MAC – 122, CNC Turning

Orientation and Introduction

Introduction

Concept Content:

In this section you will give an introduction of yourself to your class. This is an opportunity to state your relevant experiences and credentials to teach this subject along with your personal background. This can help connecting with students. You can make a video introduction and upload it to this page as well.

This course introduces the programming, setup, and operation of CNC turning centers. Topics include programming formats, control functions, program editing, part production, and inspection. Upon completion, students should be able to manufacture simple parts using CNC turning centers.e.

Course Syllabus

Concept Goals:

Insert the student learning outcomes for the course here.

Concept Content:

This is where you will upload the syllabus. You can do this either by uploading the syllabus text here or you can upload a copy of the syllabus under the resources tab for this section. If you do upload it to the resources, please be sure to give instructions to your students to look for the syllabus there.

Course Resources

Concept Goals:

You can leave this section blank provided you uploaded the student learning outcomes to the previous section.

Concept Content:

This is where you would outline student support resources such as tutoring services, listing your office ours, contact info for support for your college's learning management system, etc. If there are documents you wish to upload, be sure to upload them to the resources tab and give instructions for the students to find the documents there.

Course Overview

Concept Goals:

1. Set up and operate manual and computerized metal-cutting machine tools in a safe and efficient manner.
2. Perform part inspection using hand-held and computerized precision measuring instruments.
3. Apply basic math, algebra, geometry, and trigonometry concepts to shop projects.
4. Develop and follow manufacturing processes and procedures.

Concept Content:

This course introduces the programming, setup, and operation of CNC turning centers. Topics include programming formats, control functions, program editing, part production, and inspection. Upon completion, students should be able to manufacture simple parts using CNC turning centers.

A computerized numerical control (CNC) machine controls the tool with a computer and is programmed with a machine code system that enables it to operate with repeatability and minimal supervision. The same principles used in operating a manual machine are used in programming a CNC machine. The main difference is that instead of using handles to position the tool to a certain location, the location is stored in the memory of the machine control. The control moves the tool to this position each time the program is run. To operate and program a CNC machine, a basic understanding of machining practices and math are necessary. It is also important to be familiar with the machine control and the placement of the keys, switches, displays, etc., that are pertinent to the operation of the machine. This course provides the basic principles necessary to program the Haas lathe. It is not intended as an in-depth study of all ranges of machine use. More training and information are necessary before attempting to program the machine.

Module	Module Learning Objectives
Module 1: Intro to CNC Turning Machines	• Identify and describe CNC turning machine types. (SLO 4) • Identify parts of CNC machines. (SLO 1) • Define what CNC machining is (SLO 1)
Module 2: CNC Safety	• Correctly identify and recognize safety symbols. (SLO 1, SLO 4)

Module 3: Coordinate Systems	• Understand what X and Z axes are and how they are used in machining. (SLO 3) • Understand how X, Y, and Z axes are defined for cartesian coordinate systems. (SLO 3) • Be able to explain the differences between cartesian and polar coordinate systems. (SLO 3)
Module 4: Machine Shop Math	• Understand and calculate problems using the Pythagorean theorem (SLO 3) • Explain the differences between sine and cosine (SLO 3) • Understand when a problem calls for sine, cosine, or tangent (SLO 3)
Module 5: Programing with Codes	• Understand how G Codes Function (SLO 4) • Understand how M Codes Function (SLO 4) • Know the basics of CNC programming with G and M Codes (SLO 4)
Module 6: Speeds and Feeds	• Calculate feeds and speeds for high speed steel and cobalt drills (SLO 3)
Module 7: Mid-Term Exam	• Demonstrate understanding of the course material thus far
Module 8: Program Formatting	• Understand the following concepts: character, word, block, positive signs, leading zeros, modal commands, preparatory functions, miscellaneous function, sequence numbers. (SLO 4) • Know how to properly set up a program structure for a CNC machine. (SLO 1, SLO 4)
Module 9: Machine Defaults and Lathe Cycles	• Understand how canned cycles work, including how to end a cycle (SLO 4).
Module 10: Rapid Position, Linear Interpolation, and Circular Interpolation Commands	• Understand the uses for rapid position, linear interpolation, and circular interpolation commands (SLO 4).
Module 11: Reference Point of Return, Work Coordinate Section, Spindle Feed Commands and Feed Commands	• Understand the uses for reference point of return, work coordinate section, spindle feed, and feed commands. (SLO 4).
Module 12: Part Inspection	• Understand various methods of inspecting CNC machined tools (SLO 2) • Be familiar with how Coordinate Measurement Machine inspections work (SLO 2)
Module 13: Turning Project Week 1	• Demonstrate the ability to accurately read a blueprint to complete a turning project (SLO 4)
Module 14: Turning Project Week 2	• Demonstrate the ability to accurately read a blueprint to complete a turning project (SLO 4)
Module 15: Final Exam	• Demonstrate understanding of the course material.

Notes/Helpful Tips

Next Steps…

Your Census assignments are REQUIRED in order to remain in the class and they MUST be completed prior to the Census Date **[insert census date here]**. **If you do not have a census date requirement, you can delete this section.**

Effective note taking is also important for not only this course, but for your career as well. Note taking is a great way to retain information. The process of taking notes can keep you alert and focused on the information being presented. It also keeps your mind engaged with what you are hearing, increasing the likelihood you will retain that information. Note taking can also allow you to better organize your thoughts on the information being discussed.

Here is a video that provides some tips for effective note taking.

Module 1 - Intro to CNC Turning Machines

1.1 Module Overview

Concept Goals:

By the end of this module, you should be able to:

- Identify and describe CNC turning machine types. (SLO 4)
- Identify parts of CNC turning machines. (SLO 1)
- Define what CNC Machining is. (SLO 1)

Concept Content:

This week we will begin our course on CNC turning. Please see module 1.2 for more detail about this week's content.

This week at a glance:

Lectures:

Intro to CNC Operations - 14 Slides

Reading:

CNC Fundamentals Handout - 20 Pages

CNC Parts from Lathe Manual - 6 Pages

Types of CNC Machines - Webpage

Hands on Activity:

HAAS Simulator Training: Activating the Machine - 11 Slides

Assignment:

Module Review Quiz - 5 Questions

1.2 Module Content Resources

Concept Content:

Lectures:

Intro to CNC Operations - 14 Slides

Reading:

CNC Fundamentals Handout - 20 Pages

CNC Parts from Lathe Manual - 6 Pages - **Note for Instructors:** Provided your school uses HAAS machines, you can use this textbook section.

Types of CNC Machines - Webpage

Hands on Activity:

HAAS Simulator Training: Activating the Machine - 11 Slides

As you take notes for this week, be sure to pay special attention to the types of CNC Machines and the layout of the parts for a CNC Machine, especially the programming buttons. The control panel can be intimidating at first but it is important to learn the function buttons and what they do.

1.3 Module Assessment/Assignment

Concept Content:

This week's assignment:

Module Review Quiz - 5 Questions

1.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

1.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

1.6 Module Wrap-Up

Concept Goals:

By the end of this module, you should be able to:

- Identify and describe CNC turning machine types. (SLO 4)
- Identify parts of CNC turning machines. (SLO 1)
- Define what CNC Machining is. (SLO 1)

Concept Content:

This week we began our course on CNC turning. We discussed the types of CNC machines as well as going over the basics of the control panel. Next week, we will go over safety with CNC machining.

This week at a glance:

Lectures:

Intro to CNC Operations - 14 Slides

Reading:

CNC Fundamentals Handout - 20 Pages

CNC Parts from Lathe Manual - 6 Pages

Types of CNC Machines - Webpage

Hands on Activity:

HAAS Simulator Training: Activating the Machine - 11 Slides

Assignment:

Module Review Quiz - 5 Questions

Module 2 - CNC Safety

2.1 Module Overview

Concept Goals:

Module Learning Objectives:

- Correctly identify and recognize safety symbols. (SLO 1, SLO 4)

Concept Content:

This week we will discuss safety with CNC machines. Please see module 2.2 for more detail.

This week at a glance:

Videos:

Basic CNC Machining Safety Tips - 6.5 Minutes

CNC Mill Safety - 7.5 Minutes

CNC Safety Lines Explained - 6.5 Minutes

Handouts:

Lathe Safety Symbols - 2 Pages

Assignment:

Module 2 Quiz - 8 Questions

2.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we will go over general safety with CNC machines. Just like manual machines, CNC machines can be very dangerous when proper safety precautions are not applied. Please view the following video for an example: CNC Accident on Camera. The following safety videos and handouts will provide information on safety practices as well as get you familiar with safety symbols. This knowledge is important if you wish to do CNC machining safely.

Videos:

Basic CNC Machining Safety Tips - 6.5 Minutes

CNC Mill Safety - 7.5 Minutes

CNC Safety Lines Explained - 6.5 Minutes

Handouts:

Lathe Safety Symbols - 2 Pages. **Note to the instructor: provided your school uses HAAS equipment, you have permission to use this handout.**

2.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week's assignment:

Module 2 Quiz -8 Questions

2.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

2.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

2.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Correctly identify and recognize safety symbols. (SLO 1, SLO 4)

Concept Content:

This week we discussed safety with CNC machines. As you can see, it is important to know how to interact with CNC machines safely in order to avoid accidents and injuries.

This week in review:

Videos:

Basic CNC Machining Safety Tips - 6.5 Minutes

CNC Mill Safety - 7.5 Minutes

CNC Safety Lines Explained - 6.5 Minutes

Handouts:

Lathe Safety Symbols - 2 Pages

Assignment:

Module 2 Quiz - 8 Questions

Module 3 - Coordinate Systems

3.1 Module Overview

Concept Goals:

By the end of this module, you should be able to:

- Understand what X and Z axes are and how they are used in machining. (SLO 3)
- Understand how X, Y, and Z axes are defined for cartesian coordinate systems. (SLO 3)
- Be able to explain the differences between cartesian and polar coordinate systems. (SLO 3)

Concept Content:

This week we will discuss CNC coordinate systems. See module 3.2 for more detail.

This week at a glance:

Reading:

Embedded in module 3.2

Presentations:

Coordinate Systems - 5 Slides

Videos:

How X, Y, and Z are defined for Cartesian Coordinate System - 3 Minutes

Cartesian vs Polar Coordinate Systems - 8 Minutes

Handouts:

Intro to CNC Lathe Coordinate System - 1 Page

Assignments:

X Diameter and Z Location Exercise - 5 Questions

3.2 Module Content Resources

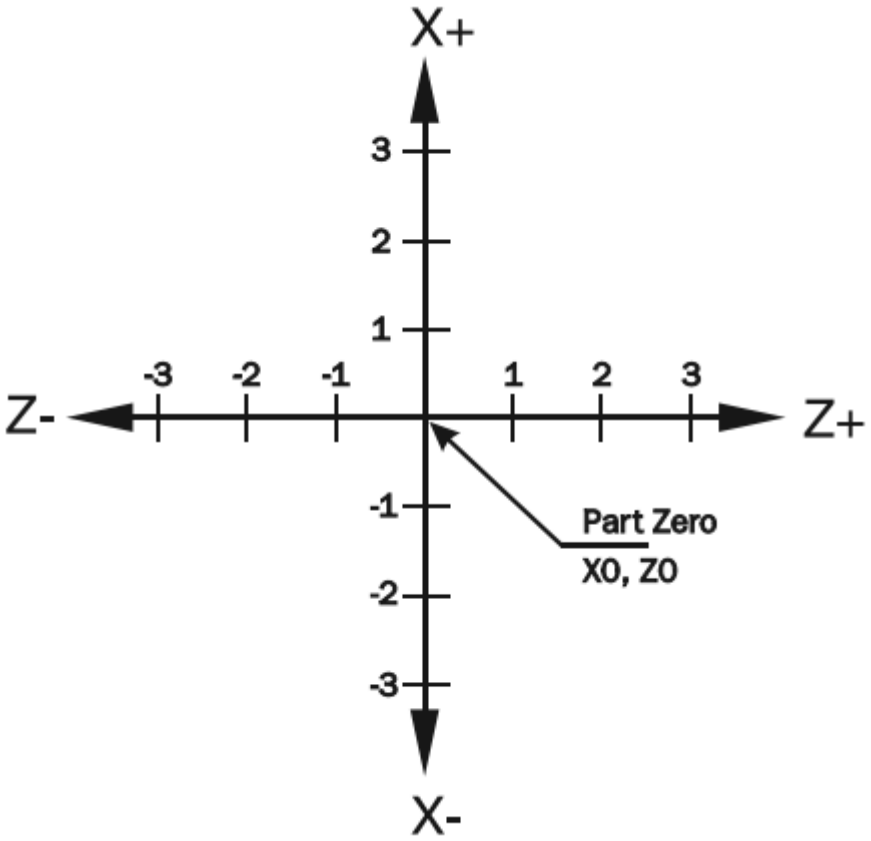
Concept Goals:

Outline the learning goals for this module here.

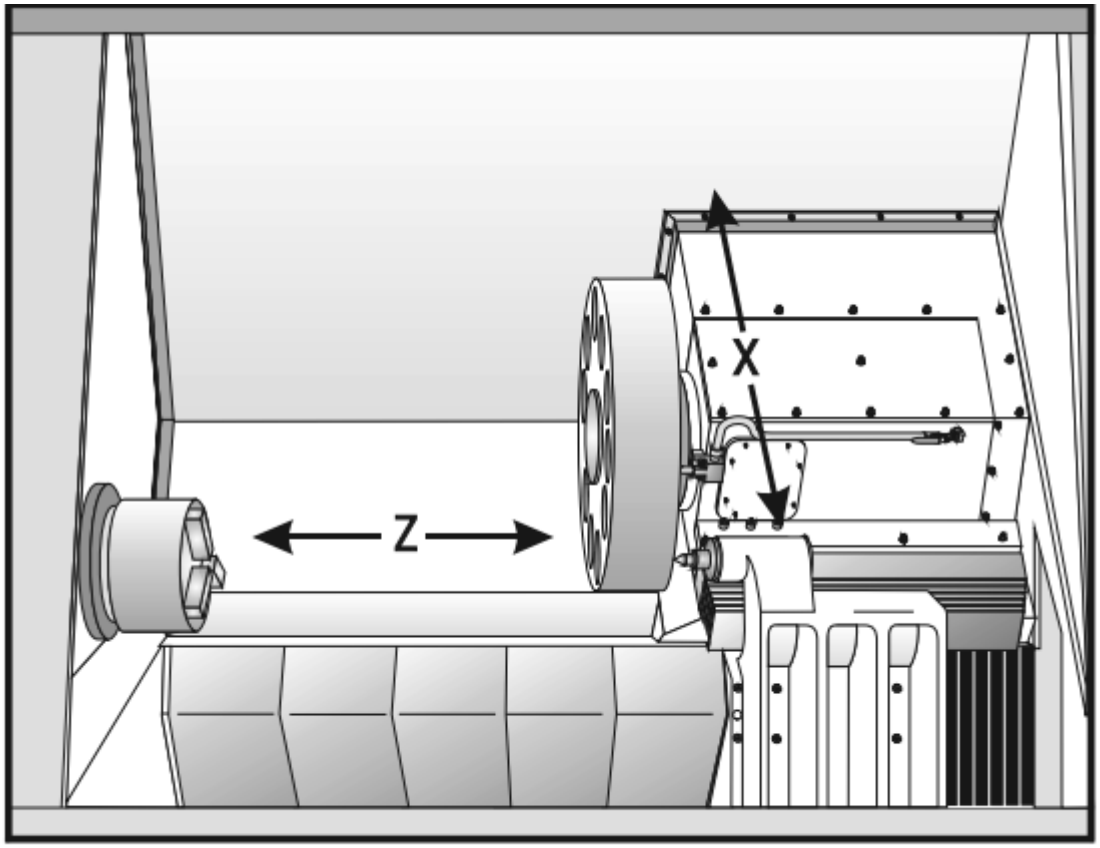
Concept Content:

This week we will discuss coordinate systems. In order to properly move the parts of a CNC machine, you must understand how the coordinate systems that govern those machines work.

The first diagram that we are concerned with is called NUMBER LINES.



This number line has a reference zero point that is called ABSOLUTE ZERO and may be placed at any point along the number line. The number line also has numbered increments on either side of absolute zero. Moving away from zero to the right are positive increments. Moving away from zero to the left are negative -" increments. The "+", or positive increments, are understood, therefore no sign is needed. The "-" sign is always needed if it's a negative value. We use positive and negative along with the increment's value to indicate its relationship to zero on the line. In the case of the previous line, if we choose to move to the third increment on the minus (-) side of zero, we would call for -3. If we choose the second increment in the plus range, we would call for 2. Our concern is with distance and direction from zero. Remember that zero may be placed at any point along the line and that once placed, one side of zero has negative increments and the other side has positive increments. The next illustration shows the two directions of travel on a lathe. To carry the number line idea a little further, imagine such a line placed along each axis of the machine. The first number line is easy to conceive as belonging to the left-to-right, or "Z", axis of the machine. If we place a similar number line along the front-to-back, or "X" axis, the increments toward the operator are the negative increments, and the increments away from the operator are the positive increments. The increments on a number line on the Haas lathe equals .0001 inches. While a line theoretically has an infinite length in either direction, the two lines placed along the X and Z axes of the machine do not have unlimited accessibility. That is to say, we are limited by the range of travel on the machine. For the Haas SL-20 for example, we have access to 8.45 inches in the X-axis and 20 inches in the Z-axis.



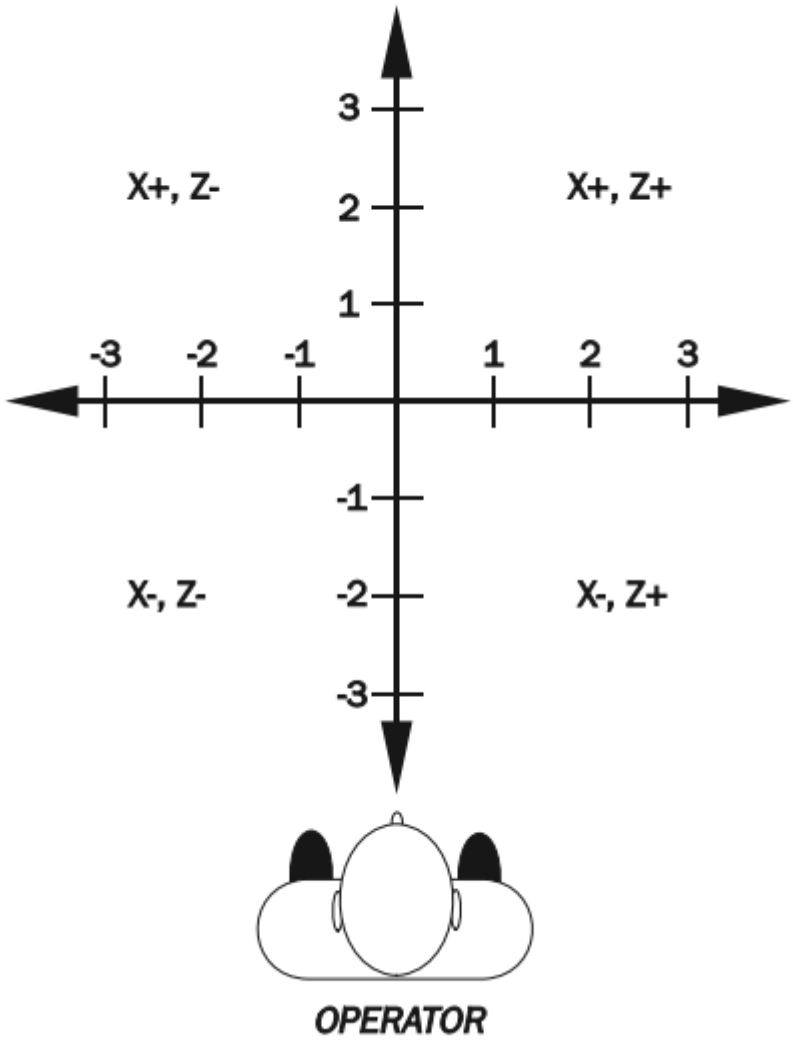
Haas Lathe X and Z axis lines

Remember, when we are moving the machine, we are concerned with positioning the turret around our workpiece. And our coordinates for positioning the turret are based off a floating zero point or our part origin.

Note: The Haas lathe use X dimensions based on the part diameter, not the radius. Thus an X move from 0. to 1.0 (X1.0) will only move the tool up .5 on the X axis.

The zero position may be placed at any point along each of the two number lines, and in fact, will probably be different for each setup of the machine. It is noteworthy to mention here that the X-axis is usually set with the machine zero position on the center line of the spindle, while the Z-axis zero is usually set at the finished right end surface of the part being machined. This places all X-axis cutting in a positive range of travel, whereas the Z-axis cutting would be in the negative range of travel.

The diagram below shows a front view of the grid as it would appear on the lathe. This view shows the X and Z axes as the operator faces the lathe. Note that at the intersection of the two lines, a common zero point is established. The four areas to the sides and above and below the lines are called “QUADRANTS” and make up the basis for what is known as rectangular coordinate programming.



QUADRANT 1 is on the Top Right at X+, Z+

QUADRANT 2 is on the Top Left at X+, Z-

QUADRANT 3 is on the Bottom Left at X-, Z-

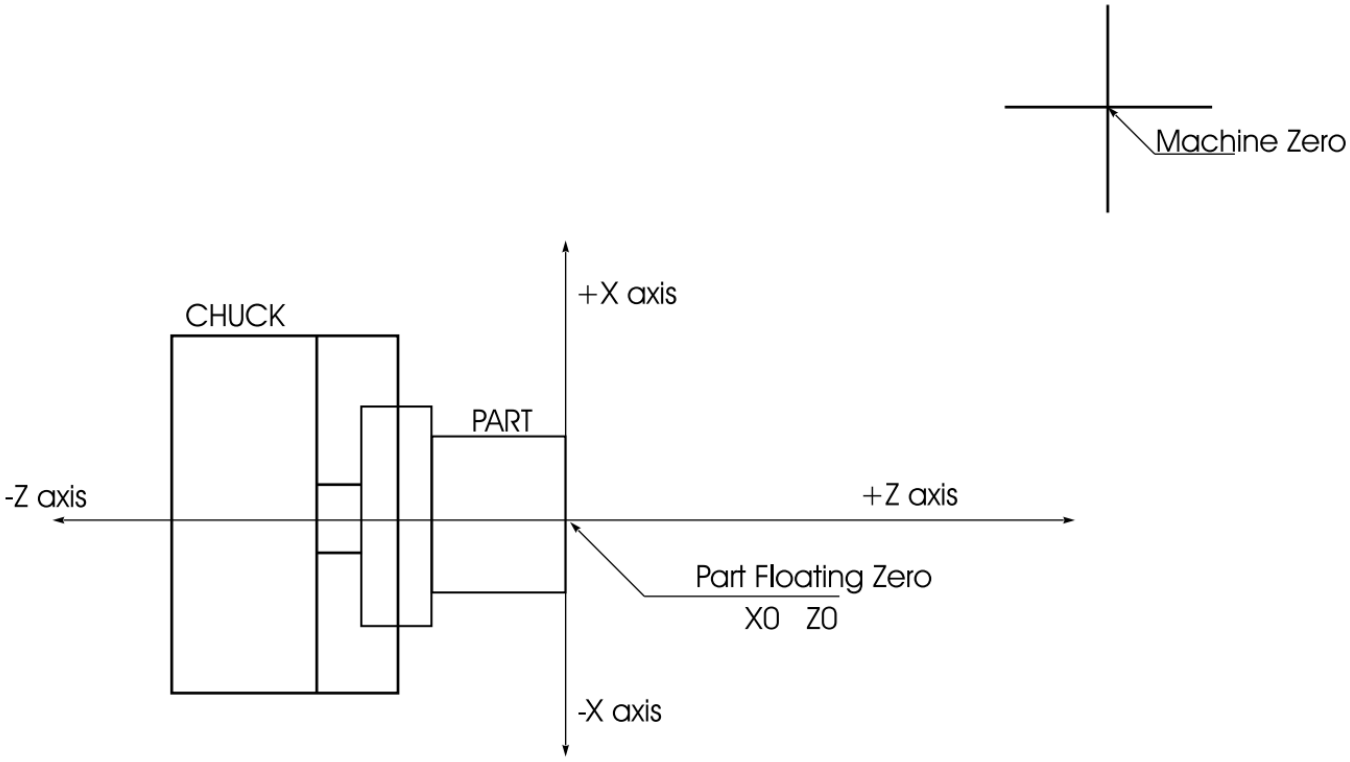
QUADRANT 4 is on the Bottom Right at X-, Z+

Whenever we set a zero somewhere on the X axis and somewhere on the Z axis, we have automatically caused an intersection of the two lines. This intersection where the two zeros come together will automatically have the four quadrants to its sides, above, and below it. How much of each quadrant is accessible is determined by where we place the zeros on the travel axes of the lathe.

For example, if we set zero exactly in the middle of the Z axis and if we set the X axis zero on the spindle center line, we have created four quadrants. For an SL-20 for example, the upper two quadrants of the Z travel is 10 inches and the X travel is 7.45 inches. The lower two quadrants will have Z travel of 10 inches and X travel of 1 inch. The Haas lathes have 1 inch of negative travel beyond the center line of the spindle.

The principle of machine home may be seen when doing a reference return of all machine axes at machine startup. A zero return (POWER UP/RESTART) is performed when you power on the machine. All axes will then move to the furthest positive locations, to the upper right of the machine, until the limit switches are reached. When this condition is satisfied, the only way to move any of the axes is in the negative direction. This is because the machine zero is set to the furthest positive point to the upper right of the machine when the machine was sent Home with a POWER UP/RESTART.

The principle of machine home may be seen when doing a reference return of all machine axes at machine startup. A zero return (POWER UP/RESTART) is performed when you power on the machine. All axes will then move to the furthest positive locations, to the upper right of the machine, until the limit switches are reached. When this condition is satisfied, the only way to move any of the axes is in the negative direction. This is because the machine zero is set to the furthest positive point to the upper right of the machine when the machine was sent Home with a POWER UP/RESTART. Machine Home is placed at the edge of each axes travel. In effect, now the positive quadrants cannot be reached, and all the X and Z moves will be found to be in the X-, Z- quadrant. It is only by setting a new location with, Tool Geometry and Work Zero Offsets somewhere within the travel of each axis that other quadrants are able to be reached.



It would not be convenient to program our parts from the machine zero, so a secondary floating zero point is established with offsets. This floating zero is referred to by either, PART ZERO or PART ORIGIN, both having the same meaning.

To create the new part zero location, each tool is manually touched off of the part being set up, on the diameter and the length. Then through a series of control keystrokes, that distance from machine zero to the part's zero is stored for X & Z axes, in tool offsets, and activated later, from the part program, when that tool is used for cutting apart.

The centerline of the lathe spindle will always be “X” zero and the “Z” zero location will “fl oat” to the face on the part that reflects most of the part length dimensions. Normally the front face is used because it’s usually easier to access for the touch-off procedures and also easier to program.

(Author Credit: HAAS Automation, INC)

Presentations:

Coordinate Systems - 5 Slides

Videos:

How X, Y, and Z are defined for Cartesian Coordinate System - 3 Minutes

Cartesian vs Polar Coordinate Systems - 8 Minutes

Handouts:

Intro to CNC Lathe Coordinate System - 1 Page

3.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

Assignment:

X Diameter and Z Location Exercise - 5 Questions

Download the exercise worksheet and upload the completed copy under quiz in the assignments tab.

3.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

3.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

3.6 Module Wrap-Up

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we discussed CNC coordinate systems. Next week, we will do a refresher on math needed for machining, particulary trigonometry.

This week in review:

Reading:

Embedded in module 3.2

Presentations:

Coordinate Systems - 5 Slides

Videos:

How X, Y, and Z are defined for Cartesian Coordinate System - 3 Minutes

Cartesian vs Polar Coordinate Systems - 8 Minutes

Handouts:

Intro to CNC Lathe Coordinate System - 1 Page

Assignments:

X Diameter and Z Location Exercise - 5 Questions

Module 4 - Machine Shop Math

4.1 Module Overview

Concept Goals:

By the end of this module, you should be able to:

- Understand and calculate problems using the Pythagorean theorem. (SLO 3)
- Explain the differences between sine and cosine. (SLO 3)
- Understand when a problem calls for sine, cosine, or tangent. (SLO 3)

Concept Content:

This week we will go over math you will need to know in order to successfully utilize a CNC machine. See module 4.2 for more detail.

This week's content:

Videos:

Intro to Pythagorean Theorem - 11 Minutes

Pythagorean Theorem: A Step by Step Guide - 14 Minutes

Basic Trigonometry: Sin, Cos, and Tan - 12 Minutes

3,4,5 Traingle - 4 Minutes

Finding Angles Using Sin, Cos, and Tan - 5 Minutes

Assignment:

Module 4 Quiz - 8 Questions

4.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we will go over math you will need to know to successfully utilize a CNC machine. This is math you will also need to know for manual machines. This week we will touch on trigonometry including the Pythagorean theorem, sine/cosine/tangent, and finding angles.

This week's content:

Videos:

Intro to Pythagorean Theorem - 11 Minutes

Pythagorean Theorem: A Step by Step Guide - 14 Minutes

Basic Trigonometry: Sin, Cos, and Tan - 12 Minutes

3,4,5 Traingle - 4 Minutes

Finding Angles Using Sin, Cos, and Tan - 5 Minutes

4.3 Moudle Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week's assignment:

Module 4 Quiz - 8 Questions

4.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted

to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

4.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

4.6 Module Wrap-Up

Concept Goals:

By the end of this module, you should be able to:

- Understand and calculate problems using the Pythagorean theorem. (SLO 3)
- Explain the differences between sine and cosine. (SLO 3)
- Understand when a problem calls for sine, cosine, or tangent. (SLO 3)

Concept Content:

This week we will go over math you will need to know in order to successfuly utilize a CNC machine. See module 4.2 for more detail.

This week's content:

Videos:

Intro to Pythagorean Theorem - 11 Minutes

Pythagorean Theorem: A Step by Step Guide - 14 Minutes

Basic Trigonometry: Sin, Cos, and Tan - 12 Minutes

3,4,5 Traingle - 4 Minutes

Finding Angles Using Sin, Cos, and Tan - 5 Minutes

Assignment:

Module 4 Quiz - 8 Questions

Module 5 - Programing with Codes

5.1 Module Overview

Concept Goals:

By the end of this module, you should be able to:

- Understand how G Codes Function (SLO 4)
- Understand how M Codes Function (SLO 4)
- Know the basics of CNC programming with G and M Codes (SLO 4)

Concept Content:

This week we will go over programming codes. See module 5.2 for more details.

This week at a glance:

Reading:

Embedded in module 5.2

Videos:

G & M Codes - 26.5 Minutes - This is a good video that covers the G and M codes discussed above.

This Weeks Assignment:

Module Review Quiz - 10 Questions

5.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we will discuss the codes used for CNC machining. These include G and M codes. Understanding how these codes work is vital when it comes to programming the machines to do what you want them to do.

What is a Part Program

The definition of a part program for any CNC consists of movements of the tool and speed changes to the spindle RPM. It also contains auxiliary command functions such as tool changes, coolant on or off commands, or external M code commands.

Tool movements consist of rapid positioning commands, straight line moves, or movement along an arc of the tool at a controlled speed.

The Haas lathe has two (2) linear axes defined as the X-axis and Z-axis. The X-axis moves the tool turret toward and away from the spindle center line, while the Z-axis moves the tool turret along the spindle axis. The machine zero position is where the tool is at the upper right corner of the work cell farthest away from the spindle axis. Motion in the X-axis will move the turret toward the spindle centerline with negative values and away from the spindle center with positive values. Motion in the Z axis will move the tool toward the spindle chuck with negative values and away from the chuck with positive values.

A program is written as a set of instructions given in the order they are to be performed. The instructions, if given in English, might look like this:

LINE #1 = SELECT CUTTING TOOL

LINE #2 = TURN THE SPINDLE ON AND SELECT THE RPM

LINE #3 = RAPID TO THE STARTING POSITION OF THE PART

LINE #4 = TURN COOLANT ON

LINE #5 = CHOOSE THE PROPER FEED RATE AND MAKE THE CUT(S)

LINE #6 = TURN THE SPINDLE AND COOLANT OFF

LINE #7 = RETURN TO CLEARANCE POSITION TO SELECT ANOTHER TOOL

BUT our machine control only understands these messages when given in machine code, also referred to as G and M code programming. Before considering the meaning and the use of codes, it is helpful to lay down a few guidelines.

Prepratory Function "G" Codes

1) G Codes come in groups. Each group of G codes will have a specific group number.

2) A G code from the same group can be replaced by another G code in the same group. By doing this the programmer establishes modes of operation. The universal rule here is that codes from the same group cannot be used more than once on the same line.

3) There are Modal G codes which once established, remain effective until replaced with another G code from the same group.

4) There are Non-Modal G codes (Group 00) which once called, are effective only in the calling block, and are immediately forgotten by the control.

The rules above govern the use of the G codes used for programming the Haas Lathe. The concept of grouping codes and the rules that apply will have to be remembered to effectively program the Haas Mill. The following is a list of Haas G codes. If there's a (Setting number) listed next to a G code, that setting will in some way relate to that G code. A single asterisk (*) indicates that it's the default G code in a group. A double asterisk (**) indicates available options.

Code	Group	Function
G00*	01	Rapid Motion (Setting 10, 56, 101)
G01	01	Linear Interpolation Motion
G01	01	Linear Interpolation Motion with Chamfering and Corner Rounding
G02	01	CW Interpolation Motion
G03	01	CCW Interpolation Motion
G04	00	Dwell (P) (P=seconds"." milliseconds)
G05 **	00	Fine Spindle Control Motion (Live Tooling)
G09	00	Exact Stop, Non-Modal
G10	00	Programmable Offset Setting
G14 **	00	Main Spindle/Sub Spindle Swap
G15**	00	Main Spindle/Sub Spindle Swap Cancel
G17**	02	Circular Motion XY Plane Selection (G02, G03) (Live Tooling)
G18*	02	Circular Motion ZX Plane Selection (G02, G03) (Setting 56)
G19**	02	Circular Motion YZ Plane Selection (G02, G03) (Live Tooling)
G20 *	06	Verify Inch Coordinate Positioning (Setting 9 will need to be INCH) (Setting 56)
G21	06	Verify Metric Coordinate Positioning (Setting 9 will need to be METRIC)
G28	00	Rapid to Machine Zero Thru Reference Point, Cancel Offsets (Fanuc)
G29	00	Move to Location Thru G29 Reference Point (Fanuc)
G31 **	00	Feed Until Skip Function
G32	01	Thread Cutting Path, Modal
G40 *	07	Tool Nose Compensation Cancel (Setting 56)
G41	07	Tool Nose Compensation Left (Setting 43, 44, 58)
G42	07	Tool Nose Compensation Right (Setting 43, 44, 58) Code Group Function

G50	11	Spindle Maximum RPM Limit
G51	11	Rapid to Machine Zero, Cancel Offsets (YASNAC)
G52	00	Work Offset Positioning Coordinate (Setting 33, YASNAC)
G52	00	Global Work Coordinate System Shift (Setting 33, FANUC)
G53	00	Machine Zero Positioning Coordinate Shift, Non-Modal
G54*	12	Work Offset Positioning Coordinate #1 (Setting 56)
G55	12	Work Offset Positioning Coordinate #2
G56	12	Work Offset Positioning Coordinate #3
G57	12	Work Offset Positioning Coordinate #4
G58	12	Work Offset Positioning Coordinate #5
G59	12	Work Offset Positioning Coordinate #6
G61	13	Exact Stop, Modal
G64*	13	Exact Stop G61 Cancel (Setting 56)
G65 **	00	Macro Sub-Routine Call
G70	00	Finishing Cycle
G71	00	O.D./I.D. Stock Removal Cycle (Setting 72, 73)
G72	00	Face Stock Removal Cycle (Setting 72, 73)
G73	00	Irregular Path Stock Removal Cycle
G74	00	Face Grooving Cycle or High Speed Peck Drill Cycle (Setting 22)
G75	00	O.D./I.D. Grooving Cycle (Setting 22)
G76	00	Threading Cycle, Multiple Pass (Setting 86, 95 ,96, 99)
G77 **	00	Flatting Cycle (Live Tooling)
G80*	09	Canned Cycle Cancel (Setting 56)
G81	09	Drill Canned Cycle
G82	09	Spot Drill / Counterbore Canned Cycle
G83	09	Peck Drill Deep Hole Canned Cycle (Setting 22, 52)
G84	09	Tapping Canned Cycle
G85	09	Bore in~Bore out Canned Cycle
G86	09	Bore in~Stop~Rapid out Canned Cycle
G87	09	Bore in~Manual Retract Canned Cycle
G88	09	Bore~Dwell~Manual Retract Canned Cycle
G89	09	Bore~Dwell~Bore out Canned Cycle
G90	01	O.D./I.D. Turning, Modal
G92	01	Threading Cycle, Modal (Setting 95, 96)
G94	01	End Facing Cycle, Modal
G95 **	09	End Face Rigid Tap (Live Tooling)
G96	12	Constant Surface Speed, CSS On
G97	12	Constant Non-Varying Spindle Speed, CSS Off (Setting 56)
G98	05	Feed Per Minute
G99*	05	Feed Per Revolution (Setting 56)
G100	00	Mirror Image G101 Cancel
G101	00	Mirror Image (Setting 45, 47) Code Group Function
G102	00	Programmable Output to RS-232
G103	00	Limit Block Lookahead (P0-P15 max. for number control looks ahead)
G105	00	Servo Bar Command
G110	12	Work Offset Positioning Coordinate #7
G111	12	Work Offset Positioning Coordinate #8
G112**	00	Live Tooling Cartesian to Positioning Polar Transformation
G113**	00	Live Tooling Cartesian to Positioning Polar Transformation Cancel
G114	12	Work Offset Positioning Coordinate #11

G115 12 Work Offset Positioning Coordinate #12
G116 12 Work Offset Positioning Coordinate #13
G117 12 Work Offset Positioning Coordinate #14
G118 12 Work Offset Positioning Coordinate #15
G119 12 Work Offset Positioning Coordinate #16
G120 12 Work Offset Positioning Coordinate #17
G121 12 Work Offset Positioning Coordinate #18
G122 12 Work Offset Positioning Coordinate #19
G123 12 Work Offset Positioning Coordinate #20
G124 12 Work Offset Positioning Coordinate #21
G125 12 Work Offset Positioning Coordinate #22
G126 12 Work Offset Positioning Coordinate #23
G127 12 Work Offset Positioning Coordinate #24
G128 12 Work Offset Positioning Coordinate #25
G129 12 Work Offset Positioning Coordinate #26
G154 12 Select Work Offset Positioning Coordinate P1-99 (P)
G159** 00 Background Pickup / Part Return
G160** 00 APL Axis Command ON
G161** 00 APL Axis Command OFF
G184** 00 Reverse Tapping Canned Cycle
G186 00 Live Tooling Reverse Rigid Tapping
G187** 00 Accuracy Control for High-Speed Machining (Setting 85)
G194 00 Sub-Spindle / Tapping Canned Cycle
G195** 00 Live Tooling Radial Tapping
G196** 00 Live Tooling Radial Tapping Reverse
G200 00 Index on the fly

*Defaults
** Options

Each **G** code is a part of a group of **G** codes. The Group 0 codes are nonmodal; that is, they specify a function applicable to this block only and do not affect other blocks. The other groups are modal and the specification of one code in the group cancels the previous code applicable from that group. A modal **G** code applies to all subsequent blocks so those blocks do not need to re-specify the same G code.
There is also one case where the Group 01 **G** codes will cancel the Group 9 (canned cycles) codes. If a canned cycle is active (G81 through G89), the use of G00 or G01 will cancel the canned cycle

Often Used Preparatory G Codes

G00 Rapid traverse motion; Used for non-cutting moves of the machine in positioning quick to a location to be machined, or rapid away after program cuts have been performed. Maximum rapid motion (I.P.M.) of a Haas machine will vary on machine model.

G01 Linear interpolation motion; Used for actual machining and metal removal. Governed by a programmed feed rate in inches (or mm) per revolution (G99).

Maximum feed rate (I.P.M.) of a Haas machine will vary on machine model.
(Inch Per Minute = R.P.M. x Inch Per Revolution).

G02 Circular Interpolation - Clockwise.
G03 Circular Interpolation - Counterclockwise.
G28 Machine Home (Rapid traverse)
G40 Tool Nose Compensation CANCEL
G41 Tool Nose Compensation LEFT of the programmed path
G42 Tool Nose Compensation RIGHT of the programmed path
G50 Spindle Speed Maximum RPM limit
G70 Finishing Cycle
G71 O.D./I.D. Stock Removal Cycle
G72 End Face Stock Removal Cycle
G76 O.D./I.D. Thread Cutting Cycle
G80 Cancel Canned Cycle
G81 Drill Canned Cycle
G82 Spot Drill Canned Cycle
G83 Peck Drill Canned Cycle
G84 Tapping Canned Cycle
G96 Constant Surface Speed On
G97 Constant Surface Speed Cancel
G98 Feed Per Minute
G99 Feed Per Revolution

Miscellaneous Function Codes "M" Codes

All M codes are activated or cause an action to occur after everything else on a block has been completed. Only one M code is allowed per block in a program.

If there is a (Setting number) listed next to an M code, that setting will in some way relate to that M code.

The following list is a summary of Haas M codes. A double asterisk (**) indicates options available.

M00 Program Stop (Setting 42)

M01 Optional Program Stop (Setting 17)

M02 Program End

M03 Spindle On Forward (S) (Setting 144)

M04 Spindle On Reverse (S) (Setting 144)

M05 Spindle Stop

M08 Coolant On (Setting 32)

M09	Coolant Off
M10	Chuck Clamp (Setting 92)
M11	Chuck Unclamp (Setting 92)
M12**	Auto Air Jet On
M13 **	Auto Air Jet Off
M14**	Main Spindle Clamp
M15**	Main Spindle Unclamp
M17	Rotate Turret Forward (T) (Setting 97)
M18	Rotate Turret Reverse (T) (Setting 97)
M19**	Orient Spindle (R, P)
M21**	Tailstock Advance (Setting 93, 94, 106, 107, 121, 145)
M22**	Tailstock Retract (Setting 105)
M23	Angle Out of Thread On (Setting 95, 96)
M24	Angle Out of Thread Off
M30	Program End and Reset (Setting 2, 39, 56, 83)
M31 **	Chip Auger Forward (Setting 114, 115)
M33	Chip Auger Stop
M36**	Parts Catcher On
M37**	Parts Catcher Off
M41	Spindle Low Gear Override
M42	Spindle High Gear Override
M43	Turret Unlock (For Service Use Only)
M44	Turret Lock (For Service Use Only)
M51-M58	Optional User M Code Set
M59	Output Relay Set
M61-M68	Optional User M Code Clear
M69	Output Relay Clear
M76	Program Displays Active

M77	Program Displays Inactive
M78	Alarm if skip signal found
M79 A	larm if skip signal not found
M85**	Automatic Door Open (Setting 131, 51)
M86**	Automatic Door Close (Setting 131, 51)
M88**	High Pressure Coolant On (Setting 32)
M89**	High Pressure Coolant Off
M93**	Axis Position Capture Start
M94**	Axis Position Capture Stop
M95	Sleep Mode (hh:mm)
M96	Jump If No Signal (P,Q)
M97	Local Sub-Routine Call (P,L)
M98	Sub-Program Call (P,L)
M99	Sub-Program/Routine Return Or Loop (Setting 118)
M110**	Tailstock Chuck Clamp (Setting 122)
M111**	Tailstock Chuck Unclamp (Setting 122)
M119**	Sub-Spindle Orient (P,R)
M121-M128	Optional User M Code Interface with M-Fin Signal
M133**	Live Tool Drive Forward (P)
M134**	Live Tool Drive Reverse (P)
M135**	Live Tool Drive Stop
M143**	Sub-Spindle Forward (P)
M144**	Sub-Spindle Reverse (P)
M145**	Sub-Spindle Stop
M154**	C-Axis Engage (Setting 102)
M155**	C-Axis Disengage

** Options

Often Used Miscellaneous M Codes

M00 The M00 code is used for a Program Stop command on the machine. It stops the spindle, turns the off coolant and stops lookahead processing. Pressing CYCLE START will continue the program on the next block in the program. You You will need to command an M code to turn the spindle back on and the coolant back on in the program.

M01 The M01 code is used for an Optional Program Stop command. Pressing the OPT STOP key on the control panel signals the machine to perform a stop command when the control reads an M01 command in a program. It will then perform like an M00. Pressing CYCLE START will continue the program on the next block in a program. You will need to command an M code to turn the spindle back on and the coolant back on in the program.

M03 Starts the Spindle FORWARD. Must have a spindle speed defined.
M04 Starts the Spindle REVERSE. Must have a spindle speed defined.
M05 STOPS the Spindle
M08 Coolant ON command
M09 Coolant OFF command
M30 Program End and Reset to the beginning of program
M97 Local subroutine Call
M98 Subprogram Call
M99 Subprogram Return (M98) or S

Alphabetical Address Codes

B LINEAR B-AXIS MOTION (Tailstock) (Setting 93, 94, 105, 106, 107, 121, 145)
The B address character is currently reserved for the tailstock. It is used to specify the absolute position or motion of the tailstock along the B axis. B-axis commands in the negative direction move the tailstock toward the spindle, and a B-axis command in the positive direction moves it away from the spindle.

F FEED RATE (Setting 19, 77)
The F address character is used to select the feed rate applied to any interpolating G codes or canned cycles. This command value is in inches per revolution or mm per revolution. Inches per revolution (G99) is the default. But it can be changed to units/minute with G98. Traditionally, the F code was capable of only 4 fractional position accuracy; but on this control, you can specify F to six place accuracy. Code E and F are equivalent.

G PREPARATORY FUNCTIONS (G codes)
The G address character is used to specify the type of operation to occur in the block containing the G code. The G is followed by a two or three-digit number between 0 and 187. Each G code defined in this control is part of a group of G codes. The Group 0 codes are non-modal; that is, they specify a function applicable to this block only and do not affect other blocks. The other groups are modal and the specification of one code in the group cancels the previous code applicable from that group. A modal G code applies to all subsequent blocks so those blocks do not need to re-specify the same G code. More than one G code can be placed in a block in order to specify all of the setup conditions for an operation.

I CIRCULAR INTERPOLATION / CANNED CYCLE DATA
The I address character is used to specify data used for some canned cycles and circular motions. It is either in inches with four fractional positions or mm with three fractional positions.

J CANNED CYCLE DATA
The J address character is used to specify data used for some canned cycles.

K CIRCULAR INTERPOLATION / CANNED CYCLE DATA
The K address character is used to specify data used for some canned cycles and circular motions. It is formatted just like the I data.

L LOOP COUNT TO REPEAT A COMMAND LINE
The L address character is used to specify a repetition count for some canned cycles and auxiliary functions.

M MISCELLANEOUS FUNCTIONS (M Codes)
The M address character is used to specify an M code for a block. These codes are used to control miscellaneous machine functions. Note that only one M code is allowed per block of the CNC program and all M codes are performed at the end of the block.

N LINE/BLOCK NUMBER
The N address character is entirely optional. The only function of an N number is to identify and locate a certain block or line within a program.

O PROGRAM NUMBER (PROGRAM name in parenthesis)
The O address character is used to identify a program. It is followed by a number between 0 and 99999. A program saved in memory always has an Onnnnn identification in the first block. Altering the Onnnnn in the first block causes the program to be renumbered. If you enter a program name (Name) between parenthesis in the first three lines of a program, that program name will also be seen in your list of programs. You can have up to 500 program numbers (200 programs on an older machine) in your List of Programs. You can delete a program number from the LIST PROG display, by cursor-selecting the program, and pressing the ERASE PROG key. You can also delete a program in the advanced editor using the menu item DELETE PROGRAM FROM LIST.

P DELAY OF TIME OR M97 SEQUENCE NUMBER CALL / M98 PROGRAM NUMBER CALL OR LIVE TOOLING SPINDLE SPEED

The P address character is used as a delay of time in seconds for a dwell command or as a P number to search for a sequence number in a local subroutine call, or as a P number to search for a program number in your list of programs for a subprogram call. P is also defined with Q, and is used in canned cycles G70, G71, G72, and G73 to specify the starting block number of the part geometry defined for machining with these cycles.

Q CANNED CYCLE DATA

The Q address character is used in a G83 canned cycle and is a positive number for the peck amount. Q is also defined with P, in the canned cycles G70, G71, G72, and G73 to specify the ending block number of the part geometry defined for machining with these cycles.

R CIRCULAR INTERPOLATION / CANNED CYCLE DATA (Setting 52)

The R address character is used in canned cycles and circular interpolation. It is usually used to define the reference plane for canned cycles.

S SPINDLE SPEED COMMAND (Setting 20, 144)

The S address character is used to specify the spindle speed. The S command does not turn the spindle on or off; it only sets the desired speed. By default, S specifies RPM. When used with G96, S specifies surface feet per minute.

T TOOL SELECTION CODE (Setting 42, 87, 97)

The T number address calls a tool and an offset when initiating a tool change. Txxyy is the T command format. The first two digits (xx) specify the turret position and are used to call up a tool that's between 1 and the number of tool turret positions on the machine. The second two digits (yy) calls up a tool geometry/wear offset that is going to be used for that tool and will be a number between 1-50.

U INCREMENTAL X-AXIS MOTION

The U address character is used to specify motion for the X-axis. It specifies an incremental position or distance along the X-axis relative to the current machine position. It is defined either in inches with four fractional positions or in mm with three fractional positions.

W INCREMENTAL Z AXIS MOTION

The W address character is used to specify motion for Z-axis. It specifies an incremental position or distance along the Z-axis relative to the current machine position. It is defined either in inches with four fractional positions or in mm with three fractional positions.

X ABSOLUTE X AXIS MOTION (Setting 45)

The X address character is used to specify absolute motion for the X-axis. It specifies a position or distance along the X-axis. It is either in inches with four fractional positions or mm with three fractional positions.

Z ABSOLUTE Z AXIS MOTION (Setting 47)
The Z address character is used to specify absolute motion for the Z-axis. It specifies a position or distance along the Z-axis. It is either in inches with four fractional positions or mm with three fractional positions.

Author Credit: HAAS Automation, INC

Videos:

G & M Codes - 26.5 Minutes - This is a good video that covers the G and M codes discussed above.

5.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This Weeks Assignment:

Module Review Quiz - 10 Questions

5.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

5.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to

a least one other student's answer to foster discussion.

5.6 Module Wrap-Up

Concept Goals:

Module learning objectives:

- Understand how G Codes Function (SLO 4)
- Understand how M Codes Function (SLO 4)
- Know the basics of CNC programming with G and M Codes (SLO 4)

Concept Content:

This week we discussed programming codes. There was a lot of information provided this week and this is a dense subject, so be sure to review the material as we move forward in the course. Next week, we will look at speeds and feeds.

This week in review:

Reading:

Embedded in module 5.2

Videos:

G & M Codes - 26.5 Minutes - This is a good video that covers the G and M codes discussed above.

This Weeks Assignment:

Module Review Quiz - 10 Questions

Module 6 - Speeds and Feeds

6.1 Module Overview

Concept Goals:

By the end of this module, you should be able to:

- Calculate feeds and speeds for high speed steel and cobalt drills (SLO 3)

Concept Content:

This week we will discuss feeds and speeds. See module 6.2 for more details.

This week at a glance:

Videos:

Metal Lathe Tutorial 21: Speeds and Feeds - 21 Minutes

Finding the Spindle Speed on a Lathe - 6 Minutes

Handouts:

Feed Rate per Rev for Drills - 1 Page

HSS Speed Steel & Cobalt Drills Speed and Feed Recommendations - 2 Pages

Assignment:

Feeds and Speeds for HSS Drills Worksheet - 1 Page

6.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we will cover speeds and feeds. It is important to know how to calculate these in order to properly turn a part in the machine.

Videos:

Metal Lathe Tutorial 21: Speeds and Feelds - 21 Minutes

Finding the Spindle Speed on a Lathe - 6 Minutes

Handouts:

Feed Rate per Rev for Drills - 1 Page

HSS Speed Steel & Cobalt Drills Speed and Feed Recommendations - 2 Pages

6.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week's assignment:

Feeds and Speeds for HSS Drills Worksheet - Download the worksheet and use the High Speed Steel and Cobalt Drills Feed Recommendation chart provided in module 6.2 to complete it. Upload the completed worksheet under quiz in the assignments tab.

6.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

6.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

6.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Calculate feeds and speeds for high speed steel and cobalt drills (SLO 3)

Concept Content:

This week we discussed feeds and speeds. Next week, we will have our mid-term exam.

This week at a glance:

Videos:

Metal Lathe Tutorial 21: Speeds and Feeds - 21 Minutes

Finding the Spindle Speed on a Lathe - 6 Minutes

Handouts:

Feed Rate per Rev for Drills - 1 Page

HSS Speed Steel & Cobalt Drills Speed and Feed Recommendations - 2 Pages

Assignment:

Feeds and Speeds for HSS Drills Worksheet - 1 Page

Module 7 - Mid-Term Exam

7.1 Module Overview

Concept Goals:

Demonstrate understanding of the course material thus far.

Concept Content:

This week is our mid-term exam. You will find the exam in the assignments tab, click on test to open up the exam.

7.2 Module Wrap-Up

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

Thank you for taking this week's mid-term exam. Next week we will start the second half of the course. Please let me know if you have any questions or concerns about your performance in the class thus far.

Module 8 - Program Formatting

8.1 Module Overview

Concept Goals:

By the end of this module, you should be able to:

- Understand the following concepts: character, word, block, positive signs, leading zeros, modal commands, preparatory functions, miscellaneous function, sequence numbers. (SLO 4)
- Know how to properly set up a program structure for a CNC machine. (SLO 1, SLO 4)

Concept Content:

This week, we will go over program formatting. See module 8.2 for more detail.

This week at a glance:

Reading:

Embedded in module 8.2

Assignments:

HAAS Simulator Training

HAAS Productivity Exercise - 2 Pages

8.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we will go over program formatting. In order to successfuly code your CNC machine, you must understand proper formatting for programs you enter into it. Without knowledge of how it works, you will be unable to get the CNC machine to function properly. This week's readings will cover how to properly format your programs.

There is no positional requirement for the G and M codes. They may be placed in any order within a line of program, which is also known as a block. Each individual can format their programs in many
different ways. But, program format or program style is an important part of CNC machining. There are some program command formats that can be moved around, some commands need to be a certain way, and there are some standard program rules that are just good to follow. The point is that a programmer needs to have an organized program format that's consistent and efficient so that any CNC machinist in your shop can understand it.

Rules to Consider

Program X and Z in alphabetical order on any block. The machine will read Z or X in any order, but we want to be consistent. When X and Z are both on a command line in a program, they should be listed together and in order. Write X first, and Z second.
You can put G and M codes anywhere on a line of code. But, in the beginning, when N/C programming
was being developed G codes had to be at the beginning of a program line, and M codes had to be at the end. And this rule, a lot of people still follow, and is a good standard to continue.
Some CNC machines allow you to write more the one M code per line of code and some won't.
On the Haas, only one M code may be programmed per block and all M codes are activated or cause an action to occur after everything else on the line has been executed.
Program format is a series and sequence of commands that a machine may accept and execute.
Program format is the order in which the machine code is listed in a program that consists of command
words. Command words begin with a single letter and then numbers for each word. If it has a plus (+) value, no sign is needed. If it has a minus value, it must be entered with a minus (-) sign. If a command word is only a number and not a value, then no sign or decimal point is entered with that command. Program format defines the "language of the machine tool."

...
G82 Z-0.2 P0.3 R0.1 F0.003 ;
G80 G00 Z1. M09 ;
G28 ;
MO1 ;
;
N4 (Drill .312 Dia. x 1.5 Depth) ;

G28 ;
T404 (5/16 DIA. DRILL) ;
G97 S2400 M03 ;
G54 G00 X0. Z1. M08 ;
G83 Z-1.5 Q0.3 R0.1 F0.006 ;
G80 G00 Z1. M09 ;
G28 ;
MO1 ;
…
…

Example of a program's first few lines:

THE FIRST LINE or block of a program, should be a return to machine zero (using G28 or G51 codes). Any tool change should be after a return to machine zero or a tool change location. Although this is not necessary it is a good safety measure.

THE SECOND LINE of code should apply to any appropriate tool selections and tool geometry offsets or tool shifts.

THE THIRD LINE may optionally contain a spindle speed maximum for the tool being used.

THE FOURTH LINE or block should cancel any constant surface speed mode (G97). And it should specify a constant spindle speed command (S____) along with a spindle ON clockwise command (M03).

THE FIFTH LINE should contain a work offset (G54), a preparatory code (G00) for rapid command with an X and Z location for positioning the turret, and turn on the coolant (M08).

THE SIXTH LINE may choose to, optionally specify a Constant Surface Speed with (G96) and a surface feet per minute (SFM) defined with a (S____) command.

An example of the program’s startup lines might look like this:

With Constant
Surface Speed

N11 G28 ; (All axes move to the machine zero position)
N12 T101; (Tool 1, tool offset 01)
N13 G50 S2800 ; (Maximum spindle speed set to 2800 rpm)
N14 G97 S650 M03 ; (Turns the spindle on (M03) at a constant rpm (G97) of 650 rpm)
N15 G54 G00 X1.85 Z1. M08; (Use G54 work offset, rapid move to coordinates, coolant on)

N16 G96 S315 ; (Constant surface speed of 315 feet per minute regardless of part diameter)

Without Constant Surface Speed

N21 G28 ; (All axes move to the machine zero position)
N22 T101 ; (Tool 1, tool offset 01)
N23 G97 S1600 M03 ; (Turn spindle on (M03) to constant speed of 1600 rpm)
N24 G54 G00 X0. Z1. M08; Use G54 work offset, rapid move to coordinates, coolant on)

All the necessary codes for each operation are listed in the following sections. This tool startup format
is a good example and defines a commonly used program style.

Definitions Within the Format

1. **CHARACTER**: A single alphanumeric character value or the “+” and “-” sign.

2. **WORD**: A series of characters defining a single function such as G codes, M codes, and “X” axis moves, or “F” feed rate. A letter is the first character of a word for each of the different commands. There may be a distance and direction defined for a word in a program. The distance and direction in a word are made up of a value, with a plus (+) or minus (-) sign. A plus (+) value is recognized if no sign is given in a word.

3. **BLOCK**: Series of words defining a single instruction. An instruction may consist of a single linear motion, a circular motion, or canned cycle, plus additional information such as a feed rate or miscellaneous commands (M-codes).

4. **POSITIVE SIGNS**: If the value following an address letter command such as A, B, C, I, J, K, R, U, V, W, X, Y, and Z, is positive, the plus sign need not be programmed in.
If it has a minus value it must be programmed in with a minus (-) sign.

5. **LEADING ZEROS**: If the digits proceeding a number are zero, they need not be programmed in. The Haas control will automatically enter the leading zeros.
EXAMPLE: G0 for G00 and M1 for M01,
Trailing zeros must be programmed: M30 not M3, G70 not G7.

6. **MODAL COMMANDS**: Codes that are active for more than the line in which they are issued

are called MODAL commands. Rapid traverse, feed rate moves, and canned cycles are all examples of modal commands. A NON-MODAL command which once called, are effective only in the calling block, and then immediately forgotten by the control.

7. **PREPARATORY FUNCTIONS**: “G” codes use the information contained on the line to make the machine tool does specific operations, such as :
1.) Move the tool at rapid traverse.
2.) Move the tool at a feed rate along a straight line.
3.) Move the tool along an arc at a feed rate in a clockwise direction.
4.) Move the tool along an arc at a feed rate in a counterclockwise direction.
5.) Move the tool thru a series of repetitive operations controlled by “fixed cycles” such as spot drilling, drilling, boring, and tapping.

8. **MISCELLANEOUS FUNCTIONS**: “M” codes are effective or cause an action to occur at the end of the block and only one M code is allowed in each block of a program.

9. **SEQUENCE NUMBERS**: N1 thru N99999 in a program are only used to locate and identify a line or block and its relative position within a CNC program. A program can be with or without SEQUENCE NUMBERS. The only function of SEQUENCE NUMBERS is to locate a certain block or line within a CNC program.

The machining cycles G70, G71, G72, and G73 require the use of sequence numbers to call up specified blocks in a program that in it define the part geometry to rough and finish

Program Start Up Lines

To Rapid all axes to machine home
G28

G28-Rapids all axes to machine home.
Or send specified axes to the machine home
G28 U0. W0. G28 - Rapids any specified axes to machine home.
U0. - U0. along with G28 will rapid X-axis home.
W0. - W0. along with G28 will rapid Z-axis home.
B0. - B0. along with G28 will rapid tailstock home.

Send turrent to a tool change location defined from the machine home
G53 G00 X-3.0 Z-4.0 T0 G53 - Is non-modal command and ignores any active work offset for this line only.
G00 - Rapid position command.
X__ - X-axis minus location from machine home.
Z__ - Z axis minus location from machine home.

T0 - Cancels any active tool offsets.

After sending the turret to a safe tool change location, perform a tool change

T101 - Txxyy is a tool change command for a tool number “xx” with the “yy” that defines the tool offset number that’ll be used with a tool.

G50 S3200 G50 - With a spindle speed defines a maximum spindle RPM to limit how high the calculated
S3200 - G96 spindle speed will go.

G97 S1025 M03 G97 - Cancels constant surface speed along a spindle speed, defines a constant speed
S1025 - Designates a spindle speed.
M03 - Turns spindle on forward.

G54 G00 X1.6 Z0.1 M08 G54 - Selects work coordinate offset #1
G00 - Rapid position command.
X__ - Move the axis to the initial X position.
Z__ - Move the axis to the initial Z position.
M08 - Turns on coolant.

G96 S1200 G96 - This command turns on Constant Surface Speed when needed. If multiple diameters are being machined with a single-pointed tool, you should then use a G96 to vary the RPM as it goes up and down the diameters on your part to maintain the surface speed.

S325 - Designates surface speed (not spindle speed) when defined with a G96 that’ll be used, by the machine, to calculate the spindle RPM

Program Ending Lines

G00 Z1. M09 G00 - Rapid position command.
Z1. - Moves Z-axis 1.0 from part zero for a return to machine home or tool change location.
G97 S1025 M09 G97 - Cancels constant surface speed along with a spindle speed defines a constant speed.
S1025 - Designates the spindle speed.
M09 - Turns off coolant.

To Rapid all axes to machine home

G28 G28 - Rapids all axes to machine home.

Send specified axes to the machine home

G28 U0. W0. G28 - Rapids any specified axes to machine home.
U0. - U0. along with G28 will rapid only X axis home.
W0.- W0. along with G28 will rapid only Z axis home.
B0. - B0. along with G28 will rapid tailstock home.

Send turrent to a tool change location defined from the machine home

G53 G00 X-3.0 Z-4.0 T0 G53 - Is non-modal command and ignores any active work offset for this line only.
G00 - Rapid position command.
X__ - X-axis minus location from machine home.
Z__ - Z axis minus location from machine home.
T0 T0 - Cancels any active tool offsets.

.....

M30 M30 - Signals End of Program and Reset, to the control. The program will reset to the first block of the program and stop. It also stops the spindle, turns off the coolant, and cancels tool length offsets. It also makes the control ready for the next cycle.

Safe Start Up Line

G18 G20 G40 G54 G80 G97 G99

Do you need a safe startup line to be sure all the commands are canceled before starting a program? Many programs have a G code default line (the CAD/CAM programming system may output a default line) at the beginning. To ensure the machine control is in a safe start condition,
use these conditions on a Haas control, to help you decide if you need a safe start line at the beginning of a program.

G18 Circular Motion ZX Plane Selection - The G18 is the default condition on a Haas lathe, and is the only one available in the X and Z axis to do an arc. If you try an arc in the G17 XY plane, or the G19 YZ plane, the machine would stop and give you an alarm. There is no need to program a G18, it is already active when you power up.

G20 Inch / G21 Metric Dimensioning - The G codes G20 and G21 are used to select between inch and metric. On the Haas control, the G20 (inch) and G21 (mm) codes are used to make sure the inch/metric setting (Setting 9) is set correctly for that program.

G54 Work Offset Command - Work offsets on a CNC lathe are not used like they are on a CNC mill. Many lathe users don't even have values in their work offsets, because all the offsets that are needed on a lathe, for most users, are entered in the Tool Geometry display. The only time someone might use a work offset on a lathe is to shift all the tools in the Z-axis the same distance.

Example: Let's say you want to move all of your tools -.015 in the Z axis, to make the parts that much shorter. Instead of changing all of the tool offsets, you could shift all of the tools in -0.015 by changing the work offset. Just be sure to take this offset shift out when you're done using it.

WORK ZERO OFFSET

G CODE	(X)	(Z)	(B)
G52	0.	0.	0.
G54	0.	-0.015	0.
G55	0.	0.	0.

Another main reason to use a work offset on a lathe is when you're touching off your tools on a tool probe. The X-axis value is usually always good to the center of the spindle using the value touched off in the X-axis on a tool probe along with the value in Setting 59 or 60. The problem is that the distance from the edge of the tool probe in the Z-axis to the face of the part. Compensate for that difference in the Work Zero Offset, which has the extra distance from the edge of the tool probe to the face of the part, in the Z axis.

Be aware that even if you're not using a work offset command in a program, G54 is still active as a default G code. This is why you see G54 in the program example of this book, as a reminder that the work offset is active whether you program it in or not. If you never use work offsets, you may choose to leave this G54 command out.

G40 Cancels Tool Nose Compensation - You may see a G40 at the beginning of every program. Sometimes it is put at the beginning of every tool, to be sure cutter compensation is off before you start a program.

You should always cancel cutter compensation (G40) when you're done using it. If you forget to cancel cutter compensation, and you run a program in graphics, you will get a 349PROG STOP W/O CANCEL CUTTER COMP alarm. This tells you that you ended a program without canceling cutter compensation. Pressing RESET or POWER OFF will also cancel cutter compensation. Because of these conditions that cancel cutter compensation, you don't need to put a G40 at the beginning of every program.

G80 Cancels Canned Cycles - If you forget to cancel a canned cycle with a G80, RESET, G00, G01, M30, and Power Down will also cancel any active canned cycle.

G97 Constant Non-Varying Spindle Speed - For safety reasons you should program in a G97 at the beginning and end of every tool cycle, with a spindle speed.

G99 Feed Per Revolution - This command changes how the "F" address is interpreted. The F command indicates inches per spindle revolution when Setting 9 is set to INCH. If Setting 9 is set to METRIC, F indicates millimeters per revolution. G99 and G98 are modal commands. G99 is the default command and the one you'll usually want. In certain situations, you may choose G98 for Feed Per Minute and then switch back to G99. But most customers will already be in G99 and never switch to G98, so there's usually no need to have it in a safe start line. If a mistake is made, using the wrong feed command, you'll easily see when it happens, when running the program on the machine.

Example: if you program a feed of F0.005 to feed per revolution, and you accidentally program in a G98 (Feed Per Minute), then it would take forever to feed 0.005 a minute. If you programmed F10 for feed per minute (G98). and you were accidentally in Feed Per Revolution (G99), and you had the spindle on, at a low speed of S200, the Calculated feed rate for this would be 2000ipm, which is above the machine's maximum feed rate. The machine would stop and give you an alarm

Program Structure

A CNC part program consists of one or more blocks of commands. When viewing the program, a block is the same as a line of text. Blocks shown on the screen are always terminated by the ";" symbol which is called an EOB (End Of Block). Blocks are made up of alphabetical address codes which are always an alphabetical character followed by a numeric value. For instance, the specification to move the X-axis would be a number proceeded by the X symbol.

Programs must begin and end with a percent (%) sign. After the first percent (%) sign with nothing else on that line, the next line in a program must have a program number beginning with the letter O (not zero) and then the number that defines that program. Those program numbers are used to identify and select the main program to be run, or as a subprogram called up by the main program. The % sign will "not" be seen on the control. But they must be in the program when you load a program into the control. And they will be seen when you download a program from the machine. The % signs are automatically entered in for you if you enter a program on the Haas control.

A program may also contain a " / " symbol. The " / " symbol, sometimes called a slash, is used to define an optional block. If a block contains this symbol, any information that follows the slash in a program block will be ignored when the **BLOCK DELETE** button is selected when running a program.

On the following page is a sample program as it would appear on the screen. The words following the ":" are not part of the actual program but are put there as further explanation.

This program will rough and finish turn and face for a part with two diameters along with drilling and tapping for a 3/8-16 x 1.0 deep threaded hole on one end.

%	Program must begin and end with a %
O00018	Letter "O" and a five-digit program number
(CNC LATHE PROGRAM EXAMPLE)	Comment statement between parenthesis
N1 (Rough O.D.)	First operation
G28	Return to machine zero for a tool change, cancel tool offset
T101 (O.D. TOOL x .031 TNR)	Select tool 1 with offset 1
G50 S2600	Set spindle speed max. clamp 2600 RPM
G97 S414 M03	Cancel CSS, 415 spindle speed, on forward
G54 G00 X3.6 Z0.1 MO8	Work offset, rapid X, Z axes, coolant on

G96 S390 CSS on at 390 SFM, coolant on
G00 Z0.005 Rapid to .005 from the end of part
G01 X-0.063 F0.005 Rough face end of part
G00 X3.6 Z0.1 Rapid to start point above part
G71 P10 Q20 U0.01 W0.005 D0.1 F0.01 Rough turning G71 canned cycle using the path defined between N10 thru N20, leaving .010 stock on the X-axis diameters, .005 stock on the linear faces, with 0.1 depth of cut each pass, feeding .010 per revolution.
N10 G42 G00 X0.82 N10 is the starting block called by the P block in the G71
G01 Z0. F0.004 Line that defines in it the geometry to rough out.
X0.9 Lines that define the part geometry to rough out
G03 X1. Z-0.05 R0.05 " " " " " "
/ G01 Z-1.75 " " " " " "
/ X1.75 " " " " " "
/ G03 X2.25 Z-2. R.25 " " " " " "
G01 Z-3.25 F0.004 " " " " " "
X2.94 " " " " " "
X3. Z-3.28 " " " " " "
Z-4.1 " " " " " "
N20 G40 X3.6 End of geometry with Q20 in G71 line, cancel cutter comp.
G97 S414 M09 Cancel CSS, coolant off
G28 Return to machine zero for a tool change, cancel tool offset
MO1 Optional program stop

N2 (FINISH O.D.) Second operation
G28 Return to machine zero for a tool change, cancel tool offset
T202 (O.D. TOOL x .031 TNR) Select tool 2 with offset 2
G50 S2600 Spindle speed max. of 2600 RPM
G97 S1354 M03 Cancel CSS, 1350 spindle speed, on forward
G54 G00 X1.1 Z0.1 M08 Work offset, rapid X and Z axis, coolant on
G96 S390 CSS on at 390 SFM
G00 Z0. Rapid to the end of part
G01 X-0.032 F.003 Finish face end of part
G00 X3.6 Z0.1 Rapid to start point above part
G70 P10 Q20 Finishing cycle calling N10thru N20 to do a finish pass
G97 S414 Cancel CSS, 475 spindle speed
G00 Z1.0 M09 Rapid Z axis, coolant off
G28 Return to machine zero for a tool change, cancel tool offset
M01 Optional program stop

N3 (Drill .750 Dia. x 2.5 Depth) Third operation
G28 Return to machine zero for a tool change, cancel tool offset
T303 (3/4 DIA. DRILL) Select tool 3 with offset 3
G97 S1986 M03 Cancel CSS, 1950 spindle speed, on forward
G54 G00 X0. Z1. M08 Work offset, rapid X and Z axis, coolant on
G83 Z-2.5 Q0.3 R0.1 F0.005 Deep hole peck drill 2.5 deep with a 0.3 peck
G80 G00 Z0.1 M09 Cancel canned cycle, rapid Z axis, coolant off
G28 Return to machine zero for a tool change, cancel tool offset
T100 Select tool 1 to get ready for the next part

M30	Stop program, rewind to the beginning
%	Program must end with a %

It is common to begin each tool in a part program with preparatory codes, turning on commands associated with that tool, and then ending by returning to the machine home, or a safe location, to position for a tool change. There might be a number of commands that are repeated throughout the program. This is done for safety to insure that the proper commands are attained if the operator has to begin, at the start of a tool in the middle of a program, in the event of tool breakage, to rerun a tool, or finish a part after powering up the machine. This is a common programming practice.

Author Credit: HAAS Automation, INC

8.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This Week's Assignment:

HAAS Simulator Training - Download the PowerPoint. We will go over the process within in class. The first part of this will seem familiar as we went over the machine start up process at the beginning of the semester.

Simulator Training Pt 1

Simulator Training Pt 2

Simulator Training Pt 3

Simulator Training Pt 4

HAAS Productivity Exercise - 2 Pages. We will also go over this assignment in class.

8.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on

the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

8.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

8.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Understand the following concepts: character, word, block, positive signs, leading zeros, modal commands, preparatory functions, miscellaneous function, sequence numbers. (SLO 4)
- Know how to properly set up a program structure for a CNC machine. (SLO 1, SLO 4)

Concept Content:

This week, we discussed program formatting. Next week, we will discuss lathe cycles and machine defaults. We will also have our first project for the semester.

This week in review:

Reading:

Embedded in module 8.2

Assignment:

HAAS Simulator Training

Module 9 - Machine Defaults and Lathe Cycles

9.1 Module Overview

Concept Goals:

By the end of this module, you should:

- Understand how canned cycles work, including how to end a cycle (SLO 4).

Concept Content:

This week, we will discuss machine defaults and canned cycles. See module 9.2 for more detail.

This week at a glance:

Reading:

Embedded into Module 9.2

Videos:

Intro to Canned Cycles - 8 Minutes

Assignment:

Plumb Bob Assignment

9.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we will discuss Machine Defaults and Canned Cycles. Both of these concepts are important to understand when it comes to using G codes on a CNC machine.

Machine Defaults

A **DEFAULT** is an automatic function of the machine tool control. After powering up the machine, the control will recognize the default "**G**" code values. The machine will go to the part zero that was entered in for G54 if no other work coordinate code was specified in the actual program because the machine automatically recognizes the **G54** column upon start-up. That is a **DEFAULT**. The defaults for the Haas mill are indicated by an asterisk (*) in the "Preparatory Functions (**G** Codes)" list of this workbook.

The control automatically recognizes these G codes when your Haas lathe is powered up:

G00	Rapid Traverse
G18	XZ Circular Plane Selection
G40	Cutter Compensation Cancel
G54	Work Coordinate Zero #1 (1 of 26 available)
G64	Exact Stop Cancel
G80	Canned Cycle Cancel
G97	Constant Surface Speed Cancel
G99	Feed Per Revolution

There is no default feed rate (F code) or spindle speed (S code), but once an F or S code is programmed, it will apply until another feed rate or spindle speed is entered or the machine is turned off.

Machining Cycles for Turning and Grooving

The following is a list of the canned cycles that can be used for turning and grooving for the HAAS lathe controls.

MACHINING CYCLES

G70	Finishing Cycle
G71	O.D./I.D. Stock Removal Cycle
G72	End Face Stock Removal Cycle
G73	Irregular Path Stock Removal Cycle
G74	End Face Grooving or Z axis turn with a Chip Break Cycle
G75	O.D./I.D. Grooving or X axis turn with a Chip Break Cycle
G76	Thread Cutting Cycle, Multiple Pass
G90	O.D./I.D. Turning Cycle Modal
G92	Thread Cutting Cycle Modal
G94	End Face Cutting Cycle Modal

A Machine Cycle is used to simplify the programming of a part. Machine Cycles are used for the most common repetitive operations such as turning, facing, threading, and grooving. There are both modal and non-modal machine cycles. Modal cycles G90, G92 and G94 remain in effect after they are defined. After any subsequent X or Z axis positioning, the canned cycle is executed again. Modal machining cycles remain in effect until canceled by a G80, G00, an End of Program (M30), or RESET. Non-modal G71, G72, G73, G74, G75, and G76 Machine cycles are effective for only the block that contains them but will be performing a series of machining moves to execute that command block.

Canned Cycles for Drilling, Tapping, and Boring

Once selected, a canned cycle is active until canceled with a G80 code. There are six operations involved in every canned cycle:

1.) Positioning of X and Z axes.

2.) Rapid traverse to the reference R-plane.

3.) Drilling, boring, or tapping action.

4.) Operation at the bottom of the hole.

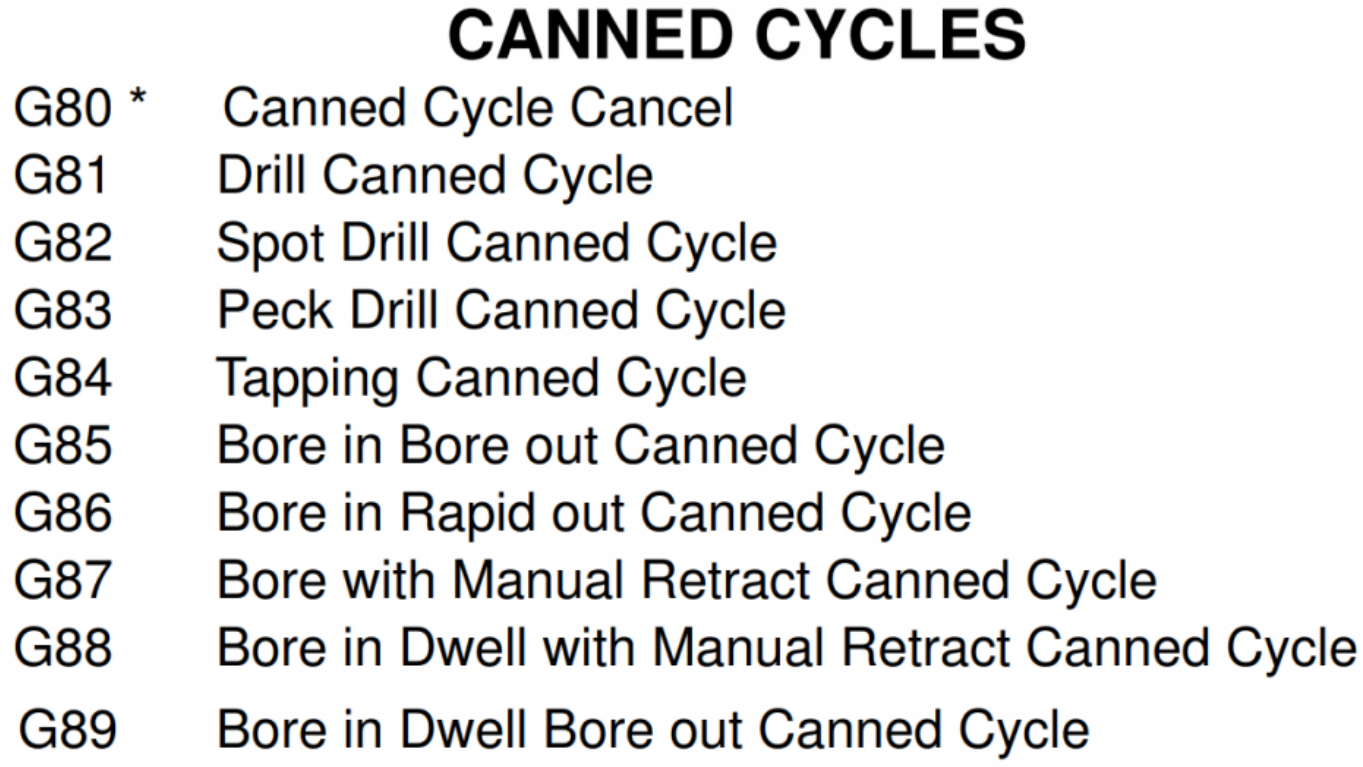
5.) Retraction to the reference R-plane.

6.) Rapid traverse to the initial starting point.

These cycles are modal, which remain in effect after they are defined and are executed in the Z axis for each positioning of X axes in a program. Some of the cycle command values can also be changed after the cycle has been defined. The command values most often changed during a cycle are the R plane value and the Z depth value. These modal cycles will be canceled with the G80, G01, or G00 commands. The X-axis moves in these modal machine cycles are performed as rapid moves.

The operation of a canned cycle will vary according to whether incremental (U, W) or absolute (X, Z) is specified. Incremental motion is often useful in a canned cycle. If a loop count (Lnn code number) is defined within the block, the canned cycle will repeat that many times with an incremental U (X-axis) move between each cycle.

The following is a list of the canned cycles that can be used on the Haas lathe:



Videos:

Intro to Canned Cycles - 8 Minutes

9.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This Week's Assignment:

Plumb Bob Assignment - Download the document and use the CNC machines in class to create the part.

9.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

9.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

9.6 Module Wrap-Up

Concept Goals:

By the end of this module, you should:

- Understand how canned cycles work, including how to end a cycle (SLO 4).

Concept Content:

This week, we discussed machine defaults and canned cycles.

This week in review:

Reading:

Embedded into Module 9.2

Videos:

Intro to Canned Cycles - 8 Minutes

Assignment:

HAAS Productivity Exercise - 2 Pages

Module 10 - Rapid Position, Linear Interpolation, and Circular Interpolation Commands

10.1 Module Overview

Concept Goals:

By the end of this module, you should:

- Understand the uses for rapid position, linear interpolation, and circular interpolation commands (SLO 4).

Concept Content:

This week we will talk about Rapid Position, Linear Interpolation, and Circular Interpolation Commands. See module 10.2 for more details.

This week at a glance:

Reading:

Embedded into module 10.2

Assignment:

X Diameter and Z Location Exercise 2 - 12 Questions

10.2 Module Content Resources

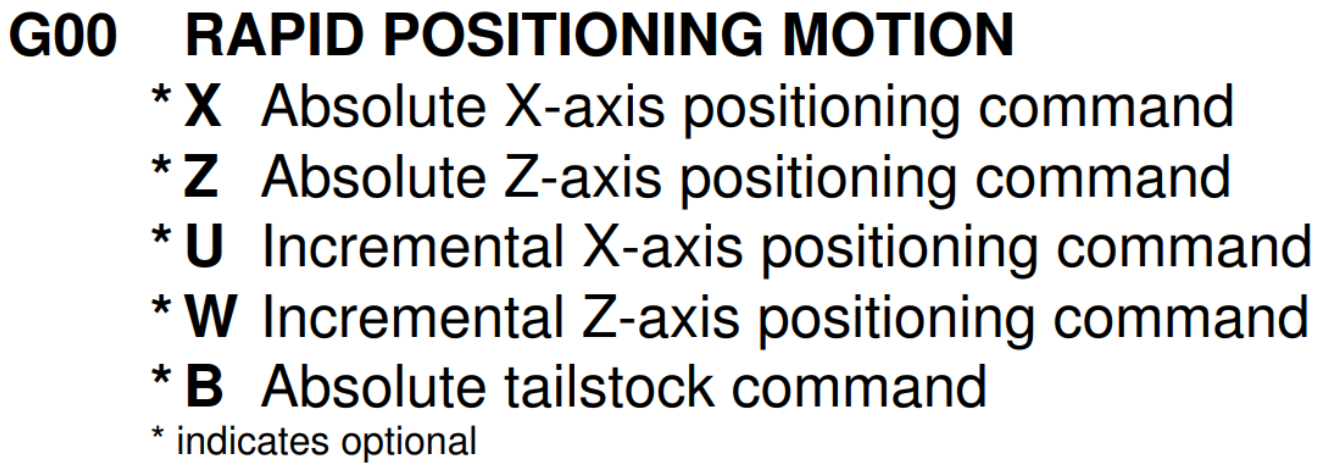
Concept Goals:

Outline the learning goals for this module here.

Concept Content:

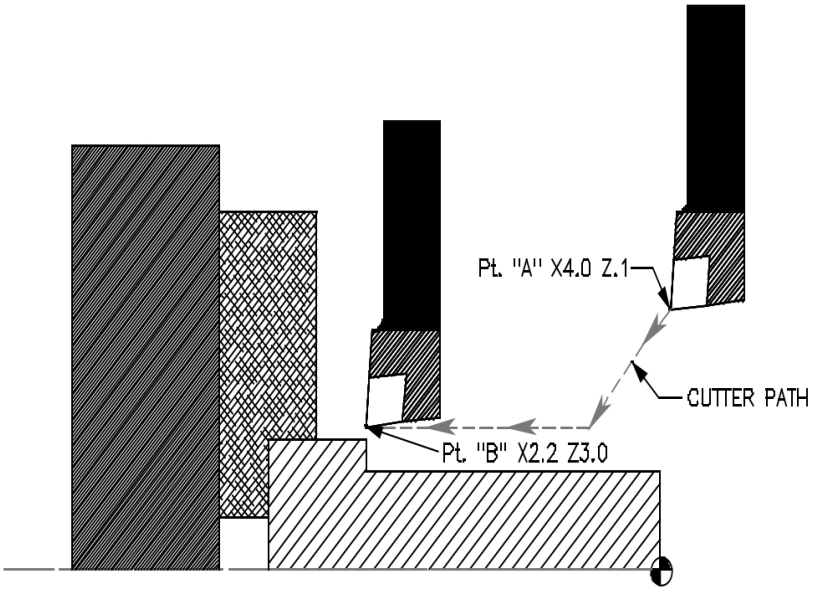
This week we will talk about Rapid Position, Linear Interpolation, and Circular Interpolation Commands. These are commands related to positioning your spindle in your CNC machine.

Rapid Position Command



This G code is used for the rapid traverse of the two axes of the machine from one program point to the next program point. The auxiliary (tailstock) B axis, can also be moved in rapid with a G00. This G code is modal so that all following blocks will be in rapid motion until another group 01 G code is specified. Generally, rapid motion done in both X and Z axes will not be in a straight line from one program location to the next program location. All the axes specified are moved at the maximum speed and will not necessarily complete each axis move at the same time. So you need to be careful of any obstructions to avoid this type of rapid movement. The tool will first move from the current position in a straight line along a 45-degree angle to an intermediate location until one of these axes has completed its move. Then the machine will move parallel to the X or Z axis to complete the rapid move to the final location. These rapid moves may be in ABSOLUTE or INCREMENTAL coordinate command values which will change how those values are interpreted. The “U” letter address relates to X-axis incremental moves and the “W” letter address relates to Z-axis incremental moves.

To move from point “A” to point “B”, the programmed line can be either:



G00 X2.2 Z-3.0
or **G00** U-1.8 W-3.1
or **G00** X2.2 W-3.1
or **G00** U-1.8 Z-3.0

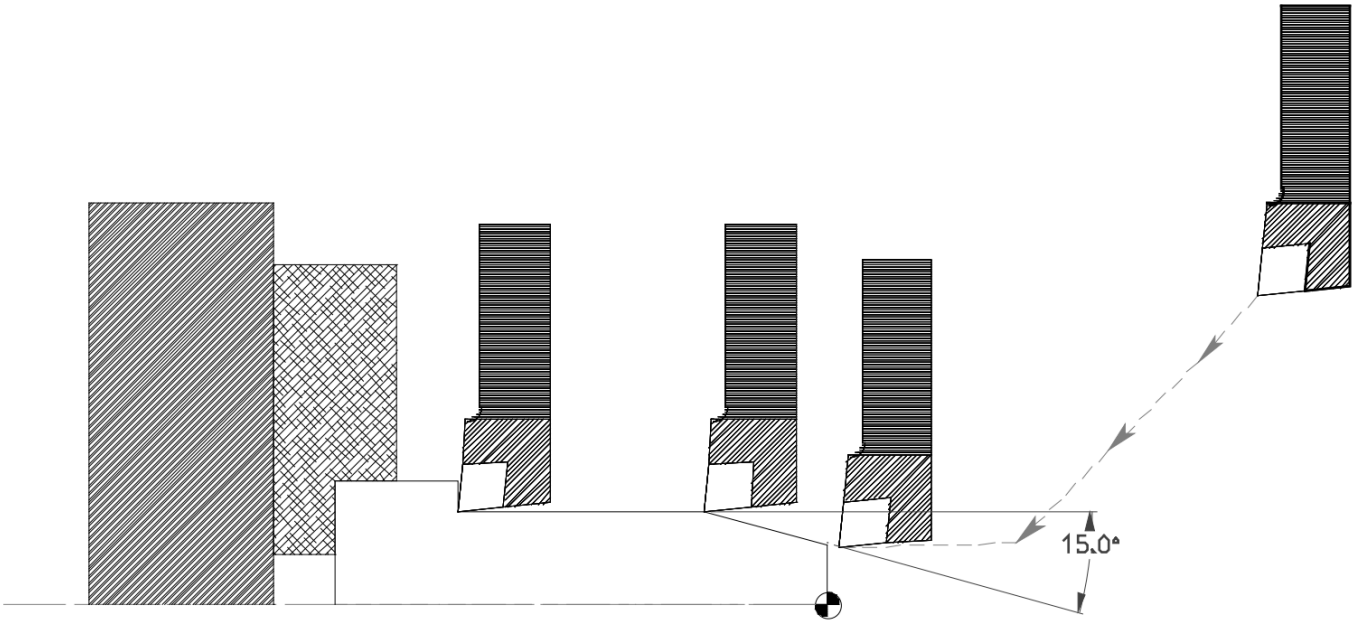
Linear Interpolation Command

G01 LINEAR INTERPOLATION MOTION

* **X** Absolute X-axis motion command
* **Z** Absolute Z-axis motion command
* **U** Incremental X-axis motion command
* **W** Incremental Z-axis motion command
* **F** Feed rate in inches (or mm) per minute
*indicates optional

This G code provides for straight line (linear) motion from point to point. Motion can occur in 1 or 2 axes. Both axes will start and finish motion at the same time to move the tool along a straight line path parallel to an axis or at a slope (angled) line. The speeds of all axes are controlled so that the feed rate specified is achieved along the actual path. The F (Feedrate) command is modal and may be specified in a previous block. These moves may be made in ABSOLUTE or INCREMENTAL coordinate command values which change how those values are interpreted. The **"U"** letter address relates to X-axis incremental moves, and the **"W"** letter address relates to Z-axis incremental moves. Only those axes specified are moved in either absolute **X Z**, or incremental **U W** commands.

G00 X0. 9106 Z0.1 **(ABSOLUTE)** **G01 X1.5 Z-1. F0.006** **Z-3** **X2.**	**G00 X0. 9106 Z0.1** **(INCREMENTAL)** **G01 U.5894 W-1.1. F0.006** **W-1.0** **U0.5**	**G00 X0. 9105 Z0.1** **(ABS. AND INC.)** **G01 X1.5 W-1.1 F0.006** **Z-3** **U0.5**



Circular Interpolation Commands

G02 CIRCULAR INTERPOLATION MOTION CW

- ***X** Absolute circular end point X-axis motion command
- ***Z** Absolute circular end point Z-axis motion command
- ***U** Incremental circular end point X-axis motion command
- ***W** Incremental circular end point Z-axis motion command
- ***I** X-axis Incremental distance from the start point to arc center (If R is not used)
- ***K** Z-axis Incremental distance from the start point to arc center (If R is not used)
- ***R** Radius of the arc (If I and K are not used)
- ***F** Feed rate in inches (or mm) per minute

*indicates optional

G03 CIRCULAR INTERPOLATION MOTION CCW

- ***X** Absolute X-axis arc end point motion command
- ***Z** Absolute Z-axis arc end point motion command
- ***U** Incremental X-axis arc end point motion command
- ***W** Incremental Z-axis arc end point motion command
- ***I** X-axis incremental distance from the start point to arc center (If R is not used)
- ***K** Z-axis incremental distance from the start point to arc center (If R is not used)
- * R Radius of the arc (If I and K are not used)
- * F Feed rate in inches (or mm) per minute

*indicates optional

G03 will generate counterclockwise circular motion but is otherwise defined the same as G02 clockwise circular motion.

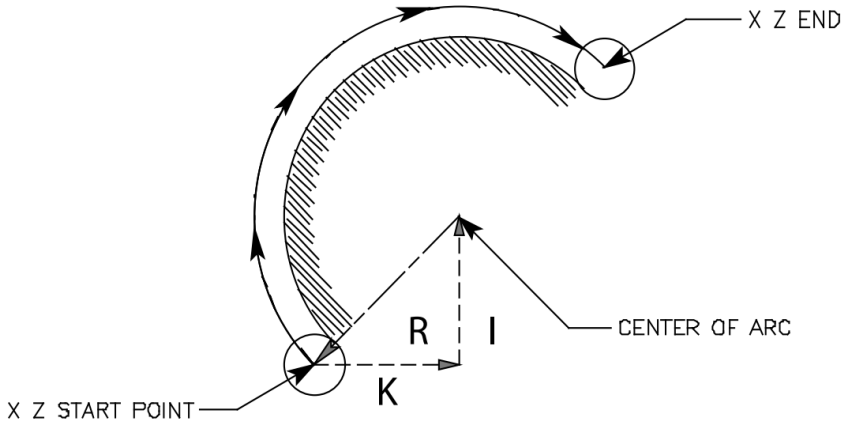
Circular interpolation commands are used to move a tool along a circular arc to the commanded end position. Five pieces of information are required for executing a circular interpolation command: Plane selection, arc start position coordinates, rotation direction, arc end position coordinates, and arc center coordinates or arc radius.

There are two ways to define a circular interpolation move. The first way is using I and K method, which defines the distance and direction from the start point to the arc center. The R method, is easier to define, and is the actual arc radius value your interpolating.

When using I and K, it does not need to be entered in the program if either one has a zero value. It will be recognized as zero if it's not listed in the G02 or G03 line.

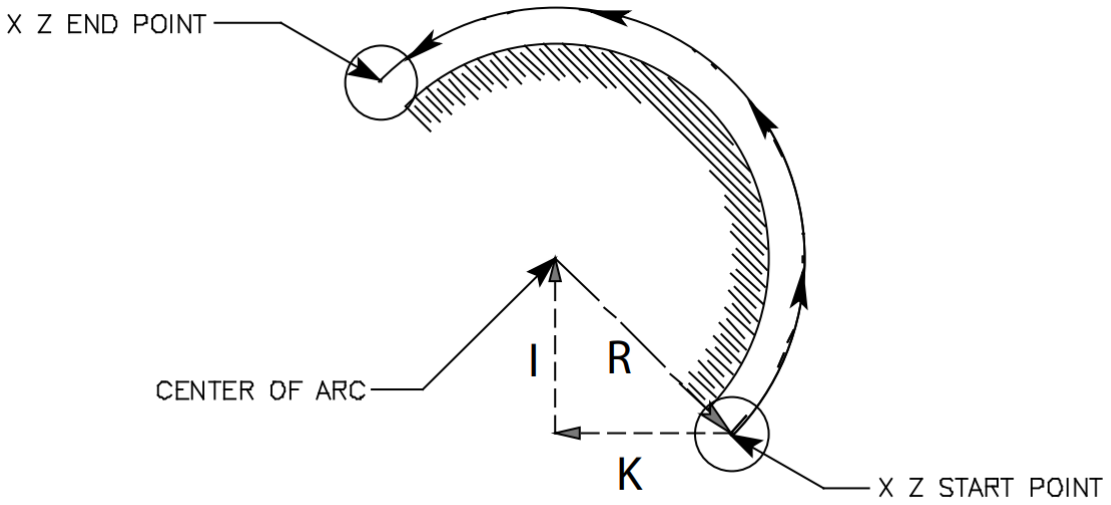
R: Using an R is the distance from the starting point to the arc center. With a positive R value, the control will generate a circular path of 180 degrees or less, but to generate a circular path of over 180 degrees, you specify a negative R.

When feeding to a location using G02 or G03, it should be using a previously defined tool geometry "Txxyy" offset commanmd. The Txxyy code defines the tool and offset that are being used.

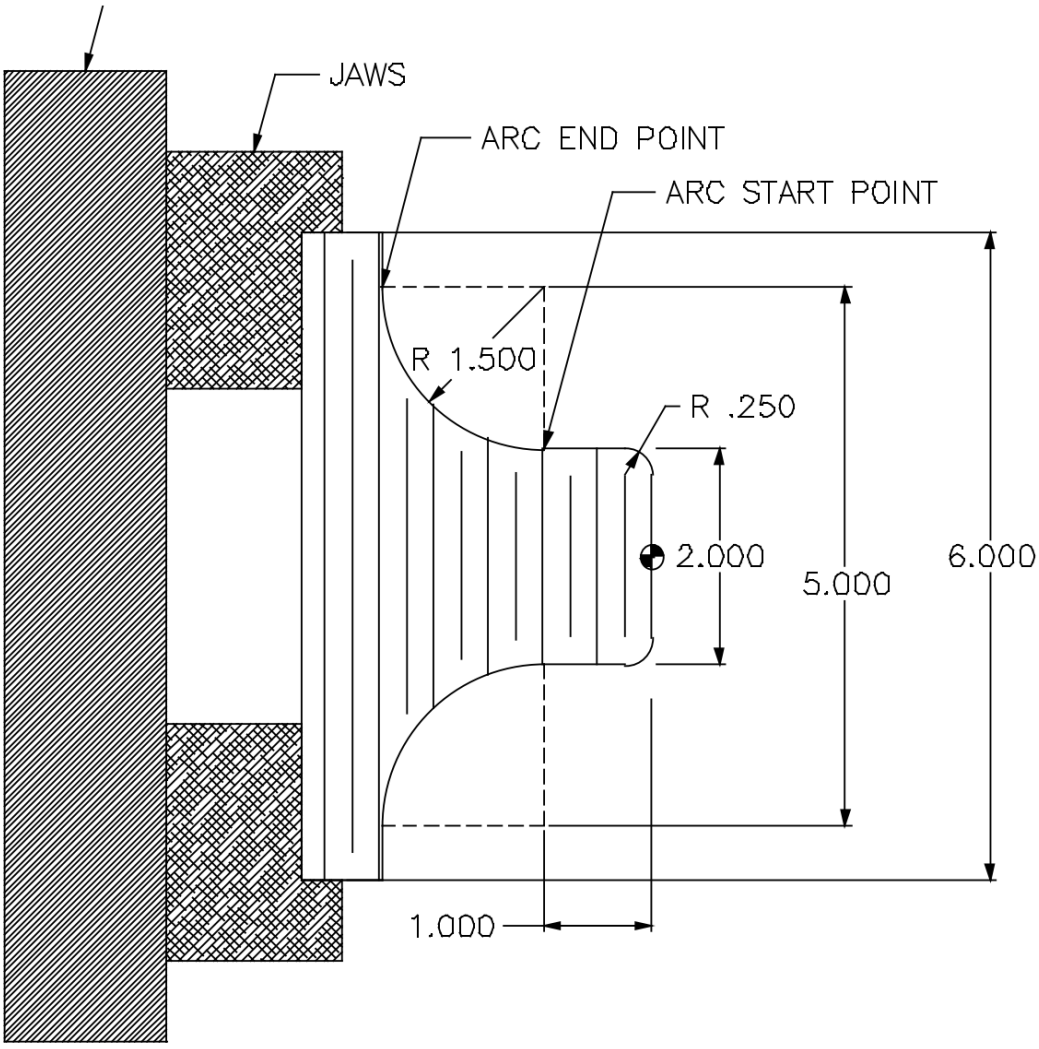


Five pieces of information are required for executing a circular interpolation command.

	Item	Command	Remark
1	Plane Selection Command Plane Selection Command Plane Selection Command	G17 G18 G19	Arc parallel to X Y-plane (Not Available) Arc parallel to Z X-plane (Default) Arc parallel to Y Z-plane (Not Available)
2	Arc start position coordinate	X, Z	Coordinates to the start position of arc
3	Rotation direction	G02 G03	Clockwise interpolation direction Counterclockwise interpolation direction
4	Arc end position Absolute Arc end position Incremental	X, Z U, W	End position coordinates of the arc defined from part origin zero point. End point, incremental distance and direction defined from the start point of the arc to the end point in the X and Z axis.
5	I and K method (arc center coordinates) ("I" is the X axis incremental distance and direction from the arc start point to the center of the radius)("K" is the Z axis incremental distance and direction from the arc start point to the center of radius) R method (actual part arc path radius value)	I,K R	Incremental distance and direction from start point of arc to the arc center for X and Z axes using the "I" for the X axis, and "K" for the Z axis. Arc radius value of tool path. Easier to define than I and K.



CIRCULAR INTERPOLATION MOTION EXAMPLE



This program example starts machining with the tool at the beginning of the .250 radius to machine in a counterclockwise direction. The cutter moves to the start of 1.500R and around in a clockwise direction from START POINT to END POINT. For each radius, the programming code would look like this:

N3	(Tool nose compensation is not being used in this example. This
N4	program example is defined to a sharp tool point.)
N5 G00 X1.4 Z0.1	(Tool rapids to X1.4 Z0.1 in front of part.)
N6 G01 Z0. F0.008	(Tool feeds to Z0. face of part.)
N7 X1.5	(Tool feeds up to 1.5 dia. in X-axis to the start of .250 radius.)
N8 G03 X2. Z-0.25 R0.25	(CCW .25 Radius to the arc end point using R.)
or G03 X2. Z-0.25 I0. K-0.25	(or, CCW .25 Radius to the arc end point X2. Z-.25 with I and K)
N9 G01 Z-1.	(Machines the 2 dia. to the arc start point at Z-1.)
N10 G02 X5. Z-2.5 R1.5	(CW 1.5 Radius to the arc end point using R.)
or G02 X5. Z-2.5 I1.5 K0.	(or, CW 1.5 Radius to the arc end point X5. Z-2.5 with I and K)
N11 G01 X6.0	(Feeds up X-axis to 6. dia.)
N12 G00 Z0.1	
N...	

G04 Dwell
P The dwell time in seconds or milliseconds

G04 is used to cause a delay or dwell in the program. The block containing G04 will delay for the time specified in the P code, coolant and spindle will remain on. If the P has a fraction part, the delay is in milliseconds (0.001 seconds); otherwise the delay is in seconds.

EXAMPLE: G04 P1.0 (for a delay of 1.0 second)
-or-
G04 P2.5 (for a delay of 2.5 seconds)

The G codes G20 and G21 are used to select between inch and metric. In the Haas control, the G20 (inch) and G21 (mm) codes can only be used to make sure that the inch/metric setting is set correctly for that program.

Selecting between inch and metric programming can only be done from the Setting page, Setting 9. Changing the setting from inches to metric or back again will interpret the content of any program listed in memory. You must load in a program with metric values after changing this setting to MM.

10.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This Week's Assignment:

X Diameter and Z Location Exercise 2 - 12 Questions

Download the worksheet and upload the completed version to the assignments tab under quiz.

10.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task

for your specific subject, please feel free to delete this section from your class.

10.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

10.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Understand the uses for rapid position, linear interpolation, and circular interpolation commands (SLO 4).

Concept Content:

This week we talked about Rapid Position, Linear Interpolation, and Circular Interpolation Commands. Next week we will continue to discuss various types of commands.

This week in review:

Reading:

Embedded into module 10.2

Assignment:

X Diameter and Z Location Exercise 2 - 12 Questions

Module 11 - Reference Point of Return, Work Coordinate Section, Spindle Feed Commands and Feed Commands

11.1 Module Overview

Concept Goals:

By the end of this module, you should be able to:

- Understand the uses for reference point of return, work coordinate section, spindle feed, and feed commands. (SLO 4).

Concept Content:

This week we will discuss more feed commands. See module 11.2 for more details.

This week at a glance:

Reading:

Embedded into module 11.2

Assignment:

CNC Turning Soft Jaws - 1 Page

CNC Jack Screw Body Project

11.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we will finish discussing commands. We will cover four different types of commands as outlined in the readings below.

G28 Fanuc Return to Machine Zero Through Reference Point

The G28 code is used to return to the machine zero position on all axes. If an X or Z axis is specified on the same block, only those axes will move and return to the machines' zero point. If X or Z specifies a different location than the current position, then the movement to machine zero will be through the specified point. If no X or Z is specified, all axes will be moved directly to machine zero, along with the tailstock, B axis (for the machines that have a tailstock). If you do not want to position through an intermediate point while specifying an X or Z axis to position to machine home, then use an incremental U0 and/or W0 command for the specific axis that you want to send to machine zero. This will command those axis specified to position incrementally to a zero distance as an intermediate point, and then those axes specified will go directly to machine zero. G28 will not recognize any tool length offsets in this move, though the offset is not cleared. The tool offsets will not be cleared until a T0, M30, Reset, or a new offset is called up.

G28 U0 W0

The G28 command will send turret in the X and Z axes to machine home along with the tailstock. If tailstock is home, not being used, then this is not a problem. If tailstock is being used and it's holding a part, you may not want to send the tailstock home when a G28 command is in a program line. To command and send only the turret to go home, and not the tailstock, program in G28 U0. W0. to send only the X and Z axes home and the tailstock will then remain in place. If you want to send the X axis home first and then the Z axis, you could program G28 U0. on a separate line and then on the next line, program a G28 W0.

G51 is used on a YASNAC control, to cancel out any existing tool shift and tool wear offsets and then return to the machine zero position.

Spindle Speed Commands

G50 SPINDLE SPEED CLAMP

Spindle speed, on any CNC lathe, is limited by the maximum permissible speed of the power chuck. The effects of centrifugal force on its gripping force, unbalanced condition of the workpiece causing machine vibration, and so on, can be constrained by programming a G50 code at the start of each tool.

G50 can be used to clamp the maximum spindle speed. The control will not allow the spindle to exceed the S address value specified in the most recent G50 command. This is most often used in constant surface feed mode G96.

N1 G50 S2500 ; (SPINDLE RPM WILL NOT EXCEED 2500 RPM)
N2 G96 S390 M03 ; (ENTER CONSTANT SURFACE SPEED MODE, SPINDLE ON)

The MAXIMUM spindle speed is designated by the “S” word along with the preparatory G50 command with the “S” word in the same block with no other commands. Once the MAXIMUM spindle speed is established, any direct RPM programmed in G97 (direct revolution per minute) or control calculated RPM from the G96 (constant surface footage mode) that exceeds the RPM established by the G50 block, is ignored and the G50 S word spindle speed is used.

The second use of G50 is for older equipment that does not have work zero and geometry offset capabilities. This will not be covered in this workbook because it’s not desirable anymore because it’s not as easy to use as what is now available.

G96 Constant Surface Speed On

A G96 commands the control to maintain a constant surface speed, with the spindle speed and where the tool tip is relative to the part diameter. Surface speed is based on the distance of the tool tip to the spindle centerline. This is the radius of cut. Surface speed is maintained by adjusting the spindle speed based on the radius of cut. The current S code is used to determine the surface speed. G96 is modal.

The formula the machine performs while machining up or down a diameter with G96 on a part is calculating this formula: (SFM x 3.82) Dia. = RPM
Example on a 3/4 part diameter: (225 x 3.82) .750 = 1146

G97 Constant Surface Speed Off/Revolution Per Minute On

A G97 commands the control to not adjust the spindle speed based on the radius of cut. This command will cause the spindle to be at a constant speed. It will cancel any current G96 command. When G97 is actively on, any S command is in revolution per minute (RPM). G97 is modal.

The formula to find out what surface speed when machining on a part diameter, when the machine is at a constant spindle speed: Diameter x .2618 x RPM = SFM
The surface speed for a 5/8 drill at a 1400 spindle speed is: .625 x .2618 x 1400 = 229.075

Work Coordinate Section Commands

G52 SET LOCAL COORDINATE SYSTEM FANUC

This code selects the user coordinate shift value. It's non-modal. FANUC compatible.

G53 NON-MODAL MACHINE COORDINATE SELECTION

This code temporarily cancels work coordinates offset and uses the machine zero coordinate location. It's non-modal, so the next block will revert to the work coordinate that was previously active.

G54-59 WORK COORDINATE OFFSET SYSTEM #1 - #6

These codes select one of the six work coordinate (part zero) offsets that are entered in and stored on the offsets display page . All subsequent references to axes' positions will be interpreted with the new coordinate value.

G110-G129 WORK COORDINATE OFFSET SYSTEM #7-26

These codes are the same as work offsets G54 to G59.

G154 P1-99 WORK COORDINATE OFFSET SYSTEM #7-105

This was added in Lathe software version 5.02 and above. This feature has 79 additional work offsets. In all previous versions, the user was limited to a maximum of 27 work offsets. These included the standard work offsets (designated G54 through G59), and twenty additional work offsets (G110 through G129). Now with G154 P1 to P99, your able to activate additional work offsets. For example G154 P10 will select work offset 10 from the list of additional work offset. Note that G110 to G129 refer to the same work offsets as the G154 P1 through P20. I.e., they can be selected by using either method. The Work Offset display screens have been adjusted accordingly. The Position page display has also been enhanced so that when a G154 work offset is active, the heading in the upper right work offset will show the G154 P value.

Feed Section Commands

G98 FEED PER MINUTE

This command changes how the F address code is interpreted. The value of F indicates inches per minute when Setting 9 is set to INCH, and F indicates millimeters per minute when Setting 9 is set to METRIC. This code is modal.

IPM = CURRENT RPM x IPR

G99 FEED PER REVOLUTION

This command changes how the F address is interpreted. The value of F indicates inches per revolution of the spindle when Setting 9 is set to INCH, while F indicates millimeters per revolution of the spindle when Setting 9 is set to METRIC. This code is modal, and is the default feed mode.

IPR = IPM ∶ CURRENT RPM

Author Credit:

HAAS Automation Inc., HAAS Automation, Inc. Lathe Series Programming Workbook. **(Instructor Note: You can use this material provided your school uses HAAS machines).**

11.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week's assignments:

CNC Turning Soft Jaws - 1 Page

Download the worksheet and upload the completed version to the assignments tab under quiz.

Jack Screw Project

Download the blueprint and complete the project in class using the CNC Machines.

11.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

11.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

11.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Understand the uses for reference point of return, work coordinate section, spindle feed, and feed commands. (SLO 4).

Concept Content:

This week we discussed more feed commands. Next week we will pick up on part inspection.

This week in review:

Reading:

Embedded into module 11.2

Assignment:

CNC Turning Soft Jaws - 1 Page

Module 12 - Part Inspection

12.1 Module Overview

Concept Goals:

By the end of the module, you should be able to:

- Understand various methods of inspecting CNC machined tools (SLO 2)
- Be familiar with how Coordinate Measurement Machine inspections work (SLO 2)

Concept Content:

This week we will go over inspection methods. See module 12.2 for more details.

This week at a glance:

Videos:

Coordinate Measurement Machine Inspection for Geometric Tolerance - 2.5 Minutes

How to Make Final Inspection for CNC Machined Parts - 1.5 Minutes

Articles:

hmaking. (n.d.). *How to Check The Quality Of Your CNC Machining Parts*. hmaking.com. https://hmaking.com/how-to-check-the-quality-of-your-cnc-machining-parts/

VMT. (n.d.). *CNC Machining Quality Inspection process*. CNC machining quality inspection process_Blog_CNC Machined Parts Factory - Custom CNC Turning Milling Service Manufacturer - China VMT. https://www.machining-custom.com/blog/cnc-machining-quality-inspection-process.html

Assignment:

CNC Intro to Coolant Assignment - 1 Page

CNC Screw Jack Handle

12.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we will cover how to inspect the quality of the parts you are making with the CNC machine. There are a couple of ways you can do this and different types of tools. Some tools are manual and some are computerized. Please read the articles and watch the videos below for more detail.

Videos:

Coordinate Measurement Machine Inspection for Geometric Tolerance - 2.5 Minutes

How to Make Final Inspection for CNC Machined Parts - 1.5 Minutes

Articles:

hmaking. (n.d.). *How to Check The Quality Of Your CNC Machining Parts*. hmaking.com. https://hmaking.com/how-to-check-the-quality-of-your-cnc-machining-parts/

VMT. (n.d.). *CNC Machining Quality Inspection process*. CNC machining quality inspection process_Blog_CNC Machined Parts Factory - Custom CNC Turning Milling Service Manufacturer - China VMT. https://www.machining-custom.com/blog/cnc-machining-quality-inspection-process.html

12.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This Week's Assignments:

Understanding how machine tool coolant works is important for making sure your machine keeps working properly. Thus, this weeks assignment will be centered around that.

CNC Intro to Coolant Assignment - 1 Page

Download the worksheet and upload the completed copy to quiz under the assignments tab.

CNC Screw Jack Handle

Download the blueprint and use the CNC machines in class to create the part

12.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content

this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

12.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

12.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Understand various methods of inspecting CNC machined tools (SLO 2)
- Be familiar with how Coordinate Measurement Machine inspections work (SLO 2)

Concept Content:

This week we went over how to inspect the parts you create with the CNC machines. Next week, we will continue this course with another project to finish out the semester.

This week in review:

Videos:

Coordinate Measurement Machine Inspection for Geometric Tolerance - 2.5 Minutes

How to Make Final Inspection for CNC Machined Parts - 1.5 Minutes

Articles:

hmaking. (n.d.). *How to Check The Quality Of Your CNC Machining Parts*. hmaking.com. https://hmaking.com/how-to-check-the-quality-of-your-cnc-machining-parts/

VMT. (n.d.). *CNC Machining Quality Inspection process*. CNC machining quality inspection process_Blog_CNC Machined Parts Factory - Custom CNC Turning Milling Service Manufacturer - China VMT. https://www.machining-custom.com/blog/cnc-machining-quality-inspection-process.html

Assignment:

CNC Intro to Coolant Assignment - 1 Page

Module 13 - Turning Project Week 1

13.1 Module Overview

Concept Goals:

By the end of this module, you should be able to:

- Demonstrate the ability to accurately read a blueprint to complete a turning project (SLO 4)

Concept Content:

This week we will begin our final project of the semester. You will have this week, next week, and exam week if needed to complete it. Please see module 13.2 for more details.

This week at a glance:

Blueprint:

NIMS Turning Project

Reading:

Lathe Setup and Programing - 10 Pages.

Assignment:

Work on your project

13.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we will begin our big turning project that will demonstrate and test the skills you have learned so far. Using the blueprint from below, please create the part in the print over the next two-three weeks.

Blueprint:

NIMS Turning Project

Reading:

Lathe Setup and Programing - 10 Pages. This will be a good review to read through before using the machines this week.

13.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

Your assignment this week is to work on your project.

13.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content

this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

13.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

13.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Demonstrate the ability to accurately read a blueprint to complete a turning project (SLO 4)

Concept Content:

This week we began our final project of the semester. Next week we will continue working on the project.

This week in review:

Blueprint:

NIMS Turning Project

Reading:

Lathe Setup and Programing - 10 Pages.

Assignment:

Work on your project

Module 14 - Turning Project Week 2

14.1 Module Overview

Concept Goals:

By the end of this module, you should be able to:

- Demonstrate the ability to accurately read a blueprint to complete a turning project (SLO 4)

Concept Content:

This week we will continue working on our turning project. Please see module 14.2 for more info.

This week at a glance:

Blueprint:

NIMS Turning Project

Assignment:

Work on your project

14.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we will continue working on your turning project. Ideally, it would be great if you can finish it this week. If not, don't worry as you will have time next week after the exam to work on your project.

Blueprint:

NIMS Turning Project

14.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week's assignment is to work on your project.

14.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

14.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

14.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Demonstrate the ability to accurately read a blueprint to complete a turning project (SLO 4)

Concept Content:

This week we continued working on our turning project. If you did not finish it, you will have next week to complete it.

This week in review:

Blueprint:

NIMS Turning Project

Assignment:

Work on your project

Module 15 - Final Exam

15.1 Final Exam

Concept Goals:

Demonstrate an understanding of the course material.

Concept Content:

This week is the week of our final exam. To open it, go to the assignments tab and look under test.

15.2 Course Wrap-Up

Concept Content:

Thank you for completing your final exam. If you have not finished your project from week 13/14 yet, please complete it by the end of this week. If you have finished both the exam and project, congratulations on completing the course!

Faculty Resources (For Instructor Only, Do Not Publish Live)

Odigia Guide

Concept Content:

Click on the resources tab to find the guide sheet for instructors.

Module 3 Assignment Answer Key

Concept Content:

Module 3 Assignment Answer Key - 5 Questions

Module 6 Assignment Answer Key

Concept Content:

Module 6 Answer Key

Module 10 Assignment Answer Key

Concept Content:

Module 10 Assignment Answer Key - 12 Questions

Module 11 Assignment Answer Key

Concept Content:

Module 11 Worksheet Answer Key

Module 12 Assignment Answer Key

Concept Content:

Module 12 Assignment Answer Key

Extra Projects

Concept Content:

This is a repository of extra projects you can use in the classroom in case your students get through the ones built into the course.

Screw Jack V-Camp

Screw Jack Assembly

Sofe Face Hammer Handle

Soft Face Hammer Head

Soft Hammer Replacable Tip

Soft Hammer Assembly