce Key Concept nine: 5 minutes
  - Present lesson twenty-seven: 10 minutes
- Introduce Key Concept ten: 5 minutes
  - Present lesson twenty-eight: 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: 15 minutes
  - Lesson four: 5 minutes per student
  - Lesson five: 20 minutes
  - Lesson six: 20 minutes
  - Lesson seven: 5 minutes per student
  - Lesson eight: 10 minutes
- Key Concept two: 10 minutes
  - Lesson nine: none
- Key Concept three: 10 minutes
  - Lesson ten: 30 minutes
- Key Concept four: none
  - Lesson eleven: 10 minutes
  - Lesson twelve: 15 minutes
  - Lesson thirteen: 15 minutes
  - Lesson fourteen: 15 minutes
- Key Concept five: none
  - Lesson fifteen: 10 minutes
  - Lesson sixteen: 20 minutes
- Key Concept six: none
  - Lesson seventeen: 10 minutes
  - Lesson eighteen: 15 minutes
  - Lesson nineteen: 10 minutes
  - Lesson twenty: 15 minutes
  - Lesson twenty-one: 10 minutes
  - Lesson twenty-two: none
  - Lesson twenty-three: 15 minutes
- Key Concept seven: none
  - Lesson twenty-four: 15 minutes
  - Lesson twenty-five: 25 minutes
- Key Concept eight: none
  - Lesson twenty-six: 10 minutes
- Key Concept nine: none
  - Lesson twenty-seven: 15 minutes
- Key Concept ten: none
  - Lesson twenty-eight: 30 minutes

Approximate total time at the machine: 7 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.