Getting Started with the Machining Center Programming Curriculum

Our course curriculums have been designed to address the needs of instructors working in three similar, but subtly different, learning environments. First, instructors working for colleges, vocational schools, technical schools, and universities will find our ready-made course curriculums especially easy to learn and implement. They make it especially easy to prepare to teach each lesson. It has been our experience that instructors teaching their first CNC courses for technical schools come from one of but two backgrounds. Either they have extensive CNC experience (possibly through working for a manufacturing company), but limited teaching experience --- or they have extensive teaching experience (teaching other courses for the school), but limited CNC experience. It is rare to find a new CNC instructor who has extensive experience in both fields. These course curriculums will help in both cases.

Our proven curriculums, structured lesson plans, audio guidance, instructor notes for each lesson, and concise lesson format will minimize the amount of work an instructor must do in preparation for teaching a new course -- and minimize the work that must be done when getting ready to teach each lesson *during* the course. The proven *key concepts* approach and fly-in text boxes make it easy for even an inexperienced instructor to stay on track during each presentation. And - the high animated slide presentations and comprehensive student manuals will make experienced instructors who may be a little weak with their CNC skills look like CNC experts!

Second, instructors working for companies having their own in-plant training will also find this curriculum easy to implement. Few companies have the resources (or the desire) to completely develop this kind of program from scratch. Additionally, success for an industrial training program is *not* a simple a matter of the student getting a good grade. When finished, students *must* know how to safely program and/or operate the machine tools. Training failure will result in scrapped parts, crashed machines, and possibly even injured operators. The real-world exercises in the student workbook will prove the student's knowledge of presented materials each step of the way. When the student successfully completes this course, you can rest assured that they comprehend the subject matter.

Third, training consultants (or instructors working for technical schools) that provide custom training for manufacturing companies will find that this course curriculum makes it easy to teach CNC to their clients. Utilizing standard (and highly portable) computer equipment, the comprehensive slide show can be done on-the-road, meaning instruction can take place anywhere.

Five factors that contribute to learning
While experienced instructors may find this information somewhat basic, we wish to ensure that beginning instructors understand the importance of setting up a good learning environment. Of course, the better the learning environment, the better the training program will be, and the faster, and easier it will be for students to learn. This course curriculum is well suited to facilitating the learning environment in all five ways.

Motivation is the most important factor in any learning environment. First and foremost, students must be highly motivated to learn. Motivation will be the driving force that makes students *stick with it* even when they are having trouble understanding information being presented during training. Indeed, *any* problem with learning can be overcome if the student's motivation is high enough. But just as important, the instructor must also be highly motivated to teach. You must have a burning desire to relate information during training in a way students can understand. When student are having problems, the instructor must be motivated enough to spend the extra time it takes to ensure that the student eventually understands presented material. This can be very challenging since students' aptitude levels vary. This course curriculum inspires motivation on both counts. The colorful and illustrative slide show, the tutorial nature of the manual, the pertinent practice exercises, and the highly structured key concepts

approach will capture and hold your students' attention, and make it easy for them to *stay* motivated. For the same reasons, instructors should find this course almost *fun to teach* - and it is easy to stay motivated with tasks you enjoy.

Aptitude will determine how quickly and easily learning will occur. The aptitude of the instructor for making interesting and attention-holding presentations, giving pertinent analogies, preparing illustrative visuals, designing realistic practice exercises, and in general, keeping the students interested level high will contribute to determining how quickly and easily students catch on to new material. *Instructors with high aptitude make it easy for students to learn.* In similar fashion, the students' aptitude for learning manufacturing related functions and specifically CNC also contribute to how quickly and easily learning takes place. *Students with high aptitude make it easy for the instructor to teach.* This course curriculum pays particular attention to trouble spots. From our own teaching experience, we know where students with minimal aptitude tend to have the most problems and make ample analogies to keep even the most complex topics of CNC as simple as possible to understand.

Presentation is the heart of training. The better the instructor prepares and delivers the presentation, the easier it will be for students to learn. Presentation can consist of many things, including the instructor's lectures, demonstrations, simulations, overhead and projector slides, videos, and anything else that helps to convey an idea. This is the strong point of our curriculums. You will find it very easy to get your ideas across with but a small amount of preparation before delivering each lesson. While you still have to talk, the slide shows, fly-in text boxes, and instructor notes will make sure you know what to say!

Repetition reinforces a student's understanding of learned information. Even students with extremely high aptitude will find it difficult to learn from presentations made only once. *All* training sessions should begin with a review of recent presentations. Depending upon the frequency and duration of each training session, entire sessions should, at times, should be devoted to reviewing what students should already know.
Reviews also help the instructor limit how much complex subject matter is presented during each session. Knowing that certain information will be reviewed, the instructor can avoid getting too deep into complicated topics during the first time the information is presented. Only after students have a firm grasp of basics concepts will the instructor dive deeper and introduce more complicated variations. While you may wish to do more, this curriculum includes a "Review" presentation. For each key concept, we first name the key concept and list the related lessons. We then provide a review slide for each lesson in outline format that lets you reiterate key points about each major topic in the lesson. These are simply bullet points, so you'll have to describe topics being reviewed without the benefit of graphics and animations. If students are having trouble during reviews, it should be taken as a signal that they don't truly understand the subject matter. In this case, you can easily go back to the lesson being reviewed, call up the *presentation links* slide, and click on the link that brings you to the topic in question.

Practice with reinforcement acts as the gauge to judge the success of training. Well designed practice exercises should be realistic, forcing the student to do things in the same way they must when training is completed. Reinforcement must come as the result of the students' practice. If the student demonstrates a firm understanding of the presented information, reinforcement should praise the success. On the other hand, if practice exposes a student's lack of understanding, reinforcement should come in the form of repeated presentations (review) and more practice, ensuring that the student eventually catches on. This course curriculum includes a comprehensive set of practice exercises as well as several quizzes in the student manual, to confirm understanding each step of the way.

The Key Concepts approach
Most training experts agree that students learn best when they have *a light at the end of the tunnel*. The more complex the subject matter, the more important it is to tell (better yet, show) the student early in the training program just what they must master to complete the course. One important benefit of our proven key concepts approach you can do just that. With our curriculum, there are four key concepts to CNC setup and operation. Early on in the course you can truthfully say *"If you can understand just four basic ideas, you are well on your way to becoming proficient as a CNC setup person or operator."* This gives your students a light at the end of the tunnel and makes learning CNC seem as easy as possible. Also, as you go through the course, students will know where they stand each step along the way.

Another benefit of our key concepts approach is that it lets you work at a very broad level. There are many different CNC machine and control manufacturers making this kind of equipment. Though there are many commonalties in how basic machine utilization is done among current CNC machines, no two machines will be handled exactly the same. For this reason, students will likely have to adapt to their own machines once they complete your course (especially if you teach in a technical school). In each key concept you will be stressing the broad and general reasoning behind how CNC functions, showing students *why* they must do things as importantly as *how* they do them. This reasoning can be applied to any form of CNC machine tool. Once the student understands this reasoning, *then* you will show specific techniques needed to apply the key concept to one very popular CNC control.

By the way, all of the specific examples we give in the student manual and during the slide shows are in the format for a Fanuc control. We chose Fanuc as our sample control since it is the most popular control in the industry, and since several control manufacturers (Mitusbishi Meldas, Mazatrol, Yasnac, HAAS, Fadal, and others) claim to be *Fanuc-compatible*. However, you must be prepared for the possibility that your students will be working with a control made by a manufacturer that is not Fanuc-compatible. Again, rest assured that the ten key concepts apply to every current CNC control on the market. Only the specific techniques needed to apply each key concept must change.

Yet another benefit (from an instructor's standpoint) of the key concepts approach is that the course can end at any time once students understand the key concepts. One major problem in any training program is the limited time available for training. While manufacturing companies may have the luxury of extending a course if students are slow in picking up the material, technical schools commonly work on a very rigid time frame. If students are slow in picking up the material and getting bogged down with the details of CNC machine utilization (asking many time consuming questions), the course may end before you get through all the material. While this presents a problem even with the key concepts approach, the effects of the problem can be minimized. As long as students understand the reasoning behind each key concept, it will be just a matter of time until they figure out the details. Given our extensive and highly tutorial course manual, any student with motivation will eventually figure out the details (possibly even *after* the course ends).

One last benefit we'll mention for the key concepts approach is that it makes reviewing the material easy. Since there are only six key concepts, they are easy to remember. You can simply restate each key concept (using the "review" slide presentation) and ask pertinent questions to confirm your students retention of material previously presented.

Lesson structure
These six key concepts are divided further concise lessons. For the machining center programming curriculum, there are sixteen lessons. Lessons do vary in length (from about 10 minutes of presentation to about an hour), and you can find an approximate time of presentation at the end of the Lesson Plans manual.

Here are the Key Concepts and lessons. Key Concepts are shown in bold. Lessons are indented.

1: Know your machine from a programmer's viewpoint
   1: Machine configurations
   3: Visualizing program execution
   3: Program zero and the rectangular coordinate system
   4: Introduction to programming words
2: You must prepare to write programs
   5: Preparation steps for programming
3: Understand the motion types
   6: Programming the three most basic motion types
4: Know the compensation types
   7: Introduction to compensation
   8: Tool length compensation
   9: Cutter radius compensation
   10: Fixture offsets
5: You must provide structure to your CNC programs
   11: Introduction to program structure
   12: Four types of program format
6: Special features that help with programming
   13: Hole-machining canned cycles
   14: Working with subprograms
   15: Other special programming features
   16: Programming rotary devices

Student's Course Materials:
All student course materials are copyrighted and must be purchased from CNC Concepts, Inc. Current pricing can be found on our website (www.cncci.com).

Student Manual - These highly tutorial manuals precisely follows the slide show presentations you will be making. They are very detailed and will provide the student with an excellent way to review information you present, during the course and long after the course is finished. There is one student manual for each curriculum. Exercises are included right in the course manual. Of course, you will also want to have this manual to use as a master as you present the course.

Special note about answers to the exercises
Since the student manual is a self-study manual, answers to the exercises are provided at the end of the manual (on page 333). If you do not want your students to have access to these answers as they go through your class (you want to grade them), we suggest that you ask your students to pull these pages from the manual and give them to you during your first session. Ask them to be careful when removing the answer pages (remove one or two pages at a time) so as not to damage the binding of the manual.

Instructor's Course Materials:
CD-rom disk including all instructor materials – The disks include the following instructor materials:

PowerPoint Slide shows:
Each lesson includes a slide show to provide the visuals you need to teach. They are all computer generated graphics and animations that really help you make your points during the class.

Lesson topics links slide – Every lesson begins with this special slide. The lesson topics slide provides links to every main topic for the lesson. Every slide in the lesson will have a "Lesson topics" link at the bottom that returns you to this slide. Again, the lesson topics slide lists the main topic to be covered by

the lesson. When you click on a topic, the presentation will skip to the slide that begins that topic. This can help in three ways. First, it provides a very interactive way to get to topics of interest (you will be clicking on topics during your presentation) as you teach. Second, you will be able to easily skip topics that you don't want to cover. Just remember that once you click on a topic, the presentation will simply continue from the slide that begins the topic. If you get into another topic (that you don't want to cover), simply click the Lesson topics link in the lower left corner. And third, this special slide should really help when reviewing. It makes an excellent list of bullet points that you can review in a nice logical order. And of course, if students are having problems, you can easily click on the trouble-causing topic and re-present the information related to the topic. The lesson topics slide also makes a great "presentation plan". You'll be able to quickly introduce the main topics as you start each lesson and summarize them when your finihed with the lesson.

Fly in text boxes – In addition to graphics and animations throughout the slide shows, we also include *fly-in text boxes* on most slides to help you understand where the presentation is going. In some cases, the *fly-in text box* will remind you to make some important point about the topic at hand. In other cases, they are included at a change of pace in the presentation, and provide the next few words of your presentation. In all cases, they will help you stay on track and minimize the amount of preparation you must do to get ready to teach.

Review slide presentation – A link from the startup page, this special presentation is intended to help you review material throughout the course. As stated, at least ten percent of each session should be spent reviewing information that you previously presented. This presentation is made up of at least two slides for each key concept. The first slide for each key concept shows the key concept name and the lesson/s related to the key concept. This is followed by a list of bullet points for each lesson in the key concept. Each list of bullet points is identical to the presentation links slide for the lesson. So if students are having problems during the review, you can easily call up the related lesson, go to the presentation links slide, and click on the topic of interest. Since this collection of slides makes for a very good outline for the entire course, we include a printout of the Review slide show later in this Instructor Manual.

All presentation files are easily started from the startup page, named a_startup.ppt on the CD-rom disc.

PowerPoint Viewer software:
All slideshows are prepared in PowerPoint. In order to view/display/print the slide shows, you must either have the actual PowerPoint software (PowerPoint 95 or above) or the PowerPoint Viewer. We provide the PowerPoint Viewer on disk one. Since the PowerPoint Viewer does not let you modify slides, and since it requires that slide shows always start from the first slide (not very convenient when you're trying to get to a slide in the middle of a lengthy presentation), we urge you to get the actual PowerPoint software. It can be purchased separately, and it comes with most versions of Microsoft Office.

What you still need:
In order to show the PowerPoint slide presentations to a group of people, you need the following items.

A computer with Windows - Just about any current model computer will work. If using a desktop computer, you can easily watch the monitor of the computer (facing your audience) to see the slide show while slides are displayed behind you by the projection system. Since the left mouse button advances the slides, you even have a remote slide advance button. If portability is an issue, keep in mind that all current notebooks have ample power to run the presentation software. However, be careful in your selection. Many notebooks do not allow you to send data out through the video port *and* see the slide show on the LCD screen of the notebook at the same time. Without this ability, you may have to turn around to see your slides, which can be distracting to your audience.

A way of displaying the screen show - You have several alternatives in this regard. All involve using a device that takes data from the video port of your personal computer. First, many schools already have a projection system that can display information from a personal computer. Basically, anything that can be shown on the computer screen can be displayed through the projection system. Second, you can use a device that sits on top of an overhead projector to display your screen shows. In essence, this device makes a transparency of what ever is on the display screen of the computer (we don't like this kind of device because of its brightness – it can really hurt your eyes if you try to use it as a normal overhead projector). Third, and especially if price is a concern, you can use a simple scan converter (about $200.00 - $300.00) and display your screen show on any television that has a *video in* connector (as most do). If you must use the RF connector of the television (where an antenna plugs in), an RF converter must be purchased. Since there are so many alternatives for displaying your slide shows, we welcome phone calls (847) 639-8847 if you have questions about your alternatives.

Putting It All Together

Getting Ready To Teach
As stated earlier, though these course curriculums dramatically reduce the amount of preparation you must do, they do not eliminate it completely. And as any experienced instructor will agree, the key to successful presentations is in becoming comfortable with the material you present. And the only way to get comfortable is through adequate preparation.

Before your first course:
Skim the entire curriculum - Though you do not have to be perfectly comfortable with every detail of the curriculum to begin teaching, you will at least need to understand where the course is going. You can use the course outline, lesson plans, instructor notes, and student manual to gain an appreciation for the six key concepts and the lesson structure being used.

Before beginning each key concept:
Get comfortable will all lessons in the key concept - While some key concepts have but one lesson, most have more. Be sure you feel comfortable with all major points you need to make before you begin teaching. Again, use the course outline, instructor notes, and student manual to increase your comfort level with the entire concept.

Before you deliver a lesson:
Get ready to teach! - Study the lesson plan and go through the in order to gain an understanding of key points that must be delivered during your presentation.

Practice! - Especially before your first few lessons, get comfortable with your equipment and the material you present by practicing your presentation. In addition to getting you ready to deliver each lesson, this should give you a rough idea of how long it will take to deliver each lesson at your own pace.

During your presentation of each lesson:
Tell them what you're going to tell them - The Lesson topics slide (first slide of every lesson will help you prepare your students for what they will be learning. While you don't have to dwell on this slide too long, it will let them see what is included in the lesson.

Tell them - Go though the lesson, using your slide show as a guide. Be sure to point out the page numbers and sections in the student manual where the information is also included for their own independent study. Be sure everyone is catching on. Encourage participation, questions, and comments. While you should find adequate analogies in the slide show to stress the most complex topics, you must be prepared to handle special questions and concerns. Have a blackboard or overhead available for making special points.

Tell them what you told them – Get back to the Lesson topics page and review the key points of the lesson. Again, encourage questions and participation. The idea is to confirm that students truly understand presented material.

After you finish each lesson:
Assign and check practice exercises - The students' responses to the practice exercises makes an excellent way to gauge your students' understanding of the subject matter. If you find that students are not doing well, it should be taken as a signal that you must review key information. If students are doing well, be sure to praise them.

As you get deeper into the course:

Review often - No student will retain every word of every presentation you make during a course as lengthy as these. On average, you should spend about 10% to 20% of your session time in review, depending upon how well your students are doing. The more problems they are having, the more time you should spend on review. One excellent way to review is to question students from the previous exercises to confirm their retention. The review slide presentation can be activated from the startup page, and will help you review all key topics.

Let students know where they stand - Be sure everyone knows how they are doing as they progress through the course. Assign special exercises and labs for those students having the most problems. Push those students doing well to go further.

Answers to Exercises

These answers are in the student manual!

Since the student manual is intended to be a self-contained self-study manual, we include answers to all the exercises at the end of the manual (starting on page 333). This is fine if you would like your students to check their own work. But if you would rather not let them have access to answers (you want to check and grade their work), you must have them remove these pages from the student manual early on in the class. You can do so during your first session – before they've started reading the student manual – while you are introducing the class. Do ask them to be careful when removing the pages (remove only one or two pages at a time) to keep from damaging the binding of the manual.

This is the message placed in the student manual just prior to the exercise answers: If you are attending a class that uses this text, your instructor may ask you not to look at these answers prior to submitting your work. (Each exercise can be removed from this text without removing any content.) On the other hand, if you are reading this text on your own, you can use these answers to check your work.

Exercise 1: Machine configurations

1) true

2) true

3) false

4) a. The X axis is the left/right direction (as viewed from the front of the machine)
b. The Y axis is the in/out direction (as viewed from the front of the machine)
c. The Z axis is the up/down direction (as viewed from the front of the machine)
d. If the machine has a rotary table on the table, it could be called the A, B, or C axis, depending upon orientation

5) a. As viewed from the spindle, left/right motion is the X axis.
b. As viewed from the spindle, up/down motion is the Y axis.
c. As viewed from the spindle, in/out motion is the Z axis.
d. If the machine has a rotary axis mounted in the table, it is called the B axis.

6) a. As viewed from the front, tool motion toward the right
b. Tool motion away from the operator
c. Tool motion in an upward direction

7) Answers include speed, direction and possibly range

8) d. in feed per minute

9) c. S word

10) b. F word

Extension question answers:

11) All roughing should be done before any finishing is done.

12) When the machine is rigid enough to withstand the tendency for the tool to pull into the cut. Most machinists agree that climb milling will render a better finish than conventional milling, but should not be done with knee style machines.

13) Because with most machines, the tool does not actually move along with one or more of the axes. To try to consider table motion plus versus minus can be quite confusing.

Exercise 2: Visualizing program execution

1) The person performing the operation manually has everything in front of them and available. The person preparing the CNC program must do so with only print, pen, and paper.

2) true

3) true

4) true

5) false

6) true

7) false

8) a. X0.0007

9) A modal word will set a state the remains in effect until changed or canceled.

10) An initialized word will be automatically instated when the machine power is turned on.

Extension question answers:

11) In order to write CNC programs, a programmer must be able to visualize machining operations as the program is written. Without this ability, it will be impossible to write the program.

12) First the tool will move into the hole center position, just above the hole to tap. With the spindle running cw, the tool is fed into the hole. At the hole bottom, the spindle is reversed and the tool fed out of the hole. Finally the spindle is reversed again to its original direction.

13) The tool is brought to the pocket center and just above the work surface. Then the end mill is plunged into the pocket to the pocket depth. Forming a circular motion, the end mill will then sweep the inside of the hole, forming the circular pocket.

14) This set of answers will vary with company. The instructor of this course can compare the students' answers with the operations the company performs.

Exercise 3: Program zero and the rectangular coordinate system

1) true.

2) true

3) b. 0.0001 in

4) c. 0.0010 mm

5) Since the resolution of the metric mode is less than half that of the inch mode, positions can be specified more precisely in metric mode.

6) false

7) true

8) c. It should be placed at the location from which dimensions are taken on the print.

9) true

10) d. absolute mode

11) a. incremental mode

12) The absolute mode is better. Mistakes are not compounded, values going into the program make sense, and programs written in the absolute mode are easy to follow.

13) Absolute mode: G90
Incremental mode: G91

14) True

15)

#	X	Y
1	0.375	0.375
2	1.0	0.375
3	3.625	0.375
4	3.625	2.625
5	3.0	2.625
6	0.375	2.625
7	2.0	1.5

Extension question answers:

16) Since you will be working primarily in the absolute mode, it would be best if the machine initializes to this state. This could help if the programmer forgets to instate the absolute mode at the beginning of the program.

17) The machine would move in a very unusual manner, since motions have been specified from program zero but the machine is not using the program zero point for the reference position for movements. This kind of problem should be quite easy to find during the program's verification.

Exercise 4: Introduction to programming words

1) Choose from A, B, C, X, Y, Z, I, J, K, F, Q, and R

2) false

3) false

4) false

5) true

6) a. M03

7) c. M05

8) b. M08

9) d. M09

10) b. G20

11) c. G90

12) c. M30

13) Designates the radius of a circular command: **R**
Specifies a coordinate along the Z axis: **Z**
Specifies the program number being called by a sub-programming command: **P**
This is word for a preparatory function: **G**
This is the word for a miscellaneous function: **M**
Specifies a spindle speed in rpm: **S**
Specifies feedrate: **F**
Designates program number: **O** (letter, not zero)
Designates cutter radius compensation offset number: **D**
Designates tool length compensation offset number: **H**
Specifies coordinate along the X axis: **X**

One of these three words is used if your machine has a true rotary axis: **A, B, or C**

14)

#	X	Y	Z
1	0.575	0.575	0
2	3.425	1.25	0
3	3.425	2.425	0
4	0.575	2.425	0
5	2.5	0.5	-0.125
6	3.0	0.5	-0.125
7	3.5	0.5	-0.125
8	-0.6	0.2	-0.125
9	2.0	0.2	-0.125
10	2.0	0.313	-0.125
11	2.562	0.875	-0.125
12	3.55	0.875	-0.125
13	3.8	1.125	-0.125
14	3.8	2.55	-0.125
15	3.55	2.8	-0.125
16	0.45	2.8	-0.125
17	0.2	2.55	-0.125
18	0.2	0.45	-0.125
19	0.45	0.2	-0.125

Extension question answers:

15) We cannot provide answers to this question.

16) With most controls, only the last three (right-most) G codes will be executed. All others will be ignored.

17) The latter of the two conflicting G codes will be executed.

Exercise 5: Preparation steps for programming

1) true

2) true

3) true

4) b. will cause the control to go into an alarm state and give a specific message documenting what is wrong.

5) true

6) true

7) false

8) 1) It allows easy checking of the process 2) It allows you to separate machining skills from programming skills - 3) Provides good future documentation 4) Mistakes can be corrected before machine time is wasted

9) b. 611 rpm

10) c. 4.8 ipm

11) 1) With a coordinate sheet 2) By writing coordinates right on the blueprint

12) 1) To confirm availability 2) To confirm they will reach surfaces without interference 3) To ensure they are long enough to reach work surface

13) false

14)

#	X	Y	Z
1	2.0	0.5	0
2	2.7071	0.7929	0
3	3.0	1.5	0
4	2.7071	2.2071	0
5	2.0	2.5	0
6	1.2929	1.2071	0
7	1.0	1.5	0
8	1.2929	0.7929	0
9	-0.6	0.2	-0.25
10	2.55	0.2	-0.25
11	3.8	0.6549	-0.25
12	3.8	2.55	-0.25
13	3.55	2.8	-0.25
14	1.45	2.8	-0.25
15	0.2	2.345	-0.25
16	0.2	1.45	-0.25
17	0.45	0.2	-0.25

Extension question answers:

15) Answers can include the process engineer or the CNC programmer, depending upon the company.

16) Depending upon the CAM system, it is quite likely that it can do the math and possibly determine cutting conditions. Note however, it will still be up to the programmer to determine the process, check the tools, and document the setup.

Exercise 6: Types of motion commands

1) true

2) false

3) false

4) 1) They are all modal. 2) The end point of the motion is programmed in each motion command. 3) All motions are affected by absolute/incremental modes (G90/G91). 4) Only the moving axes need to be included in a motion command. 5) Leading zeros can be suppressed (G00 and G0 mean the same thing).

5) true

6) 1) The cutter's centerline path 2) The work surface path

7) c. cutter radius compensation

8) G00 - rapid motion, G01 - straight line motion, G02 - cw circular motion, G03 - ccw circular motion

9) b. to position the tool as quickly as possible to and from its cutting positions.

10) d. To machine at a specified feedrate along a perfectly straight line.

11) c. to form circular contouring motions on the workpiece.

12) true

13) false

14) true

15) b. 0.100"

16) c. 0.250"

17) b. the vertical and or horizontal center line of an arc.

18) true

19) true

20) true

21) false

22) See programming activity answers.

Extension question answers:

23) Answers cannot be provided

24) Since the control will revert to the current motion mode, a circular motion will not be made. Since the command likely includes an R, I and/or J word, most controls will generate a program storage alarm.

25) This is a tough question to answer. With most controls, the R word is more "forgiving" that I, J, and K. a mistake is made when using the R word, it is likely that the control will do "something" (not generate an alarm). However, the circular

motion will not be as expected. If a mistake is made – even a tiny one – when directional vectors are used, an alarm will sound.

26) No answer program is supplied for this exercise.

27) Answer: G03   Though this motion appears to be a cw motion (G02), this drawing is not showing the motion as the machine will see it. We must view this motion from the PLUS side of the Y axis (we're currently looking at this motion from the Y minus side). This motion is actually ccw, and must be specified by G03. Additionally, this motion would require that the XZ plane (G18) be specified prior to the circular motion command.

Exercise 7: Introduction To Compensation

1) true

2) true

3) fixture offsets, tool length compensation, cutter radius compensation

4) false

5) true

6) false

7) c. at least three, and more if the machine has a rotary table

8) b. setting the offset prior to machining in such a way that excess stock is left on a surface by a cutting tool.

9) a. H

10) b. D

11) true

12) false

13) See programming activity answers.

Extension question answers:

14) You must talk to people currently using the machine to get this question answered.

15) You must talk to people currently using the machine to get this question answered.

Exercise 8: Tool length compensation

1) true

2) true

3) false

4) false

5) b. it is the same as the tool station number

6) d. G43

7) b. H

8) b. during the first approach movement for each tool

9) true

10) false

11) See programming activity answers.

Extension question answers:

12) You must check with an experienced person in your company or school to get this answer.

13) You must check with an experienced person in your company or school to get this answer.

Exercise 9: Cutter radius Compensation

1) true

2) c. an end mill

3) false

4) 1. To make calculation coordinates easier
2. To allow a range of cutter sizes
3. To allow sizing of the contour/surfaces being milled
4. To allow roughing with finish work surface coordinates

5) One and four.

6) false

7) b. 0.4375

8) a. make it 0.001 smaller

9) a. increase the offset value slightly

10) G40 - Cutter radius compensation cancel
G41 - Cutter radius compensation left (climb mill)
G42 - Cutter radius compensation right (conventional mill)b. the nose of the spindle would move to the programmed Z position (causing a crash)

11) c. D word

12) 1. Instate
2. Machine the contour/surface
3. Cancel

13) 1. G42
2. G41
3. G41
4. G42

14) Select a constant number greater than the machine's tool capacity and add it to the station number. Example: If the machine holds thirty-five tools, the constant number could be forty. Add forty to the tool station number.

15) It must be at least the cutter radius away from the first surface and selected so that a right angle is formed when the cutter contacts the first surface and begins milling.

16) d. all of the above

17) d. all of the above

18) false

19) The cutter's center line path is programmed and the offset value is the difference between the planned cutter size and the size currently being used.

20) a. zero

21) See programming activity answers

Extension question answers:

22) Add the amount of stock you wish to leave for finishing to the roughing cutter's offset value. Example, if you wish to leave 0.020 in for finishing, make the cutter radius compensation offset 0.020 larger than its actual size.

23) As you begin using cutter radius compensation, you will find that most problems you have will be related to the offset size. The larger the offset, the more potential problems. Since with the CAM system person's method the offset value is always close t zero, fewer cutter compensation problems will arise.

Exercise 10: Fixture offsets

1) a. you don't have to know the distances between the workpieces as you write your program.

2) true

3) true

4) true

5) b. make the common fixture offset value zero in each axis and input the distance from the machine's zero return position to each program zero point in each fixture offset.

6) c. make the common fixture offset values equal to the distance from the machine's zero return position to a logical place in the setup in each axis. Each fixture offset entry is then taken from this position.

7) false

8) d. you can save time during setups

9) true

10) c. G56

11) c. G10

12) G90 G10 L2 P5 X1.5 Y4.0 Z-0.75

13) false

14) Because the Z value of each fixture offset can be determined in a consistent, logical manner.

15) a. to shift the point of reference to a more logical point
b. to overcome problems with differences in spindle gap when tool lengths are measured off line for multiple machines
c. to overcome axis alignment problems after a crash
d. to enhance safety during program verification
e. to handle differences in pallet changers

16) See programming activity answers.

Extension question answers:

17) No answer can be provided here.

18) No answer can be provided here.

19) No answer can be provided here.

20) No answer can be provided here.

21) a. the fixture offset Z value is incorrect
b. the tool length compensation value is incorrect
c. the programmed Z position is incorrect

22) a. the tool length compensation value is incorrect
b. the programmed Z value is incorrect

Exercise 11: Introduction to program structure

1) true

2) false

3) false

4) true

5) Familiarization
Consistency
To be able to re-run tools

6) To be able to re-run tools

7) true

8) false

9) c. restart the program from the beginning of tool four.

10) b. you must include all commands necessary to get the machine running at the beginning of each tool.

11) false

12) true

13) Place the first tool in the spindle at the beginning of the program. You do not want the control to assume it is there. If it does (no tool change at the beginning of the program), the wrong tool could be in the spindle, possibly causing a crash..

14) See programming activity answers.

Extension question answers:

15) We cannot provide answers.

16) We cannot provide answers.

17) We cannot provide answers.

18) We cannot provide answers.

Exercise 12: Four kinds of format

1) Program start-up format
Tool ending format
Tool start-up format
Program ending format

2) true

3) true

4) YES! It is true that this program would work just fine without the Y words in lines N065 and N115 as long as the program runs from the beginning. But note that

this program ends with the machine at the zero return position. Without these Y words, you cannot re-run tools 2 & 3.

5) true

6) a. move nothing in Z, then go to the Z axis zero return position.

7) d. move nothing in X and Y, then go the XY zero return position.

8) c. move to the Z axis program zero point (crash), then go to the Z axis zero return position.

9) b. move plus three inches in Z, then go to the Z axis zero return position.

10) 1) to make it easier to pick up in the middle of the program.
2) to ensure that the correct tool is selected, even if the operator manually rotates the tool magazine.

11) Coolant off. It is require to keep the work area from being soaked during each tool change. Also, you don't want coolant to have a chance to get into the spindle during tool changes.

12) M19 in the return command, the spindle will already be oriented when the tool change command (M06) is given.

13) M01 is an optional stop. When used at the end of each tool, it gives the operator a chance to stop the cycle at the end of each tool. This gives them the ability to check what each tool has done before proceeding.

14) d. M30

15) d. all of the above

16) Line N050, since tool number two is not currently in the spindle.

17) Line N055 since tool number two is already in the spindle.

18) See programming activity answers.

Extension question answers:

19) We cannot provide answers to this question.

20) The machine would attempt to run you inch program in the metric mode. Your inch values would be taken in millimeters 1.00 in, for example, would be taken as 1.00 mm. At least no major mishap would occur. The machine would run a tiny workpiece very close to the zero return position. However, if you attempt to run an metric program in the inch mode, you could be in for real problems. Values of 100 mm, for example, would be taken as 100 inches!

21) We cannot provide an answer to this question

22) 1) The spindle could be started in with the first motion for each tool
2) Approach movements could be made in all axes.

Exercise 13: Hole machining canned cycles

1) true

2) c. G80

3) c. the machine will continue machining holes in the next CNC commands.

4) a. the tool will rapid to XY position
b. the tool will rapid to the R plane
c. the hole will be machined
d. the tool will retract from the hole

5) standard drilling cycle: G80
deep hole peck drilling cycle: G83
chip breaking peck drilling cycle: G73
counter boring cycle: G82
cancel cycle: G80
right hand tapping cycle: G84
left hand tapping cycle: G74
standard boring cycle: G86

fine boring cycle that leaves no witness mark: G76

6) G81 - because no machining occurs on the way out of the hole and time can be save as compared to using the G85 canned cycle.

7) sequence number: N
pause time in counter boring cycles: P
feedrate for hole machining: F
rapid plane: R
bottom position of hole: Z
number of holes the cycle will machine: L
peck depth for peck drilling cycles: Q
X axis move over amount in fine boring cycle: I

8) false

9) G98 - initial plane, which is the last programmed Z surface prior to the canned cycle.
G99 - R plane, which is the Z position specified by R in the canned cycle itself

10) G98 - initial plane

11) Because the machine cannot perfectly synchronize the spindle reversal with retract movement. With a small R word the tap could still be in the hole at the cycle's completion.

12) Multiply spindle rpm times the thread pitch, which in inch, ,is 1 divided by the number of threads per inch.

13) The relationship of drill diameter to hole depth. When the hole depth is over about three times the drill diameter, you should use the deep hole drilling cycle.

14) false

15) When you have a series (more than just a few) of evenly spaced holes.

16) See programming activity answers.

Extension question answers:

17) We cannot supply answers to this question.

18) 1) You could cancel the cycle, move around the obstruction, and reinstate the cycle once the obstruction is cleared.
2) Though it was not covered in the lesson, you could place an "L0" (K0 for 0M and 3M)n each XY obstruction clearing command. This will keep you from having to cancel the cycle while clearing the obstruction.

19) Use a short rigid drill to drill a pilot hole (the same as the hole size). This will help support the very long drill as it machines the hole. You can even set the R plane for the long drill well into the hole to minimize air cutting time and use G98 to retract all the way out between holes.

Exercise 14: Working with subprograms

1) true
2) a. repeating identical machining operations
   b. control programs
   c. utilities
3) repeating identical machining operations
4) Increased ease of program verification. If the first execution is correct, they all will be.
5) true
6) b. M98
7) a. M99
8) d. P
9) c. L
10) false
11) d. four
12) Incrementally, move over the distance between the operations.

13) a. part families
    b. user defined canned cycles
    c. utilities
    d. complex shapes and motions
    e. driving accessory devices (like probes)
14) See programming activity answers.

Extension question answers:

15) We cannot provide answers to this question.

16) The control will generate an alarm.

17) a. When running multiple identical workpieces in the same setup.
    b. When machining multiple identical pockets.
    c. When milling multiple identical threaded holes.
    d. When machining multiple dovetails. (many more answers could be correct)

Exercise 15: Other special programming features

1) c. the slash code (/)
2) false

3) true
4) false

5) a. to control coolant during program verification with machines that do not have a manual switch
b. to gain another optional stop
c. with trial machining

6) false

7) b. M99

8) a. with optional block skip, to exit a program
b. to change machining order at the machine

9) true

10) b. You multiply the percentage of a full circle being machined in the command times the pitch of the thread.

11) False

12) d. G04

13) false

14) N005 G04 X1.0
(Like X, U and P can also be used to specify dwell time.)

15) b. when you want the control to force sharp corners during a contour

16) a. G64

17) b. G10

18) a. G17

19) true

20) true

21) G20 - inch mode
G21 - metric mode

22) metric mode

23) G50 - cancel scaling
G51- instate scaling

24) true

25) Climb milling will become conventional milling and vise versa.

26) manually and through program commands

27) d. G53

28) When finish boring, and when hole positions are critical

29) See programming activity answers.

Extension question answers:

30) Answers could include (among others) Other types of trial machining, to skip a tapped hole if the tap is broken in the hole, and to machine two very similar workpieces with one program.

31) Answers could include (among others) to allow extra time for coolant to flow properly, to force sharp corners, and to allow time for M code functions to be competed.

32) We cannot provide answers to this question.

33) We cannot provide answers to this question.

Exercise 16: Programming rotary devices

1) true

2) true

3) false

4) c. is used to quickly rotate to a surface in need of machining.

5) a. can rotate the workpiece and machine it at the same time.

6) a. 90 & 45 degree indexers: One M code commands rotation
b. 5 Degree indexers: A series of M codes commands each rotation (M71: 5 deg., M72: 15 deg, M73: 30 deg, etc.)
c. 1 degree indexers: A letter address is commonly used to command angle of rotation

7) Answers could include: a . zero return position b. designation of program zero, c. absolute vs incremental modes, d. rapid vs straight line motion, and d. usage with canned cycles

8) b. in degrees per minute

9) b. B

10) false

11) b. incremental mode

12) a. When rotating from a small angle to a large one.

13) false

14) a. assigning one central program zero point
   b. assigning one program zero point per side to be machined

15) assigning one program zero point per side to be machined

16) false

17) See programming activity answers.

Extension question answers:

18) We cannot provide answers to this question.

19) We cannot provide answers to this question.

Answers to programming activities

Again, if you are attending a class that uses this text, your instructor may ask you not to look at these answers prior to submitting your work. (Each activity can be removed from this text without removing any content.) On the other hand, if you are reading this text on your own, you can use these answers to check your work.

Programming activity 1: Test comprehension of motion types

#	X	Y	Z
1	0.750	0.750	0.1/-0.730
2	4.250	0.750	0.1/-0.730
3	4.250	3.250	0.1/-0.730
4	0.750	3.250	0.1/-0.730
5	-0.475	-0.250	-0.25
6	4.500	-0.250	-0.25
7	5.250	0.500	-0.25
8	5.250	3.500	-0.25
9	4.500	4.250	-0.25
10	0.500	4.250	-0.25
11	-0.250	3.500	-0.25
12	-0.250	0.500	-0.25
13	0.500	-0.250	-0.25
14	2.000	2.000	.1/-.763/-.675
15	2.500	2.000	.1/-.763/-.675
16	3.000	2.000	.1/-.763/-.675

O0001 (Program number)

(Center Drill)
N005 T01 M06
N010 G54 G90 S2000 M03 T02
N015 G00 X0.75 Y0.75 (pt 1)
N020 G43 H01 Z0.1 M08
N025 G01 Z-0.200 F5.0
N030 G00 Z0.1
N035 X4.25 (pt 2)
N040 G01 Z-0.2
N045 G00 Z0.1
N050 Y3.25 (pt 3)
N055 G01 Z-0.2
N060 G00 Z0.1
N075 X0.75 (pt 4)
N080 G01 Z-0.2

N085 G00 Z0.1
N090 X2.0 Y2.0 (pt 14)
N095 G01 Z-0.2
N100 G00 Z0.1
N105 X2.5 (pt 15)
N110 G01 Z-0.2
N115 G00 Z0.1
N120 X3.0 (pt 16)
N125 G01 Z-0.2
N130 G00 Z0.1 M09
N135 G91 G28 Z0 M19
N140 M01

(1/4 drill)
N145 T02 M06
N150 G54 G90 S1200 M03 T03
N155 G00 X0.75 Y0.75 (pt 1)
N160 G43 H02 Z0.1 M08
N165 G01 Z-0.730 F4.8
N170 G00 Z0.1
N175 X4.25 (pt 2)
N180 G01 Z-0.73
N185 G00 Z0.1
N190 Y3.25 (pt 3)
N195 G01 Z-0.73
N200 G00 Z0.1
N205 X0.75 (pt 4)
N210 G01 Z-0.73
N215 G00 Z0.1 M09
N220 G91 G28 Z0 M19
N225 M01

(23/64 drill)
N230 T03 M06

N235 G54 G90 S800 M03 T04
N240 G00 X2.0 Y2.0 (pt 14)
N245 G43 H03 Z0.1 M08
N250 G01 Z-0.763 F4.0
N255 G00 Z0.1
N260 X2.5 (pt 15)
N265 G01 Z-0.763
N270 G00 Z0.1
N275 X3.0 (pt 16)
N280 G01 Z-0.763
N285 G00 Z0.1 M09
N290 G91 G28 Z0 M19
N295 M01

(0.3750 reamer)
N300 T04 M06
N305 G54 G90 S600 M03 T05
N310 G00 X2.0 Y2.0 (pt 14)
N315 G43 H04 Z0.1 M08
N320 G01 Z-0.675 F7.0
N325 G00 Z0.1
N330 X2.5 (pt 15)
N335 G01 Z-0.675
N340 G00 Z0.1

N345 X3.0 (pt 16)
N350 G01 Z-0.675
N355 G00 Z0.1 M09
N360 G91 G28 Z0 M19
N365 M01

(0.750 end mill)
N370 T05 M06
N375 G54 G90 S400 M03 T01
N380 G00 X-0.475 Y-0.25 (pt 5)
N385 G43 H05 Z0.1 M08
N390 G01 Z-0.25 F50.
N395 X4.5 F4.5 (pt 6)
N400 G03 X5.25 Y0.5 R0.75 (pt7)
N405 G01 Y3.5 (pt 8)
N410 G03 X4.5 Y4.25 R0.75 (pt 9)
N415 G01 X0.5 (pt 10)
N420 G03 X-0.25 Y3.5 R0.75 (pt 11)
N425 G01 Y0.5 (pt 12)
N430 G03 X0.5 Y-0.25 R0.75 (pt 13)
N435 G00 Z0.1 M09
N440 G91 G28 Z0 M19
N445 M30

Programming activity 2: Test comprehension of motion types

#	X	Y	Z
1	-1.250	0	-.025/ -.605
2	-0.8839	-0.8839	-.025/ -.605
3	0	-1.250	-.025/ -.605
4	0.8839	-0.8839	-.025/ -.605
5	1.250	0	-.025/ -.605
6	0.8839	0.8839	-.025/ -.605
7	0	1.250	-.025/ -.605
8	-0.8839	0.8839	-.025/ -.605
9	-1.7875	0	-0.125
10	-0.7125	0	-0.125
11	-0.5038	0.5038	-0.125
12	-1.2639	1.2639	-0.125
13	0	1.7875	-0.125
14	0	0.7125	-0.125
15	0.5038	0.5038	-0.125
16	1.2639	1.2639	-0.125
17	1.7875	0	-0.125
18	0.7125	0	-0.125
19	0.5038	-0.5038	-0.125
20	1.2639	-1.2639	-0.125
21	0	-1.7875	-0.125
22	0	-0.7125	-0.125
23	-0.5038	-0.5038	-0.125
24	-1.2639	-1.2639	-0.125

O0002 (Program number)
(0.375 end mill)
N005 T01 M06

N010 G54 G90 S800 M03 T02
N015 G00 X-1.7875 Y0 (pt 9)
N020 G43 H01 Z0.1 M08
N025 G00 Z-0.125
N030 G01 X-0.7125 F4.5 (pt 10)
N035 G00 Z0.1
N040 X-0.5038 Y0.5038 (pt 11)
N045 Z-0.125
N050 G01 X-1.2639 Y1.2639 (pt 12)
N055 G00 Z0.1
N060 X0 Y1.7875 (pt 13)
N065 Z-0.125
N070 G01 Y0.7125 (pt 14)
N075 G00 Z0.1
N080 X0.5038 Y0.5038 (pt 15)
N085 Z-0.125
N090 G01 X1.2639 Y1.2639 (pt 16)
N095 G00 Z0.1
N100 X1.7875 Y0 (pt 17)
N105 Z-0.125
N110 G01 X0.7125 (pt 18)
N115 G00 Z0.1
N120 X0.5038 Y-0.5038 (pt 19)
N125 Z-0.125
N130 G01 X1.2639 Y-1.2639 (pt 20)
N135 G00 Z0.1
N140 X0 Y-1.7875 (pt 21)
N145 Z-0.125

N150 G01 Y-0.7125 (pt 22)
N155 G00 Z0.1
N160 X-0.5038 Y-0.5038 (pt 23)
N165 Z-0.125
N168 G01 X-1.2639 Y-1.2639(pt24)
N170 G00 Z0.1 M09
N175 G91 G28 Z0 M19
N180 M01

(Center drill)
N185 T02 M06
N190 G54 G90 S2000 M03 T03
N195 G00 X-1.25 Y0 (pt 1)
N200 G43 H02 Z0.1 M08
N205 Z-0.025 (0.1 approach position)
N210 G01 Z-0.325 F5.0
N215 G00 Z0.1
N220 X-0.8839 Y-0.8839 (pt 2)
N225 Z-0.025
N230 G01 Z-0.2
N235 G00 Z0.1
N240 X0 Y -1.25 (pt 3)
N245 Z-0.025
N250 G01 Z-0.2
N255 G00 Z0.1
N260 X0.8839 Y-0.8839 (pt 4)
N265 Z-0.025
N270 G01 Z-0.2
N275 G00 Z0.1
N280 X1.25 Y0 (pt 5)
N285 Z-0.025
N290 G01 Z-0.2
N295 G00 Z0.1
N300 X0.8839 Y0.8839 (pt 6)
N305 Z-0.025
N310 G01 Z-0.2
N315 G00 Z0.1
N320 X0 Y1.25 (pt 7)
N325 Z-0.025
N330 G01 Z-0.2
N335 G00 Z0.1
N340 X-0.8839 Y0.8839 (pt 8)
N345 Z-0.025
N350 G01 Z-0.2

N355 G00 Z0.1 M09
N360 G91 G28 Z0 M19
N365 M01

(0.25 drill)
N370 T03 M06
N375 G54 G90 S1200 M03 T01
N380 G00 X-1.25 Y0 (pt 1)
N385 G43 H03 Z0.1 M08
N390 Z-0.025 (Approach position)
N395 G01 Z-0.605 F6.1
N400 G00 Z0.1
N405 X-0.8839 Y-0.8839 (pt 2)
N410 Z-0.025
N415 G01 Z-0.605
N420 G00 Z0.1
N425 X0 Y-1.25 (pt 3)
N430 Z-0.025
N435 G01 Z-0.605
N440 G00 Z0.1
N445 X0.8839 Y-0.8839 (pt 4)
N450 Z-0.025
N455 G01 Z-0.605
N460 G00 Z0.1
N465 X1.25 Y0 (pt 5)
N470 Z-0.025
N475 G01 Z-0.605
N480 G00 Z0.1
N485 X0.8839 Y0.8839 (pt 6)
N490 Z-0.025
N495 G01 Z-0.605
N500 G00 Z0.1
N505 X0 Y1.25 (pt 7)
N510 Z-0.025
N515 G01 Z-0.605
N520 G00 Z0.1
N525 X-0.8839 Y0.8839 (pt 8)
N530 Z-0.025
N535 G01 Z-0.605
N540 G00 Z0.1 M09
N545 G91 G28 Z0 M19
N550 M01
N555 M30

Programming activity 3: Test comprehension of tool length compensation

#	X	Y	Z
1	0.5	0.5	.1/-.605/-.25
2	1.0	0.5	.1/-.6/-.56
3	3.5	0.5	.1/-.605/-.25
4	2.0	1.5	.1/-.68
5	3.5	2.5	.1/-.605/-.25
6	3.0	2.5	.1/-.6/-.56
7	0.5	2.5	.1/-.605/-.25

O0003 (Program number)

(#3 center drill)
N005 T01 M06
N010 G54 G90 S1200 M03 T02
N015 G00 X0.5 Y0.5 (pt 1)
N020 G43 H01 Z0.1 M08
N025 G01 Z-0.2 F5.0

N030 G00 Z0.1
N035 X1.0 (pt 2)
N040 G01 Z-0.2
N045 G00 Z0.1
N050 X3.5 (pt 3)
N055 G01 Z-0.2
N060 G00 Z0.1
N065 X2.0 Y1.5 (pt 4)
N070 G01 Z-0.2
N075 G00 Z0.1
N080 X3.5 Y2.5 (pt 5)
N085 G01 Z-0.2
N090 G00 Z0.1
N095 X3.0 (pt 6)
N100 G01 Z-0.2
N105 G00 Z0.1
N110 X0.5 (pt 7)
N115 G01 Z-0.2
N120 G00 Z0.1 M09
N125 G91 G28 Z0 M19
N130 M01

(1/4 drill)
N135 T02 M06
N140 G54 G90 S1200 M03 T03
N145 G00 X0.5 Y2.5 (pt 7)
N150 G43 H02 Z0.1 M08
N155 G01 Z-0.605 F6.1
N160 G00 Z0.1
N165 Y0.5 (pt 1)
N168 G01 Z-0.605
N170 G00 Z0.1
N175 X3.5 (pt 3)
N180 G01 Z-0.605
N185 G00 Z0.1
N190 Y2.5 (pt 5)
N195 G01 Z-0.605
N200 G00 Z0.1 M09
N205 G91 G28 M19
N210 M01

(3/8 end mill)
N215 T03 M06
N220 G54 G90 S800 M03 T04
N225 G00 X3.5 Y2.5 M08 (pt 5)
N230 G43 H03 Z0.1
N235 G01 Z-0.25 F7.0
N240 G00 Z0.1

N245 Y0.5 (pt 3)
N250 G01 Z-0.25
N255 G00 Z0.1
N260 X0.5 (pt 1)
N265 G01 Z-0.25
N270 G00 Z0.1
N275 Y2.5 (pt 7)
N280 G01 Z-0.25
N285 G00 Z0.1 M09
N290 G91 G28 M19
N295 M01

(15/64 drill)
N300 T04 M06
N305 G54 G90 S1200 M03 T05
N310 G00 X1.0 Y0.5 (pt 2)
N315 G43 H04 Z0.1 M08
N320 G01 Z-0.6 F5.0
N325 G00 Z0.1
N330 X3.0 Y2.5 (pt 6)
N335 G01 Z-0.6
N340 G00 Z0.1 M09
N345 G91 G28 Z0 M19
N350 M01

(0.250 reamer)
N355 T05 M06
N360 G54 G90 S700 M03 T06
N365 G00 X3.0 Y2.5 (pt 6)
N370 G43 H05 Z0.1 M08
N375 G01 Z-0.56 F7.0
N380 G00 Z0.1
N385 X1.0 Y0.5 (pt 2)
N390 G01 Z-0.56
N395 G00 Z0.1 M09
N400 G91 G28 Z0 M19
N405 M01

(1/2 drill)
N410 T06 M06
N415 G54 G90 S600 M03 T01
N420 G00 X2.0 Y1.5
N425 G43 H06 Z0.1 M08
N430 G01 Z-0.68 F5.5
N435 G01 Z0.1 M09
N440 G91 G28 M19
N445 M30

Programming activity 4: Test comprehension of cutter radius compensation

#	X	Y	Z
1	-0.600	-0.35	-0.25
2	-0.600	0.25	-0.25
3	3.625	0.25	-0.25
4	3.75	0.375	-0.25
5	3.75	2.625	-0.25
6	3.625	2.75	-0.25
7	0.375	2.75	-0.25

8	0.25	2.625	-0.25
9	0.25	0.375	-0.25
10	0.375	0.25	-0.25
11	1.125	-0.5	-0.25
12	2.0	-0.1625	-0.125
13	2.0	3.1625	-0.125
14	1.625	3.1625	-0.125
15	1.625	-0.1625	-0.125
16	2.375	-0.1625	-0.125
17	2.375	3.1625	-0.125
18	2.0	0.75	-.025/-.767
19	2.0	1.5	-.025/-.767
20	2.0	2.25	-.025/-.767

O0004 (Program number)

(1 inch end mill)
N005 T01 M06
N010 G54 G90 S350 M03 T02
N015 G00 X-0.6 Y-0.35 (pt 1)
N020 G43 H01 Z-0.25 M08
N025 G42 D31 Y0.25 (2)
N030 G01 X3.625 F5.0 (3)
N035 G03 X3.75 Y0.375 R0.125 (4)
N040 G01 Y2.625 (5)
N045 G03 X3.625 Y2.75 R0.125 (6)
N050 G01 X0.375 (7)
N055 G03 X0.25 Y2.625 R0.125 (8)
N060 G01 Y0.375 (9)
N065 G03 X0.375 Y0.25 R0.125(10)
N070 G02 X1.125 Y-0.5 R0.75 (11)
N075 G00 Z0.1
N080 G40 M09
N085 G91 G28 Z0 M19
N090 M01

(5/8 end mill)
N095 T02 M06
N100 G54 G90 S650 M03 T03
N105 G00 X2.0 Y-0.1625 (pt 12)
N110 G43 H02 Z-0.125 M08
N115 G01 Y3.1625 F4.0 (pt13)

N120 G00 G41 D32 X1.625 (pt 14)
N125 G01 Y-0.1625 (pt15)
N130 G00 G40 X2.0 (pt 12)
N135 G41 D32 X2.375 (pt 16)
N140 G01 Y3.1625 (pt 17)
N145 G00 G40 X2.0 M09 (pt 13)
N150 G91 G28 Z0 M19
N155 M01

(#3 center drill)
N160 T03 M06
N165 G54 G90 S1200 M03 T04
N168 G00 X2.0 Y0.75 (pt 18)
N170 G43 H03 Z-0.025
N175 G01 Z-0.325 F5.0
N180 G00 Z-0.025
N185 Y1.5 (pt 19)
N190 G01 Z-0.325
N195 G00 Z-0.025
N200 Y2.25 (pt 20)
N205 G01 Z-0.325
N210 G00 Z0.1 M09
N215 G91 G28 Z0 M19
N220 M01

(3/8 drill)
N225 T04 M06
N230 G54 G90 S700 M03 T01
N235 G00 X2.0 Y2.25 (pt 20)
N240 G43 H04 Z-0.025
N245 G01 Z-0.767 F6.0
N250 G00 Z-0.025
N255 Y1.5 (pt19)
N260 G01 Z-0.767
N265 G00 Z-0.025
N270 Y0.75 (pt 18)
N275 G01 Z-0.767
N280 G00 Z0.1 M09
N285 G91 G28 M19
N290 M30

Programming activity 5: Test comprehension of fixture offsets

Fixture offset settings:

#	X	Y	Z	
0	-17.3654	-10.3215	-12.2464	
1	1.0	1.375	1.0	
2	6.0	1.0	2.0	
3	11.0	1.5	4.0	
4	Not needed	Not needed	Not needed	
5	Not needed	Not needed	Not needed	
6	Not needed	Not needed	Not needed	

Coordinates:

#	X	Y	Z
1	0.375	0.375	0/-.25/-1.105
2	0.375	1.625	0/-.25/-1.105
3	3.625	1.625	0/-.25/-1.105
4	3.625	0.375	0/-.25/-1.105
5	2.0	1.0	0/-1.330
6	2.0	0.5	0/-0.85
7	0.5	1.0	0/-0.85

O0005 (Program number)

(#3 center drill)
N005 T01 M06
N010 G56 G90 S1200 M03 T02
N015 G00 X0.5 Y1.0 (pt 7)
N020 G43 H01 Z0.1 M08
N025 G01 Z-0.2 F5.0
N030 G00 Z0.1
N035 G55 X2.0 Y0.5 (pt 6)
N040 Z0.1
N045 G01 Z-0.2
N050 G00 Z0.1
N055 G54 X3.625 Y0.375 (pt 4)
N060 Z0.1
N065 G01 Z-0.2
N070 G00 Z0.1
N075 Y1.625 (pt 3)
N080 G01 Z-0.2
N085 G00 Z0.1
N090 X2.0 Y1.0 (pt 5)
N095 G01 Z-0.2
N100 G00 Z0.1
N105 X0.375 Y1.625 (pt 2)
N110 G01 Z-0.2
N115 G00 Z0.1
N120 Y0.375 (pt 1)
N125 G01 Z-0.2
N130 G00 Z0.1 M09
N135 G91 G28 Z0 M19
N140 M01

(1/4 drill)
N145 T02 M06
N150 G54 G90 S900 M03 T03
N155 G00 X0.375 Y0.375 (pt 1)
N160 G43 H02 Z0.1 M08
N165 G01 Z-1.105 F4.0
N168 G00 Z0.1
N170 Y1.625 (pt 2)
N175 G01 Z-1.105
N180 G00 Z0.1
N185 X3.625 (pt 3)
N190 G01 Z-1.105
N195 G00 Z0.1
N200 Y0.375 (pt 4)
N205 G01 Z-1.105
N210 G00 Z0.1 M09

Notice how careful you must be with the Z axis commands when moving from one fixture offset to another. If moving from a high surface to a low surface, you can instate the fixture offset change in the XY movement, then drop down in Z. On the other hand, if moving from a low surface to a high surface,

N215 G91 G28 Z0 M19
N220 M01

(3/8 counter bore)
N225 T03 M06
N230 G54 G90 S700 M03 T04
N235 G00 X3.625 Y0.375 (pt 4)
N240 G43 H03 Z0.1
N245 G01 Z-0.25 F5.0
N250 G00 Z0.1
N255 Y1.625 (pt 3)
N260 G01 Z-0.25
N265 G00 Z0.1
N270 X0.375 (pt 2)
N275 G01 Z-0.25
N280 G00 Z0.1
N285 Y0.375 (pt 1)
N290 G01 Z-0.25
N295 G00 Z0.1 M09
N300 G91 G28 Z0 M19
N305 M01

(1 inch drill)
N310 T04 M06
N315 G54 G90 S350 M03 T05
N320 G00 X2.0 Y1.0 (pt 5)
N325 G43 H04 Z0.1 M08
N330 G01 Z-1.33 F6.0
N335 G00 Z0.1 M09
N340 G91 G28 Z0 M19
N345 M01

(1/2 drill)
N350 T05 M06
N355 G55 G90 S700 M03 T01
N360 G00 X2.0 Y0.5 (pt 6)
N365 G43 H05 Z0.1 M08
N370 G01 Z-0.85 F5.5
N375 G00 Z0.1
N380 G56 Z0.1
N385 X0.5 Y1.0 (pt 7)
N390 G01 Z-1.85
N395 G00 Z0.1 M09
N400 G91 G28 Z0 M19
N405 M30

you *must* instate the new fixture offset in a command that clears the next Z surface *before* moving in XY.

Programming activity 6: Test comprehension of cutter radius compensation

#	X	Y	Z
1	-0.9625	0	-0.25
2	-0.9625	0.25	-0.25
3	-1.625	0.25	-0.25
4	-1.625	-0.25	-0.25
5	-0.9625	-0.25	-0.25
6	0.9625	0	-0.25
7	0.9625	-0.25	-0.25
8	1.625	-0.25	-0.25
9	1.625	0.25	-0.25
10	0.9625	0.25	-0.25
11	0	1.625	0/-.68/-.57
12	0	-1.625	0/-.68/-.57
13	-1.625	0	-.25/-.605
14	1.625	0	-.25/-.605

O0006 (Program number)

(3/8 end mill)
N005 T01 M06
N010 G54 G90 S800 M03 T02
N015 G00 X-0.9625 Y0 (pt 1)
N020 G43 H01 Z-0.25 M08
N025 G41 D31 Y0.25 (pt 2)
N030 G01 X-1.625 F3.0 (pt 3)
N035 G03 Y-0.25 R0.25 (pt 4)
N040 G01 X-0.9625 (pt 5)
N045 G00 G40 Y0 (pt 1)
N050 X0.9625 (pt 6)
N055 G41 D31 Y-0.25 (pt 7)
N060 G01 X1.625 (pt 8)
N065 G03 Y0.25 R0.25 (pt 9)
N070 G01 X0.9625 (pt 10)
N075 G00 G40 Y0 (pt 11)
N080 G00 Z0.1 M09
N085 G91 G28 Z0 M19
N090 M01

(# 3 center drill)
N095 T02 M06
N100 G54 G90 S1200 M03 T03
N105 G00 X0 Y1.625 (pt 11)
N110 G43 H02 Z0.1 M08
N115 G01 Z-0.2 F5.0
N120 G00 Z0.1
N125 Y-1.625 (pt 12)
N130 G01 Z-0.2
N135 G00 Z0.1
N140 X-1.625 Y0 (pt 13)
N145 Z-0.15
N150 G01 Z-0.55
N155 G00 Z-0.15

N160 X1.625 (pt 14)
N165 G01 Z-0.55
N168 G00 Z0.1 M09
N170 G91 G28 Z0 M19
N175 M01

(1/4 drill)
N180 T03 M06
N185 G54 G90 S1200 M03 T04
N190 G00 X1.625 Y0 (pt 14)
N195 G43 H03 Z-0.15 M08
N200 G01 Z-0.605 F5.0
N205 G00 Z-0.15
N210 X-1.625 (pt 13)
N215 G01 Z-0.605
N220 G00 Z0.1 M09
N225 G91 G28 Z0 M19
N230 M01

(15/32 drill)
N235 T04 M06
N240 G54 G90 S500 M03 T05
N245 G00 X0 Y1.625 (pt 11)
N250 G43 H04 Z0.1 M08
N255 G01 Z-0.68 F6.0
N260 G00 Z0.1
N265 Y-1.625 (pt 12)
N270 G01 Z-0.68
N275 G00 Z0.1 M09
N280 G91 G28 Z0 M19
N285 M01

(0.500 reamer)
N290 T05 M06
N295 G54 G90 S400 M03 T01
N300 G00 X0 Y-1.625 (pt 12)
N305 G43 H05 Z0.1 M08
N310 G01 Z-0.57 F7.5
N315 G00 Z0.1
N320 Y1.625 (pt 11)
N325 G01 Z-0.57
N330 G00 Z0.1 M09
N335 G91 G28 Z0 M19
N340 M30

Programming activity 7: Test comprehension of program formatting

#	X	Y	Z
1	2.0	1.0	0/-1.005/-.375
2	2.0	1.25	-0.375
3	1.5	1.5	-0.375
4	2.0	1.75	-0.375
5	2.0	0.25	-0.375
6	2.5	1.25	-0.375
7	0.75	1.0	0/-.93
8	3.25	1.0	0/-.93
9	2.0	1.5625	-0.375/-.817
10	2.5625	1.0	-0.375/-.817
11	2.0	0.4375	-0.375/-.817
12	1.4375	1.0	-0.375/-.817

Do not be overly concerned with the order by which you machine the holes. For example, you can center drill the holes in any order. We are most concerned with the structure of this program. *Write the program in the format for a vertical machining center that uses fixture offsets.*

O0007 (Program number)

(3/4 Drill)
N005 T01 M06
N010 G54 G90 S375 M03 T02
N015 G00 X2.0 Y1.0 (pt 1)
N020 G43 H01 Z0.1
N025 M08
N030 G01 Z-1.005 F4.5
N035 G00 Z0.1
N040 M09
N045 G91 G28 Z0 M19
N050 M01

(5/8 end mill)
N055 T02 M06
N060 G54 G90 S400 M03 T03
N065 G00 X2.0 Y1.0 (pt1)
N070 G43 H02 Z-0.375
N075 M08
N080 G01 Y1.25 F4.0 (pt 2)
N085 G42 D32 X1.5 (pt 3)
N090 G02 X2.0 Y1.75 R0.5 (pt 4)
N095 Y0.25 R0.75 (pt 5)
N100 Y1.75 R0.75 (pt 4)
N105 X2.5 Y1.25 R0.5 (pt 6)
N110 G01 G40 X2.0 (pt 2)
N115 G00 Z0.1
N120 M09
N125 G91 G28 Z0 M19
N130 M01

(#3 center drill)
N135 T03 M06
N140 G54 G90 S1200 M03 T04
N145 G00 X3.25 Y1.0 (pt 8)
N150 G43 H03 Z0.1
N155 M08
N160 G01 Z-0.2 F5.0
N165 G00 Z0.1
N170 X0.75 (pt 7)
N175 G01 Z-0.2
N180 G00 Z0.1
N185 X1.4375 (pt 12)
N190 Z-0.275
N195 G01 Z-0.575
N200 G00 Z0.1
N205 X2.0 Y0.4375 (pt 11)
N210 Z-0.275
N215 G01 Z-0.575
N220 G00 Z0.1
N225 X2.5625 Y1.0 (pt 10)
N230 Z-0.275
N235 G01 Z-0.575
N240 G00 Z0.1
N245 X2.0 Y1.5625 (pt 9)
N250 Z-0.275
N255 G01 Z-0.575
N260 G00 Z0.1
N265 M09
N270 G91 G28 Z0 M19
N275 M01

(1/2 drill)
N280 T04 M06
N285 G54 G90 S500 M03 T05
N290 G00 X3.25 Y1.0 (pt 8)
N295 G43 H04 Z0.1
N300 M08
N305 G01 Z-0.93 F5.0
N310 G00 Z0.1
N315 G00 X0.75 (pt 7)
N320 G01 Z-0.93
N325 G00 Z0.1
N330 M09
N335 G91 G28 Z0 M19
N340 M01

(1/8 drill)
N345 T05 M06
N350 G54 G90 S1800 M03 T01
N355 G00 X1.4375 Y1.0 (pt 12)
N360 G43 H05 Z-0.275

N365 M08
N370 G01 Z-0.817 F3.0
N375 G00 Z0.1
N380 X2.0 Y0.4375 (pt 11)
N385 Z-0.275
N390 G01 Z-0.817
N395 G00 Z0.1
N400 X2.5625 Y1.0 (pt 10)
N405 Z-0.275
N410 G01 Z-0.817

N415 G00 Z0.1
N420 X2.0 Y1.5625 (pt 9)
N425 Z-0.275
N430 G01 Z-0.817
N435 M09
N440 G91 G28 Z0 M19
N445 G28 X0 Y0
N450 M01
N455 M30

Programming activity 8: Test comprehension of canned cycles

#	X	Y	Z
1	0.375	0.375	0/-.855/-.25
2	2.875	0.375	0/-.855/-.25
3	3.625	0.375	0/-.855/-.25
4	3.625	2.625	0/-.855/-.25
5	2.875	2.0	0/-.855/-.25
6	1.125	1.0	0/-.855/-.25
7	1.125	2.625	0/-.855/-.25
8	0.375	2.625	0/-.855/-.25
9	1.125	1.5	-0.125
10	1.125	2.125	-0.125
11	2.875	0.875	-0.125
12	2.875	1.5	-0.125

O0008 (Program number)

(1/4 end mill)
N005 T01 M06
N010 G54 G90 S900 M03 T02
N015 G00 X1.125 Y1.5 (pt 9)
N020 G43 H01 Z0.1
N025 M08
N030G01 Z-0.125 F2.0
N035 Y2.125 (pt 10)
N040 G00 Z0.1
N045 X2.875 Y0.875 (pt 11)
N050 G01 Z-0.125
N055 Y1.5 (pt 12)
N060 G00 Z0.1
N065 M09
N065 G91 G28 Z0 M19
N070 M01

(#3 center drill)
N075 T02 M06
N080 G54 G90 S1200 M03 T03
N085 G00 X0.375 Y0.375 (pt 1)
N090 G43 H02 Z2.0
N095 M08
N100 G81 R0.1 Z-0.2 F4.0 G99 (pt 1)
N105 X2.875 (pt 2)
N110 X3.625 G98 (pt 3)
N115 Y2.625 (pt 4)

N120 X2.875 Y2.0 G99 (pt 5)
N125 X1.125 Y1.0 (pt 6
N130 Y2.625 (pt 7)
N135 X0.375 (pt 8)
N140 G80 M09
N145 G91 G28 Z0 M19
N150 M01

(1/4 drill)
N155 T03 M06
N160 G54 G90 S1100 M03 T04
N165 G00 X0.375 Y2.625 (pt 8)
N170 G43 H03 Z2.0
N175 M08
N180 G73 R0.1 Z-0.855 Q0.1 F4.5 G98
N185 Y0.375 G99 (pt 1)
N190 X3.625 G98 (pt 3)
N195 Y2.625 (pt 4)
N200 G80 M09
N205 G91 G28 Z0 M19
N210 M01

(3/8 counter bore)
N215 T04 M06
N220 G54 G90 S700 M03 T05
N225 G00 X3.625 Y2.625 (pt 4)
N230 G43 H04 Z2.0
N235 M08
N240 G82 R0.1 Z-0.25 F5.5 G98 (pt 4)
N245 Y0.375 G99 (pt 3)
N250 X0.375 G98 (pt 1)
N255 Y2.625 (pt 8)
N260 G80 M09
N265 G91 G28 Z0 M19
N270 M01

(11/64 drill)
N275 T05 M06
N280 G54 G90 S1400 M03 T06
N285 G00 X1.125 Y2.625 (pt 7)
N290 G43 H05 Z0.1 (no clamp in the way)
N295 M08

N300 G83 R0.1 Z-0.831 Q0.55 F3.0 (pt 7)
N305 X2.875 Y0.375 (pt 2)
N310 G80 M09
N315 G91 G28 Z0 M19
N320 M01

(0.1875 reamer)
N325 T06 M06
N330 G54 G90 S1000 M03 T07
N335 G00 X2.875 Y0.375 (pt 2)
N340 G43 H06 Z0.1
N345 M08
N350 G81 R0.1 Z-0.81 F5.0 (pt 2)
N355 X1.125 Y2.625 (pt 7)
N360 G80 M09
N365 G91 G28 Z0 M19
N370 M01

(27/64 drill)
N375 T07 M06
N380 G54 G90 S550 M03 T08
N385 G00 X1.125 Y1.0 (pt 6)

N390 G43 H07 Z0.1
N395 M08
N400 G73 R0.1 Z-0.906 F6.0 (pt 6)
N405 X2.875 Y2.0 (pt 5)
N410 G80 M09
N415 G91 G28 Z0 M19
N420 M01

(1/2-13 tap)
N425 T08 M06
N430 G54 G90 S400 M03 T01
N435 G00 X2.875 Y2.0 (pt 5)
N440 G43 H08 Z0.25
N445 M08
N450 G84 R0.25 Z-0.97 F30.7 (pt 5)
N455 X1.125 Y1.0 (pt 6)
N460 G80 M09
N465 G91 G28 Z0 M19
N470 G28 X0 Y0
N475 M01
N480 M30

Programming activity 9: Test comprehension of sub programming

#	X	Y	Z
1	0.125	0.125	0/-.12/-0.317
2	1.375	0.125	0/-.12/-0.317
3	1.375	0.875	0/-.12/-0.317
4	0.125	0.875	0/-.12/-0.317
5	0.25	0.5	0/-.188/-0.28
6	1.25	0.5	0/-.188/-0.28

O0009 (Main program)
(#3 center drill)
N005 T01 M06
N010 G54 G90 S1200 M03 T02
N015 G00 X0.125 Y0.125 (pt 1)
N020 G43 H01 Z0.1
N025 M08
N030 M98 P1001 (Machine G54 holes)
N035 G55
N040 M98 P1001 (Machine G55 holes)
N045 G56
N050 M98 P1001 (Machine G56 holes)
N055 G57
N060 M98 P1001 (Machine G57 holes)
N065 G58
N070 M98 P1001 (Machine G58 holes)
N075 G59
N080 M98 P1001 (Machine G59 holes)
N085 M09
N090 G91 G28 Z0 M19
N095 M01

(1/8 drill)
N100 T02 M06
N105 G54 G90 S1000 M03 T03
N110 G00 X0.125 Y0.125 (pt 1)
N115 G43 H02 Z0.1
N120 M08
N125 M98 P1002 (Machine G54 holes)
N130 G55
N135 M98 P1002 (Machine G55 holes)
N140 G56
N145 M98 P1002 (Machine G56 holes)
N150 G57
N155 M98 P1002 (Machine G57 holes)
N160 G58
N165 M98 P1002 (Machine G58 holes)
N170 G59
N175 M98 P1002 (Machine G59 holes)
N180 M09
N185 G91 G28 Z0 M19
N190 M01

(1/4 end mill)
N195 T03 M06
N200 G54 G90 S800 M03 T04
N205 G00 X0.25 Y0.5 (pt 5)
N210 G43 H03 Z0.1
N215 M08
N220 M98 P1003 (Mill G54 slot)
N225 G55
N230 M98 P1003 (Mill G55 slot))
N235 G56
N240 M98 P1003 (Mill G56 slot)
N245 G57
N250 M98 P1003 (Mill G57 slot)
N255 G58
N260 M98 P1003 (Mill G58 slot)
N265 G59
N270 M98 P1003 (Mill G59 slot)
N275 M09
N280 G91 G28 Z0 M19
N285 M01

(1/8 end mill)
N290 T04 M06
N295 G54 G90 S1000 M03 T01
N300 G00 X0.25 Y0.5 (pt 5)
N305 G43 H04 Z0.1
N310 M08
N315 M98 P1004 (Mill G54 slot)
N320 G55
N325 M98 P1004 (Mill G55 slot))
N330 G56
N335 M98 P1004 (Mill G56 slot)
N340 G57
N345 M98 P1004 (Mill G57 slot)
N350 G58
N355 M98 P1004 (Mill G58 slot)
N360 G59
N365 M98 P1004 (Mill G59 slot)
N370 M09
N375 G91 G28 Z0 M19

N380 G28 X0 Y0
N385 M01
N390 M30

O1001 (Subprogram for #3 center drill)
N1 G81 X0.125 Y0.125 R0.1 Z-.12 F5.0 (pt 1)
N2 Y0.875 (pt 4)
N3 X1.375 (pt 3)
N4 Y0.125 (pt 2)
N5 G80
N6 M99

O1002 (Subprogram for 1/8 drill)
N1 G73 X0.125 Y0.125 R0.1 Z-.317 Q0.1 F4.0
N2 Y0.875 (pt 4)
N3 X1.375 (pt 3)
N4 Y0.125 (pt 2)
N5 G80
N6 M99

O1003 (Subprogram for 1/4 end mill)
N1 G00 X0.25 Y0.5 (pt 5)
N2 G01 Z-0.188 F2.75
N3 X1.25 F5.5 (pt 6)
N4 G00 Z0.1
N5 M99

O1004 (Subprogram for 1/8 end mill)
N1 G00 X0.25 Y0.5 (pt 5)
N2 G01 Z-0.28 F1.5
N3 X1.25 F3.0 (pt 6)
N4 G00 Z0.1
N5 M99

Programming activity 10: Test comprehension of thread milling

#	X	Y	Z
1	2.0	1.7315	-0.55
2	1.65	1.7315	-0.55
3	2.0	2.0815	-0.55
4	2.0	0.9185	-0.55
5	2.35	1.7315	-0.55
6	2.0	1.775	-0.55
7	1.65	1.775	-0.55
8	2.0	2.125	-.5652/-.6276
9	2.0	0.875	-0.5964
10	2.35	1.775	-0.6432
11	0.375	0.375	0/-.2/-.642
12	0.375	2.625	0/-.2/-.642
13	3.625	2.625	0/-.2/-.642
14	3.625	0.375	0/-.2/-.642
15	2.0	1.5	00/-.2/-.867

O0010 (Program number)

(#3 center drill)
N005 T01 M06
N010 G54 G90 S1200 M03 T02
N015 G00 X0.375 Y0.375 (pt 11)
N020 G43 H01 Z0.1
N025 M08

N030 G81 R0.1 Z-0.2 F5.0 (pt 11)
N035 Y2.625 (pt 12)
N040 X2.0 Y1.5 (pt 15)
N045 X3.625 Y2.625 (pt 13)
N050 Y0.375 (pt 14)
N055 G80 M09
N060 G91 G28 Z0 M19
N065 M01

(3/8 drill)
N070 T02 M06
N075 G54 G90 S600 M03 T03
N080 G00 X3.625 Y0.375 (pt 14)
N085 G43 H02 Z0.1
N090 M08
N095 G73 R0.1 Z-0.642 F5.0 Q0.1 (pt 14)
N100 Y2.625 (pt 13)
N105 X0.375 (pt 12)
N110 Y0.375 (pt 11)
N115 G80 M09
N120 G91 G28 Z0 M19
N125 M01

(1-1/8 drill)
N130 T03 M06
N135 G54 G90 S270 M03 T04
N140 G00 X2.0 Y1.5 (pt 15)
N145 G43 H03 Z0.1
N150 M08
N155 G73 R0.1 Z-0.867 Q0.1 F6.0 (pt 15)
N160 G80 M09
N165 G91 G28 Z0 M19
N170 M01

(1/2 end mill)

N175 T04 M06
N180 G54 G90 S700 M03 T05
N185 G00 X2.0 Y1.7315 (pt 1)
N190 G43 H04 Z0.1
N195 M08
N200 G01 Z-0.55 F50.0
N205 G42 D34 X1.65 F4.5 (pt 2)
N210 G02 X2.0 Y2.0815 R0.35 (pt 3)
N215 Y0.9185 R0.5815 (pt 4)
N220 Y2.0815 R0.5815 (pt 3)
N225 X2.35 Y1.7315 R0.35 (pt 5)
N230 G01 G40 X2.0 (pt 1)
N235 G00 Z0.1 M09
N240 G91 G28 Z0 M19
N245 M01

(1/2 thread mill)
N250 T05 M06
N255 G54 G90 S650 M03 T01
N260 G00 X2.0 Y1.775 (pt 6)
N265 G43 H05 Z0.1
N270 M08
N275 G01 Z-0.55 F50.0
N280 G42 D35 X1.65 F4.0 (pt 7)
N285 G02 X2. Y2.125 Z-0.5652 R0.35 (8)
N290 Y0.875 Z-0.5964 R0.625 (9)
N295 Y2.125 Z-0.6276 R0.625 (8)
N300 X2.35 Y1.775 Z-0.6432 R0.35 (pt 10)
N305 G01 G40 X2.0 (pt 1)
N310 G00 Z0.1 M09
N315 G91 G28 Z0 M19
N320 G28 X0 Y0
N325 M01
N330 M30

Programming activity 11: Test comprehension of rotary devices

| #	X	Y	Z
1	0.75	0	0.5/0.145
2	1.5	0	0.5/0.145
3	2.25	0	0.5/0.145
4	0.75	0	0.5/0.145
5	1.5	0	0.5/0.145
6	2.25	0	0.5/0.145
7	0.625	0	-0.25
8	2.375	0	-0.25

O0011 (Program number)

(Confirm upper side facing spindle:)
N005 G91 G28 A0
(#3 center drill)
N010 T01 M06
N015 G54 G90 S1200 M03 T02
N020 G00 X0.75 Y0 (pt 1)

N025 G43 H01 Z2.5 (Set up indexing plane)
N030 M08
N035 G81 R0.6 Z0.35 F5.0 G99 (1)
N040 X1.5 (pt 2)
N045 X2.25 G98(pt 3)
N050 G80
N055 G91 G00 A180.0 (Rotate 180 deg)
N060 G90 G81 R0.6 Z0.35 F5.0 G99 (6)
N065 X1.5 (pt 5)
N070 X0.75 G98 (pt 4)
N075 G80 M09
N080 G91 G28 Z0 M19
N085 M01

(1/4 drill)
N090 T02 M06

N095 G54 G90 S800 M03 T03
N100 G00 X0.75 Y0 (pt 4)
N105 G43 H02 Z2.5
N110 G81 R0.6 Z0.145 F3.5 G99 (pt 4)
N115 X1.5 (pt 5)
N120 X2.25 G98 (pt 6)
N125 G80
N130 G91 G00 A180.0 (Rotate 180 deg)
N135 G90 G81 R0.6 Z0.145 F3.5 G99 (3)
N140 X1.5 (pt 2)
N145 X0.75 (pt 1)
N150 G80 M09
N155 G91 G28 Z0 M19
N160 M01

(1/2 end mill)
N165 G91 G00 A90.0 (Rotate 90 deg)
N170 T03 M06
N175 G54 G90 S600 M03 T01
N180 G00 X0.625 Y0 (pt 7)
N185 G43 H03 Z0.6
N190 G01 Z-0.25 F2.5
N195 X2.375 F5.0 (pt 8)
N200 G00 Z0.6
N205 G91 G28 Z0 A0 (Rotate to start)
N210 M30

Slides in the presentation named "review"

Each lesson outline slide is identical to the presentation links slide for that lesson

Copyright 2009, CNC Concepts, Inc.

CNC CONCEPTS, INC.
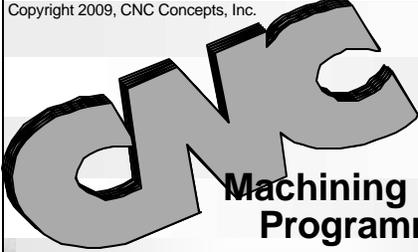
Machining Center Programming

Course Review

Home

Key Concept Number One

You must know your machine from a programmer's viewpoint!

✐ Machine configurations
✐ Visualizing the execution of a CNC program
✐ Understanding program zero
✐ Locating program zero
✐ The two ways to assign program zero

Home

1: Machine configurations

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

Home

2: Visualizing a program's execution

The importance of visualizing
    Travel instructions analogy
Program make-up
    Sequential order of execution
Machinist vs programmer
    Advantage of machinist
    Programmer's disadvantage
A job handled by:
    Manual machinist
    CNC programmer

Program structure notes
    Sequence numbers
    Word order in a command
    Decimal point usage
    Modal words
    Initialized words
    Common mistakes

Home

3: Understanding program zero

Rectangular coordinate system
    Axis drive make-up
    Graph analogy
    3d coordinate system
More on polarity
    in XY
    in Z
Where to place program zero
    in XY
    in Z

Absolute vs incremental
        Absolute
            Example
        Incremental
        Comparison
    Inch versus metric

Home

4: Introduction to programming words

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

Word types (continued)
    R – rapid plane
    I J K – directional vectors
    Q – peck depth
    P – pause time
    L – number of executions
    F – feedrate
    S – spindle speed
    T – tool changing
    M – miscellaneous functions
    D H – offset number specification
    / - block delete

Home

As you get deeper into the course, use this presentation to quickly review what you have already covered. If students are having trouble with a review topic, simply recall the presentation for the lesson, go to the presentation links slide for that lesson, and click on the topic in question to repeat the related presentation.

Slides in the presentation named "review"

Each lesson outline slide is identical to the presentation links slide for that lesson

Key Concept Number Two

You must prepare to write programs!

✎ The importance of preparation

Home

5: The importance of preparation

Key concept #2
Divide and conquer
Typical mistakes
  Syntax mistakes
  Mistakes of omission
  Motion mistakes

Preparation steps
  Study & mark up print
  Develop the machining process
  Do the math
    Number all points
    Develop a coordinate sheet
  Plan the setup
Conclusion

Home

Key Concept Number Three

You must understand the motion types!

✎ Programming motion commands

Home

6: Programming motion commands

Key concept #3
Interpolation
Three motion types
Motion commonalities
Programmed point
  For hole-making tools
  For milling cutters
Rapid motion
  Examples
Linear motion
  Examples

Circular motion
  G02 or G03?
  Specifying arc size
    With R word
      Example
      Warning about R word
    With Directional vectors
  Arc limitations
  Full circle in one command
    Example
Helical motion

Home

Key Concept Number Four

You must understand the compensation types

✎ What is compensation?
✎ Tool length compensation
✎ Cutter radius compensation
✎ Fixture offsets

Home

7: What is compensation?

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

Home

As you get deeper into the course, use this presentation to quickly review what you have already covered. If students are having trouble with a review topic, simply recall the presentation for the lesson, go to the presentation links slide for that lesson, and click on the topic in question to repeat the related presentation.

Slides in the presentation named "review"

Each lesson outline slide is identical to the presentation links slide for that lesson

8: Tool length compensation

Reasons for TLC
  Tools are different lengths
  Sizing and tool wear
  Trial machining
Two ways to use
  Offset is tool length
    Measuring on machine
    Measuring off line
    Entering offsets from program
    Review of alternatives
  Offset is tool tip to work surface

Programming TLC
  Words involved
  Example program
Typical mistakes
  Forgetting to instate
  Forgetting to enter offset
Sizing and trial machining

Home

9: Cutter radius compensation

Reasons for CRC
  When to use
  Simplifies calculations
  Range of cutter sizes
  Sizing & trial machining
  Roughing with finish coord.
Two ways to use
  Offset is cutter radius
  Offset is deviation
Three steps to using
  Instate
  Machine surfaces
  Cancel

Examples
  Mill left side of part
  Mill outside contour
  Mill counter-bored hole
Sizing and trial machining
  Sizing
  Trial machining
Roughing with finish coord.

Home

10: Fixture offsets

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

Home

Key Concept Number Five

You must know how to structure your programs

✍ Program formatting
✍ Four kinds of program format

Home

11: Program formatting

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

Home

12: The four kinds of program format

A few more notes
  G92 vs fixture offsets
  Safety commands
  Documentation
  How G28 works
  Using G53
  When to tool change

Program formats
  Vertical MC using G92
  Vertical MC using fixture offsets
  Horizontal MC using G92
  Horizontal MC using fixture offsets
Example program
  Points about example program

Home

As you get deeper into the course, use this presentation to quickly review what you have already covered. If students are having trouble with a review topic, simply recall the presentation for the lesson, go to the presentation links slide for that lesson, and click on the topic in question to repeat the related presentation.

Slides in the presentation named "review"

Each lesson outline slide is identical to the presentation links slide for that lesson

Key Concept Number Six

You have several special features to help with programming

- ✎ Hole machining canned cycles
- ✎ Sub-programming techniques
- ✎ Other special features
- ✎ Rotary device programming

Home

13: Hole machining canned cycles

Key concept #6
Intro to canned cycles
  Commonalities
Description of each cycle
  G80 - G81 – G73 - G83
  G82 - G84 - G84.1 - G74
  G86 - G89 - G76 - G85
  Efficiency notes
Simple example

Understanding G98 & G99
  Example (clamp between holes)
  Example (no clamp)
  Example (same R and initial)
  Example (G98 & G99 are modal)
Canned cycles and the Z axis
Using incremental mode

Home

14: Sub-programming techniques

Applications
  Repeated commands
  Repeated operations
  Control programs
  Utility programs
Related words
  Picking program numbers
  Nest levels

Examples
  Multiple operations on holes
    Using L0
  Rough & Finish contour milling
  Identical pockets
    Using incremental mode
    Using G52
  Control programs
Intro to parametric programming

Home

15: Other programming features

Optional block skip
  Another optional stop
  Trial machining
N word techniques
  Statement labels
    Changing machining order
Other G codes of interest
  G02-G03 helical motion
    Example
  G04 dwell
    Relieving tool pressure
    Machine problems
  G09/G61exact stop check
  G10 data setting
  G15-G16 polar coordinates
    Example

Other G codes (continued)
  G17/G18/G19 plane selection
  G20-G21 inch/metric
    Metric advantage
  G30 second reference position
  G50-G51 scaling
  G60 single direction positioning
    Example
  G64 normal cutting
  G68-G69 coordinate rotation
    Examples
  G94-G95 feedrate mode selection
  G50.1-G51.1 mirror image
Other M codes of interest
  M00 – M02 – M13/M14/M15
Understanding parameters

16: Rotary device programming

Introduction
  Indexers v s rotary axes
Programming indexers
Programming a rotary axis
  Letter address
  Polarity
  Zero return position
  No over-travels
  Program zero designation
  Absolute vs incremental
  G00 v s G01 motion
Approaching applications
Example program

Home

As you get deeper into the course, use this presentation to quickly review what you have already covered. If students are having trouble with a review topic, simply recall the presentation for the lesson, go to the presentation links slide for that lesson, and click on the topic in question to repeat the related presentation.