

Table of Contents

Preface	11
Prerequisites	12
Basic machining practice experience	12
Controls covered.....	12
Limitations	12
The need for hands-on practice.....	13
Instruction method	13
Scope.....	13
Key Concepts approach.....	14
Lesson structure.....	14
Practice makes perfect.....	14
Key Concepts and lessons	14
Enjoy!	15
Key Concept #1: Get Ready To Learn About CNC Machining Centers	17
Lesson One: Basic Machining Practices Required For CNC Machining Centers	19
Shop Safety.....	19
Safety equipment.....	19
Safety practices.....	20
Handling raw material.....	20
Finished workpiece handling.....	21
Tightening and loosening fasteners.....	21
Getting around the shop.....	21
Behavior.....	22
Machining-center-specific safety practices.....	22
Shop math.....	23
A few suggestions for choosing your calculator.....	23
Keep it simple.....	23
Watch out for trick functions.....	23
Avoid solar powered calculators	24
Big buttons – big display.....	24
Be sure it has clear entry button	24
Calculator basics	25
What is an arithmetic expression?.....	25
Practice makes perfect.....	26
The most basic arithmetic functions	27
Addition.....	27
Subtraction.....	27
Multiplication.....	27
Division.....	27
More on the priority of arithmetic operators	27
The use of parentheses in an expression.....	28
Does your calculator have parentheses?.....	28
Does your calculator have one or more memories?.....	29
What is a formula?.....	30
Who creates a formula?.....	31
More about numbers	32
Integers.....	32
Real numbers	32
Decimal format.....	32
Fractional format.....	33
Converting fractional format to decimal format.....	33
Decimal equivalents for common fractions	34
Decimal places	34
Rounding to a given number of decimal places	34

Measurement systems.....	35
Saying numeric values out loud in a machine shop.....	36
Terminology in the Metric system	37
Understanding polarity.....	37
Summary of shop math.....	38
Introduction to blueprint reading	39
Orthographic projection.....	39
Line appearance.....	40
Do you have a sample workpiece?.....	40
Drawing scale.....	40
Dimensioning.....	41
Measurement systems	42
Other descriptive notes.....	42
Title block.....	42
Revisions	42
Introduction to tolerance interpretation	44
Is a measured workpiece attribute acceptable?	45
What dimension are you shooting for?.....	45
The polarity for adjustments.....	46
Other types of tolerance specifications	47
High and low limit specification	47
Uneven plus versus minus values	47
What are geometric tolerances?.....	48
Measuring devices.....	50
A general suggestion for getting started with any measuring device.....	50
Methods by which measured values are displayed on measuring devices	51
Vernier scale	51
Dial display for small values	52
Electronic digital display.....	52
The workpiece attributes that you'll be measuring.....	53
Accuracy of the measuring device.....	55
Micrometer.....	55
Testing for calibration.....	57
Depth micrometer.....	57
Testing for calibration.....	58
Calipers	58
Testing for calibration.....	60
Lesson Two: What Does A CNC Machining Center Do?	63
Comparing a CNC machining center to other types of machines	63
Milling machines	64
Machining operations performed on machining centers	66
Introduction to cutting conditions.....	66
How cutting conditions are recommended.....	66
Spindle speed: Converting from surface feet per minute (sfm) to revolutions per minute (rpm).....	67
Spindle speed: Converting from meters per minute (mpm) to revolutions per minute (rpm).....	67
Feedrate: Converting inches per tooth (ipt) or inches per revolution (ipr) to inches per minute (ipm).....	67
Feedrate: Converting millimeters per tooth (mmpt) or millimeters per revolution (mmpr) to millimeters per minute (mmpm).....	67
Getting recommendations for cutting conditions	68
Other cutting conditions related to machining operations	68
Cutting tool materials	68
Hole-machining operations	69
Center drilling.....	69
Cutting conditions for center drilling.....	71
Dull tool replacement and sharpening.....	71
Spot drilling.....	71
Cutting conditions for spot drilling.....	72
Dull tool replacement and sharpening.....	72

Counter sinking..... 72
 Cutting conditions for counter sinking 73
 Dull tool replacement and sharpening..... 73
 Drilling with twist drills 74
 Allowing for the Drill Point 74
 Calculating the depth of through holes 75
 Peck drilling to break chips 75
 Peck drilling for deep holes 75
 Cutting conditions for twist drills 76
 Applications for drilling with twist drills 76
 Dull tool replacement and sharpening..... 76
 Other drilling tools 76
 Spade drills 76
 Carbide inserted drills 77
 Dull tool replacement and sharpening..... 77
 Counter boring..... 78
 Cutting conditions for counter boring tools 79
 Dull tool replacement and sharpening..... 79
 Reaming..... 79
 Cutting conditions for reamers..... 79
 Applications for reaming..... 80
 Boring with boring bars 80
 Cutting conditions for boring bars 80
 Dull tool replacement and sharpening..... 81
 Tapping..... 81
 The hole before tapping..... 82
 Actions required for tapping..... 83
 Cutting conditions for taps..... 83
 Dull tool replacement and sharpening..... 83
 Milling operations 84
 Milling cutters that can plunge like hole machining tools 84
 Climb versus conventional milling 86
 Radial and axial depth of cut..... 86
 Rough milling versus finish milling..... 87
 Cutting tool material..... 87
 Cutting conditions for milling operations 89
 Types of milling cutters 90
 End mills 90
 Circle milling a counter bored hole 90
 Pocket milling 90
 Keyway and slot milling 91
 Face milling..... 91
 Exterior peripheral milling..... 91
 Face mills 92
 Other milling cutters..... 93
 Ball and bull end mills 93
 Slotting cutters 95
 Thread milling cutters 95
 Motions needed for thread milling..... 96
 Machining the entire thread in one pass 96
 Summary of key points..... 97
 Conclusion to Key Concept Number One..... 97

Key Concept #2: Know Your Machine From An Operator’s Viewpoint 101

The need for hands-on experience..... 101

Lesson Three: General Flow Of The CNC Process 103

Companies that use CNC machining centers 103

What will you be doing?..... 104

Flow of the CNC process	105
Study the workpiece drawing.....	105
Decision is made as to which CNC machine to use.....	105
The machining process is developed.....	105
Tooling is ordered and checked.....	105
The program is developed	105
Setup and production run documentation is made	105
Program is loaded into the CNC control's memory.....	105
The setup is made.....	106
The program is cautiously verified (still part of setup).....	106
Production is run.....	106
Corrected version of the program is stored for future use.....	106
Lesson Four: Machine Configurations	111
Vertical machining centers	111
C-frame style	111
Directions of motion (axes) for a C-frame style vertical machining center.....	112
Axis polarity	112
Knee style vertical CNC milling machines	114
Bridge-style vertical machining center (also called gantry-style).....	115
One more point about the confusion related to axis polarity	115
Horizontal machining centers	116
Directions of motion (axes) for a horizontal machining center	116
Programmable functions of machining centers.....	118
Spindle.....	118
Spindle speed	118
Spindle activation and direction	119
Giving a spindle command.....	120
Coolant.....	120
Automatic tool changer.....	120
What else might be programmable?.....	121
Other important machine functions	121
Motion rate	121
Mode switch.....	122
Zero return position	122
Lesson Five: Buttons And Switches On The Operation Panels	127
The two most important operation panels	127
The control panel buttons and switches	127
Display screen control keys (soft keys).....	128
Position display pages	129
Program display pages.....	129
Offset display pages	130
Program check display pages	130
Other display screen modes	130
Letter Keys	131
The slash key (/)	131
Number keys	131
Decimal point key	131
The input key.....	131
Cursor control keys	131
Program Editing Keys	131
Reset key.....	132
The machine panel	132
Mode switch.....	132
Cycle start button.....	132
Feed hold button.....	132
Feedrate override switch	133
Rapid traverse override switch	133

Emergency stop button	133
Conditional switches	134
Dry run on/off switch.....	134
Single block on/off switch.....	134
Block delete on/off switch (also called optional block skip)	134
Optional stop on/off switch.....	134
Buttons and switches for manual functions	135
Axis jogging controls	135
Handwheel controls	135
Spindle control.....	135
Automatic tool changer control.....	136
Indicator lights and meters	136
Spindle rpm and horsepower meters	136
Axis drive-motor horsepower meter.....	136
Cycle indicator lights	136
Zero return position indicator lights.....	136
Optional stop indicator light.....	136
Other buttons and switches on the machine panel	136
Other operation panels on your machining center.....	137
More on the mode switch.....	137
The manual mode.....	137
What if the machine is in the wrong mode?	138
The manual data input mode.....	138
The manual data input (MDI) mode switch position.....	138
Commanding an MDI zero return.....	139
The complete procedure to give an MDI command.....	139
Commanding an MDI tool change.....	139
Commanding spindle activation with MDI.....	139
Other times when MDI is used.....	140
The edit mode switch position.....	140
To make a program in memory the active program (to call up a program).....	141
To enter a new program	141
What if you make a mistake when typing?	141
The program operation mode.....	141
To run the active program from the beginning.....	142
Lesson Six: The Key Operation Procedures	145
Key Concept #3: Know The Compensation Types	153
Lesson Seven: Introduction To Compensation	155
What is compensation and why is it needed?	155
Does this sound familiar?.....	156
More on tolerances	156
The initial setting for compensation	156
So when is trial machining required?.....	157
What happens as cutting tools begin to wear?	157
What do you shoot for?.....	157
Where do you actually make the adjustment?	158
Never make sizing adjustments by changing the CNC program!.....	159
Understanding offsets	159
Offset organization.....	160
Offsets related to cutting tools	160
Offsets related to program zero assignment.....	161
How offsets are instated.....	162
Special note for machines that have but one register per offset.....	162
Lesson Eight: Fixture Offsets And Assigning Program Zero	167
How is the program zero point determined?	167
What about program zero in the Z axis?.....	169

Assigning program zero	169
Program zero assignment values	169
How program zero assignment values are determined.....	170
Manually measuring the program zero point location on the machine.....	171
Measuring program zero assignment values in X and Y for a square or rectangular workpiece.....	171
Procedure to use a wiggler-type edge finder to measure X and Y axis program zero assignment values.....	172
Measuring program zero assignment values in X and Y for a round workpiece.....	174
Measuring program zero assignment values in the Z axis	176
Measuring the program zero point location using a spindle probe	176
Fixture offsets	177
An example of program zero assignment.....	179
The program.....	179
The setup for this example job.....	179
What is the common fixture offset?	180
What if my machining center doesn't have fixture offsets?	180
Advantages of assigning program zero with fixture offsets instead of G92	180
Safety	180
Ease of use.....	180
Efficiency.....	181
Rerunning tools.....	181
Assigning program zero in the program.....	181
Lesson Nine: Tool Length Compensation	185
The reasons why tool length compensation is needed.....	185
No two tools will have exactly the same length	185
A given tool's length will vary from one time it is assembled to the next	186
Tool length data is entered separately from the program	186
Sizing and trial machining	186
The setup person's responsibilities with tool length compensation	186
Which offset register is related to a given cutting tool?.....	187
Recommended method: Using the tool's length as the tool length compensation value.....	187
Determining tool length compensation values	188
Measuring tool lengths on the machine.....	188
1) Make the workholding setup – or place some kind of flat block on the machine table.....	188
2) Place a gauge block on the flat surface.....	188
3) Without a tool in the spindle, make the spindle face touch the block.....	188
4) Retract the Z axis and load a cutting tool to be measured. Bring the tool tip to the same block.....	188
5) Repeat step four for all cutting tools that must be measured.....	189
Measuring tool lengths off line – with a tool length measuring gauge.....	189
Using the distance from the tool tip to program zero as the tool length compensation (offset) value	190
1) Make the workholding setup and load a workpiece	191
2) Send the machine to the Z axis zero return position and reset (set to zero) the Z axis relative display.	191
3) Load a tool to be measured and manually bring the tip of the tool to the Z axis program zero surface.	191
Summary of differences between the two methods	192
Reasons for using our recommended method	192
Typical mistakes with tool length compensation	193
Forgetting to instate tool length compensation.....	193
Forgetting to enter the tool length compensation value.....	193
What if my machine doesn't have fixture offsets?.....	193
Relationship between the tool length compensation value and the machined depth	193
What is the polarity for an adjustment?.....	194
The plus input key.....	195
Trial machining with tool length compensation.....	195
Must the first workpiece be a good one?.....	195
Even when trial machining is not required.....	196
Sizing with tool length compensation	197
What happens when cutting tools get dull?.....	198
Reminder about changing the program to make sizing adjustments.....	199

Lesson Ten: Cutter Radius Compensation **205**

- Do you need to learn about cutter radius compensation? 205
- Similarities to tool length compensation 206
- Reasons why cutter radius compensation is required 206
 - Range of cutter sizes can be used..... 207
 - Do you use sharpened (re-ground) milling cutters?..... 208
 - Trial machining and sizing..... 209
 - Rough and finish milling with the same set of coordinates 209
- The setup person’s responsibilities with cutter radius compensation 210
 - Which offsets are involved? 211
 - The two ways to use cutter radius compensation..... 211
 - When the work surface is the programmed path..... 211
 - When the cutters centerline path is programmed..... 211
- Relationship between the cutter radius compensation value and the machined surfaces 212
 - What is the polarity for an adjustment? 212
- Trial machining with cutter radius compensation 212
 - Warning about milling multiple surfaces 214
 - How important is it that you make your first workpiece a good one? 214
 - When trial machining is not required..... 214
- Sizing with cutter radius compensation 215
 - Does the method of using cutter radius compensation affect the way sizing is done? 215
 - Do I have to make all these calculations when adjusting offsets?..... 215
 - Can you change programmed coordinate/s in the program to make sizing adjustments?..... 215
 - Dull tool replacement..... 215

Key Concept #4: Master Tasks To Run A CNC Machining Center **219**

Lesson Eleven: Tasks Required To Setup A CNC Machining Center **221**

- A CNC job from start to finish..... 222
 - Setup documentation..... 223
 - Tear down the previous setup and put everything away..... 224
 - Gather the components needed to make the setup..... 225
 - Make the workholding setup..... 225
 - Assign the program zero point..... 226
 - Measure program zero assignment values 226
 - Enter program zero assignment values into fixture offset number one 228
 - Assemble the cutting tools needed for the job 228
 - About the tightness of cutting tool components..... 232
 - Measure tool-length- and cutter-radius -compensation values 232
 - Reminder about the static nature of cutting tool measurements 233
 - Enter tool-length- and cutter-radius -compensation values into offsets 233
 - Load cutting tools into the machine’s automatic tool changer magazine..... 234
 - Warning about tool orientation..... 234
 - Load the CNC program 235
 - Physical tasks related to setup are now completed..... 235
 - Verify the correctness of a new or modified program 236
 - Verify the correctness of the setup..... 237
 - Dry running our example program 238
 - What if you do find a problem?..... 238
 - Rerunning a tool in the program..... 239
 - When do you want the machine to stop?..... 242
 - Procedure to restart the program from the beginning of a specific cutting tool..... 242
 - Cautiously run the first workpiece 242
 - The most dangerous time..... 243
 - The safe-approach procedure..... 243
 - Making sure the first workpiece is a good one..... 244
 - Machining the first workpiece in our example job..... 244
 - If necessary, optimize the program for better efficiency..... 246

If changes have been made to the program, save the corrected version of the program	247
Verifying a job that contains mistakes	248
When loading the program	251
The dry run to check for setup mistakes	252
Cautiously running the first workpiece	254
Lesson Twelve: Tasks Required To Complete A Production Run	259
Production run documentation	260
Tasks required for completing a production run	261
Remove the previous workpiece.....	261
Load the next workpiece.....	262
Activate the cycle	262
Monitor the cycle	263
Clean and de-burr the just-completed workpiece.....	263
Perform specified measurements	265
Which takes longer, the CNC cycle or the off-line tasks an operator must perform?	265
Make offset adjustments to maintain size for critical dimensions (sizing).....	265
Replace dull tools	267
Clean the machine.....	267
Preventive maintenance.....	267
Anything else you'll be expected to do?.....	268

Answers to practice exercises: 273

Index: 281

Preface

CNC machining centers are among the most popular forms of CNC metal-cutting machine tools. Just about every company that uses metal-cutting CNC machines has at least one machining center:

It is the intention of this text to introduce beginners and experienced shop people alike to setup and operation techniques needed for utilizing CNC machining centers. We will begin in a basic manner, ensuring that even newcomers to the machine shop environment will be able to follow the presentations. And we use a building blocks approach – so as you get deeper into the material – we will be adding to what you already know. When you're finished, you will have a thorough understanding of what it takes to setup and run a CNC machining center.

While we will mention a few programming words and commands from time to time, CNC programming is beyond the scope of this text. If you wish to learn how to create programs for CNC machining centers, we do offer a separate text that will teach you how to program (to get the CNC Machining Center Programming manual, please use the contact information on the back cover of this text).

We use a *Key Concepts* approach to presenting CNC. The Key Concepts allow us to minimize the number of major topics you must master in order to become proficient with CNC machining center usage. With this approach, there are four Key Concepts related to setting up and running CNC machining centers.

Beginners should concentrate most on understanding the points made early in each Key Concept. It is as important to know *why* you are doing things as it is to know how to do them. Concentrate first on the *whys*. It will be impossible for beginners to totally memorize and comprehend every technique used with machining centers the very first time it is presented or read. Rest assured that if you can understand the basic reasoning behind why each CNC feature is required, it will be much easier to master the use of the feature. Once this basic reasoning is understood, it will be easy to review the related material to extract specific details of how each CNC feature is used – so you can start putting your CNC machining center to good use.

For experienced CNC people, this text will provide you with alternatives. You'll be able to compare your current methods to our recommendations. If you have previous CNC machining center experience for other types of CNC controls, you will find it relatively easy to adapt what you already know to the most popular CNC control in the industry, the Fanuc control. If you have had experience with other types of CNC machine tools, such as turning centers or wire EDM machines, this text will help you adapt what you know to CNC machining centers.

As experienced people know, there are many ways to utilize CNC equipment. This text will show you one or two safe ways to accomplish your tasks. You can use your own common sense and past experience to develop your own style.

Prerequisites

Rest assured that this text will cover CNC machining centers from the ground up. We will assume that you have absolutely no previous experience with CNC. Indeed, we will not even assume you have had machine shop experience. That said, the more shop experience you have, the easier it will be to learn how to setup and run CNC machining centers. Here are some of the most important basic machining practice topics that are related to this text. (Again, we will be presenting these topics in Key Concept number one of this text. But if you currently know about them, you have a head-start.)

Basic machining practice experience

The more you know about basic machining practices, the easier it will be to learn how to setup and run a CNC machining center. If, for example, you have worked on conventional machine tools in the machine shop, much of what you already know can be directly applied to CNC machining centers.

But if you have no previous basic machining practice experience, don't worry. We will be introducing the related topics in Key Concept number one. But keep in mind that our intention will be to simply present enough information to allow you to understand upcoming presentations in this text. At the very least, you will need the help of experienced people in your company as you get started with a CNC machining center.

More likely, you'll need more training in this area. If you have no previous machining practice experience, we strongly recommend that you enroll in a training program related to basic machining practice in conjunction with reading this text. Basic machining practice courses can be found at your local technical/vocational schools, colleges, and universities.

Math

The word *numerical* in computer numerical control implies that numbers are highly involved with CNC. Indeed, CNC setup people and operators will be making countless calculations every day. However, most calculations are quite simple to make. The types of arithmetic calculations required for the typical CNC machining center include addition, subtraction, multiplication, and division.

What is more important is being able to accurately repeat simple calculations – without making any mistakes. We present the various arithmetic requirements in Key Concept number one.

Motivation

This should go without saying. We assume that you are motivated to learn. If you are highly motivated to learn about CNC machining centers, it will make your task much easier. Your motivation will help you overcome any obstacles you may have with learning the material in this text. With motivation, you'll stick to it until you understand.

Controls covered

Since the Fanuc control is the most popular CNC control available, specific presentations in this text are provided in Fanuc format. Keep in mind, however, that the Key Concepts approach we use throughout this text will make it possible for you to learn techniques that can be applied to just about any CNC control on the market. Also keep in mind that several control manufacturers claim that their controls are *Fanuc compatible*. These manufacturers include Yasnac, Mitsubishi Meldas, Tasnac, Mazatrol (EIA option), Haas, Fadal, and Flashcut (among others). Even if you do not have one of these controls, we truly feel that if you understand the basic concepts, and if you understand how specific techniques are applied to one particular control type, it will be relatively easy to apply what you know to just about any CNC machining center control being used today.

Limitations

Please take note that our first goal will be to acquaint you with the usage of three axis machining centers (having X, Y, and Z axes). For the bulk of this text, only these three linear axes will be discussed. Certain machining centers do have additional axes, and will be beyond the scope of this text.

We will be discussing both vertical as well as horizontal machining centers throughout this text. Since vertical machining centers are easier to work with and visualize, and since vertical machining centers are much more popular than horizontals, most of the examples given will be for vertical machining centers.

This text will not specifically address five axis machining centers. Though many of the same principles apply, we do not wish to present five axis machines during this introductory text.

The need for hands-on practice

We cannot overstress the need for practice on an actual machining center in order to fully master what you have learned in this text. If you are enrolled in a CNC course at a technical school, your instructor will probably provide hands-on practice with one of the machining centers in the school's lab.

If you are reading this text on your own, you will not get this needed practice. We cannot, of course, provide the experience you would get from practicing on a real CNC machining center in this text. No text can provide such practice. We can, however, provide you with the principles and concepts that you must understand before you can spend any meaningful time at the machine.

We relate this to learning how to fly an airplane. Before an aspiring pilot can spend any meaningful time in the cockpit, they must understand some basics of aerodynamics and flight. At the very least, they must understand the cockpit controls and what affects they have on the aircraft.

Just as flight school provides classroom studies to relate topics about aerodynamics, so must any study about CNC machine tools provide classroom-type study. Think of this text as a "flight school" for learning how to setup and run a CNC machining center. But when you are finished with this text, you will still need on-the-job training to fully master the use of a CNC machining center.

Instruction method

This text is commonly used with a formal CNC course. Possibly you are enrolled in a technical school's CNC course. Or maybe you are attending a company's in-plant CNC training course. In either case, you have an instructor who is making presentations and available to help you understand the material. Or you may be using this text in conjunction with a video or CD-rom course, meaning your instructor is recorded. Either way, audio presentations should help you understand the concepts more easily. And with video and CD-roms, you can review the material as often as necessary.

If you have purchased this manual separately and do not have the benefit of an instructor, your task will be a little harder. While all of the key points made during live presentations are included in this text, you will be left on your own to study hard enough to grasp the material presented.

Scope

As the name of this text implies, we address two of the three tasks a person must master in order to utilize a CNC machining center, including setup and operation.

Programming (which is beyond the scope of this text) is the act of preparing a series of commands that tell the CNC machining center how to machine a workpiece. It involves coming up with a machining process, selecting cutting tools, designing and documenting a setup, and actually creating the CNC program.

Setup is the act of preparing the CNC machining center to run a series of workpieces (called a *job* or *production run*). Tasks involve (among other things) making the workholding setup, assembling, measuring, and loading cutting tools, entering certain offsets, loading the program, and verifying that the program is correct. In this text, we will be discussing the related tasks in the approximate order that setups are made.

Operation actually involves two things. First, you must be comfortable with the general manipulation of a CNC machining center. This involves knowing the various components on the machine, its buttons and switches, and how to perform several important procedures. Second, and more importantly, you must be able to complete a production run once the setup is made. Tasks involve (among other things) workpiece load and unload, cycle activation, measuring completed workpieces and making sizing adjustments if necessary, and dull tool replacement.

Key Concepts approach

This effective presentation method will allow you to organize your thoughts as you read this text. This text includes four Key Concepts. Here are several benefits to this presentation method.

- 1) Any good training program should *put a light at the end of the tunnel*. All students want to know where they stand throughout any training course. With our Key Concepts approach, you will always have a clear understanding of your progress.
- 2) During each Key Concept, we will first present the main idea behind the concept. As stated earlier, we say it is at least as important to understand why you are doing things as it is to understand how to do them. Think of these early presentations for each Key Concept as the *why*. From there, we will present the specific techniques that are related to each concept.
- 3) The Key Concepts allow us to use a *building blocks approach* and present information in a very tutorial manner. We will be constantly building on previously presented information.
- 4) The Key Concepts approach allows us to limit the number of new ideas you must understand in order to grasp information presented within the text. Think of it this way: If you can understand but four basic ideas, you will be well on your way to becoming a proficient setup person or operator!

Lesson structure

We divide the four Key Concepts into twelve lessons. This makes it possible to further organize the most important topics related to machining center usage.

Practice makes perfect

In this edition, we've included lots of practice and exercises within the text to help you confirm your understanding of the presented material. Answers are provided right in the text, so you can quickly check your own work. If you are using this text in conjunction with a technical school's CNC courses, you may be asked to submit your exercises without checking your answers. We ask that you do so honestly.

Key Concepts and lessons

Here is a list of the four Key Concepts and the twelve lessons that comprise this text. Frankly speaking, Key Concept number one has little to do with CNC machining centers. Yet it is one of the most important Key Concepts. In Key Concept number one, we will help you get ready to learn about CNC machining centers. We will present many basic machining practice topics you must understand before you can begin working with any kind of metal-cutting CNC machine tool (including a CNC machining center). Indeed, these are important topics to anyone working in a machine shop environment.

Key Concept number one: Get ready to learn about CNC machining centers
 Lesson One: Basic machining practices required for machining centers
 Lesson Two: What does a CNC machining do?

Key Concept number two: Know your machine from a setup person's or operator's viewpoint
 Lesson Three: General flow of CNC process
 Lesson Four: Machine configurations
 Lesson Five: Understanding the control panels
 Lesson Six: Important procedures needed to run the machine

Key Concept number three: Know the three compensation types
 Lesson Seven: Understanding compensation
 Lesson Eight: Fixture offsets and assigning program zero
 Lesson Nine: Tool length compensation
 Lesson Ten: Cutter radius compensation

Key Concept number four: Master the tasks to run a CNC machining center
 Lesson Eleven: Tasks related to making setups
 Lesson Twelve: Tasks related to completing a production run

Enjoy!

We at CNC Concepts, Inc. wish you the best of luck with this text. We hope you find it easy to understand our written presentations and the presentations of your instructor (live or recorded). Once completed, we hope this text makes your introduction to CNC machining centers as easy and enjoyable as possible.

Lesson 1**Basic Machining Practices Required For CNC Machining Centers**

The more a person knows about the basic machining practices that apply to CNC machining centers, the easier it will be to become a proficient setup person or operator. We cannot over stress the importance of understanding shop practices needed for successful machining center use.

In this lesson, we introduce you to some of the most important principles you must understand *before* you begin learning about CNC machining center usage. The more you know about these topics, the easier it will be to learn to program, setup, or operate a CNC machining center.

There are five basic-machining-practice-related topics discussed in this lesson:

- ✍ Shop safety
- ✍ Shop math
- ✍ Blueprint reading
- ✍ Tolerance interpretation
- ✍ Measuring tools

Any one of these topics could probably fill a manual larger than this one. So keep in mind that we are merely *introducing* you to each topic. Our intention is to provide enough information so that a newcomer to the machine shop environment will understand the presentations made throughout this text. But this will not be enough to fully master any of these topics.

If you are a novice, you will surely need more than this text offers about basic machining practices. At the very least, you will need the help of an experienced machinist when you begin working in the shop. You can also learn more by reading books on each of these topics – or you can attend related courses offered by technical schools.

Shop Safety

A machine shop is a very dangerous place. There are hazards everywhere – and you must be very careful with everything you do and everywhere you go. While we offer some basic suggestions for keeping safe, we cannot possibly address every hazard in a machine shop. Most companies have safety instruction as part of the orientation for new-hires. We bow to an experienced person in your company or school to provide more specific safety instruction than what we show in this text. But at least this presentation should make you aware of some of the most important safety issues.

Safety equipment

Protective eyewear – most shops demand that you wear safety glasses at all times when you are in the shop – regardless of whether you are actually working on a machine or not. Most companies have signs on every shop entry door that say: **“Safety glasses required beyond this point.”** In areas where debris may be flying everywhere (like welding and grinding departments), people are often required to wear full-face shields.

Clothing – The clothing you wear when working in a machine shop has an impact on your safety. Most shops require that you wear long sleeve shirts and long pants (no tee-shirts or shorts) to protect your arms and legs from simple hazards. Your clothes should be close-fitting – don’t wear any loose fitting clothes that could get snagged or hooked in the shop – or worse – grabbed by a moving machine part. Neck ties are banned from most machine shops. Remove all jewelry (finger-rings, ear-rings, bracelets, necklaces, etc.) when

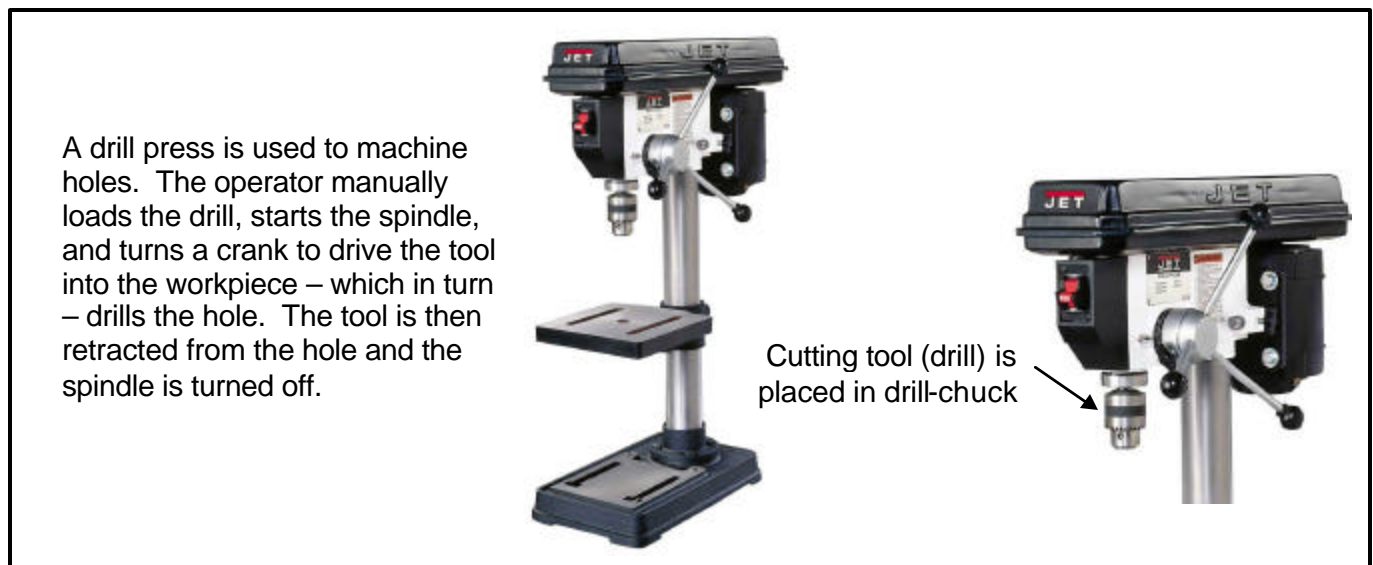
Lesson 2**What Does A CNC Machining Center Do?**

A CNC machining center is the most popular type of metal cutting CNC machine because it is designed to perform some of the most common types of machining operations. It is important to understand these machining operations in order to properly use a CNC machining center.

In this lesson, we will describe the function of CNC machining centers. We will start by comparing CNC machining centers to other types of machines with which you may have some experience. We will then introduce the cutting conditions that are related to machining operations. Finally, we will describe – in detail – the two most basic kinds of machining operations that can be done on machining centers, including hole machining operations and milling operations. At the completion of this lesson, you will know what a CNC machining center is designed to do.

Comparing a CNC machining center to other types of machines

CNC machining centers replace certain *conventional* machine tools. While you may not have experience with CNC machining centers, it is likely that you have had experience with at least one of the conventional machines they replace: the drill press. At the very least, you probably know what a drill press is and/or have seen one in use. If so, you know that the primary function of a drill press is to drill holes. The next illustration shows a typical home-shop drill press.



A typical drill press

A drill-bit is but one of many cutting tools that can be used to machine holes. And while a drill press can be used to machine holes with other types of cutting tools, most people in a home-shop don't do much more than drilling operations. We'll discuss other types of hole-machining tools a little later.

Lesson 3**General Flow Of The CNC Process**

The tasks of programming, setup, and operation are but three of the things that must be done in order to actually get a CNC job up and running. It really helps to understand how these tasks fit into the bigger picture of a company's manufacturing environment.

CNC machine tools are being used by all sorts of companies. Indeed, if a company manufactures anything, it is likely that they are using at least some CNC machine tools. With the diversity of companies and applications, there comes diversity with what is expected of CNC people. Understanding where your company fits in to this diverse group should help you understand what will be expected of you.

Companies that use CNC machining centers

There are many factors that contribute to how a CNC-using company applies its CNC machining centers. These factors include (among others) lot sizes, lead times, percentage of new jobs, size of tolerances held, materials machined, and company type. The most important of these factors is company type.

When it comes right down to it, there are only four types of companies that use CNC machine tools:

- ✍ **Product producing companies** – get revenue from the sale of a product
- ✍ **Workpiece producing companies** – (also called job-shops or contract-shops) get revenue from the sale of workpieces to product producing companies
- ✍ **Tooling producing companies** – get revenue from the sale of manufacturing tooling (fixtures, jigs, molds, dies, gauges, cutting tools, etc.) to product producing and workpiece producing companies
- ✍ **Prototype producing companies** – get revenue from the sale of prototypes to product producing companies

There are also overlaps in company type. For example, some product producing companies have a tool-room in which CNC machine tools are used – or they may have a research and development department that produces prototypes. Some workpiece producing or tooling producing companies have a product of their own.

While there will be exceptions to what we say here, some pretty good generalizations can be made based upon the company type alone, especially when it comes to what CNC people will be doing.

Product producing companies tend to have more resources than workpiece producing, tooling producing, and prototype producing companies. Since their profit is one step removed from manufacturing (a product won't come to market unless the company can make a profit), they tend to engineer all facets of the manufacturing environment. For this reason, they commonly break up the tasks related to CNC machine tool usage. People will specialize in the tasks they perform.

This will maximize machine tool utilization. It ensures that machines are running for as high a percentage of time as possible. Many CNC tasks will be performed while the machine is running production (like programming for upcoming jobs, gathering components needed for future setups, and assembling cutting tools, among others). So while a CNC operator is running a job on a CNC machine, other people are getting ready to run the next (and other upcoming) jobs. Again, this minimizes the amount of time that the CNC machining center is down between production runs.

On the other hand, workpiece producing, tooling producing, and prototype producing companies tend to charge an hourly rate for machine usage time. Their resources will be much smaller – and they will require

Lesson 4

Machine Configurations

While you don't have to be a machine designer in order to run a CNC machining center, it helps to understand the configuration of the machine/s you'll be working with. At the very least, you must be able to recognize the major components and be able to identify the moving components (called axes).

Most beginners tend to be a little intimidated when they see a machining center in operation for the first time. Admittedly, there will be a number of new functions to learn. The first point to make is that you must not let the machine intimidate you. As you go along in this text, you will find that a machining center is very logical and is easy to understand with proper instruction.

As stated in Lesson Two, you can think of any CNC machine as being much like than the standard type of equipment it is replacing with very sophisticated and automatic motion control added. Instead of activating things manually by hand-wheels and manual labor, a *program* tells the machine what to do. Virtually anything that needs to be done on a machining center can be activated through a program.

There are two basic types of machining centers that we will be addressing in this text. They are vertical machining centers and horizontal machining centers. Because they are the more popular of the two, we'll be placing the emphasis on vertical machining centers. Let's start by describing the most common components of each.

Vertical machining centers

The classifications *vertical* and *horizontal* refer to spindle orientation. A vertical machining center has its spindle oriented in the vertical position (just as it is with the drill press and knee-style milling machine introduced in Lesson Two). The cutting tool will be pointing downward toward the machine's table. Indeed, a vertical machining center is such a popular type of CNC machining center because it closely resembles a knee mill – and again – a knee mill is a very popular conventional machine tool. For anyone having experience with a knee mill, a vertical machining center should be pretty familiar.

C-frame style

Figure 4.1 shows a C-frame style vertical machining center.

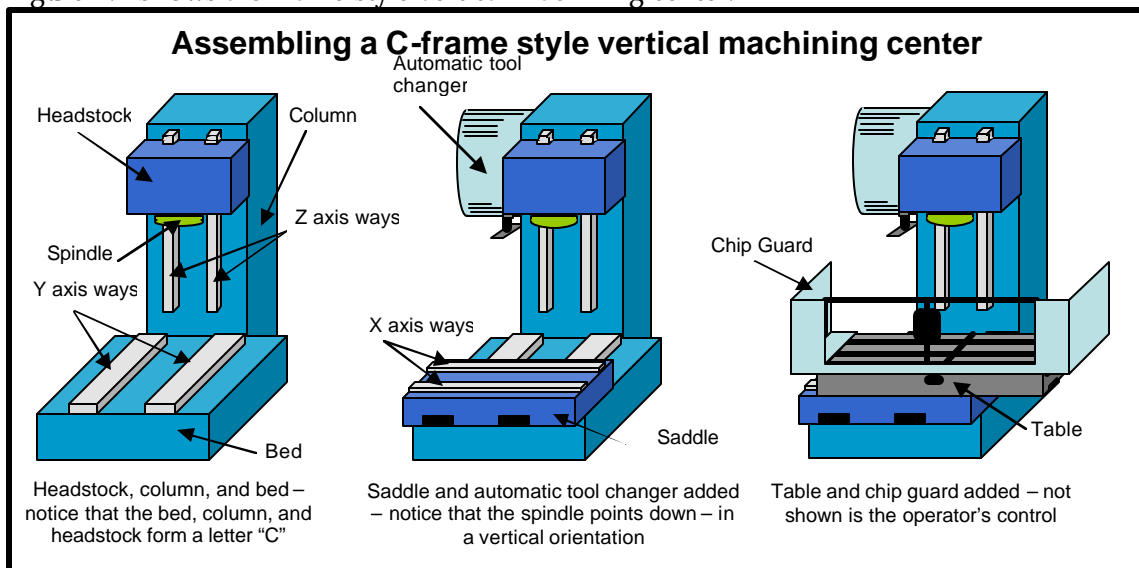


Figure 4.1 – Components that make-up a C-frame style vertical machining center

Lesson 5**Buttons And Switches On The Operation Panels**

While there are many buttons and switches on a CNC machining center, you must try to learn the reason why each one exists. If you don't, you may be overlooking a helpful – if not necessary – machine function. Worse, you may have a switch in the wrong position and not even know it.

You now know the configurations for the most popular types of CNC machining centers. You know the most basic components, the axes, and the programmable functions. However, we have not addressed some very important machine components – the operation panels. These panels provide the access you need to activate machine functions, and are the topic of this lesson.

*A proficient setup person or operator knows the function of **all** buttons and switches on their machine/s. While some may be seldom or never used, you must not consider yourself fully capable of running a machine until you know the function of all buttons and switches. If we don't cover a given button or switch in this text, ask someone in your company what it is – or look it up in the operation manual. Don't stop until you know why every button and switch is on your machining center.*

If you are just starting out, this may be quite a challenge. You will be striving just to learn the most important buttons and switches. Learning all buttons and switches may seem like too much to ask. While we sympathize, we cannot stress enough the importance of learning all of the access you have to activate machine functions. Again, don't stop until you know all the buttons and switches on your machine/s.

The two most important operation panels

Most machining centers have at least two distinct operation panels. We'll be calling them the *control panel* (designed by the control manufacturer) and the *machine panel* (designed by the machine tool builder).

For machining centers that have a Fanuc control (Fanuc is by far the most popular control in the industry), the control panel will be remarkably similar from one machining center to the next. Indeed there are few variations, even among different Fanuc control models.

But since the machine panel is designed by the machine tool builder, machining centers that have been manufactured by different machine tool builders will have substantially different machine panels. To compound this problem, machine tool builders can't seem to agree on the specific functions needed by CNC setup people and operators to run their machines. So one machine may have an important button or switch while another may not.

While we can be pretty specific about the function of buttons and switches found on the (Fanuc) control panel, we will be a little vague about machine panel buttons and switches. Also, there may be buttons and switches on your machining center's machine panel that we do not cover in this text. If you find one, be sure to ask an experienced person or reference the machine tool builder's operation manual to determine the function for the button or switch.

The control panel buttons and switches

Figure 5.1 shows the control panel for one popular control model. Again, this operation panel is made by the control manufacturer (Fanuc in our case). For the most part, it is used in conjunction with the display screen (on the left side of this control panel).

Notice the power buttons on the left side. The power-on button is usually the *second* switch used to power up the machine (the first is usually a main breaker switch that is placed behind the machine). The power-off button is used, of course, as part of the procedure to turn off the machine.

Lesson 6**The Key Operation Procedures**

Step-by-step procedures can keep you from having to memorize every function that you must perform on your CNC machining center. You will soon memorize procedures for task that you perform on a regular basis – but written procedures will always help you perform lesser used tasks.

While you have been introduced to the various buttons and switches on a typical machining center, it is unlikely that you have memorized them all. It is also unlikely that you know the appropriate order by which you should press buttons and activate switches to do anything on the machine.

For example, think of what it takes to perform one of the most basic tasks a setup person or operator must do – to power up the machine. Just because you know the related buttons and switches, does not mean you know the step-by-step *procedure* needed to turn on a CNC machining center. The procedure for a given machining center might look something like:

- ? Step 1: Turn on the main breaker (in back of the machine).
- ? Step 2: Press the power on button on the control panel.
- ? Step 3: Press the hydraulic system on button on the machine panel.
- ? Step 4: Follow the procedure to do a manual zero return.

With such a procedure, turning on the machine is not at all difficult. While you should still know the function of the related buttons and switches, this procedure provides you with sequential order (steps) you need. And similar procedures for other important tasks will be just as easy to follow (if you have them).

Admittedly, the most often used procedures will be soon memorized – and because most experienced setup people and operators have them memorized, they haven't written them down. And unfortunately, most companies don't have a written set of procedures for their CNC machines. Though this may be the case, nothing stops you from developing a written set of procedures for yourself. While you'll need the help of an experienced person, you can easily develop a set of procedures that will help you (and others) remember the steps necessary to do just about anything on the machine.

We divide the procedures needed to run a CNC machining center into five categories:

- ✍ Manual procedures
- ✍ Setup procedures
- ✍ Manual data input (MDI) procedures
- ✍ Program editing procedures
- ✍ Program operation procedures

Here are specific procedures that we recommend you write down in each category:

Manual procedures:

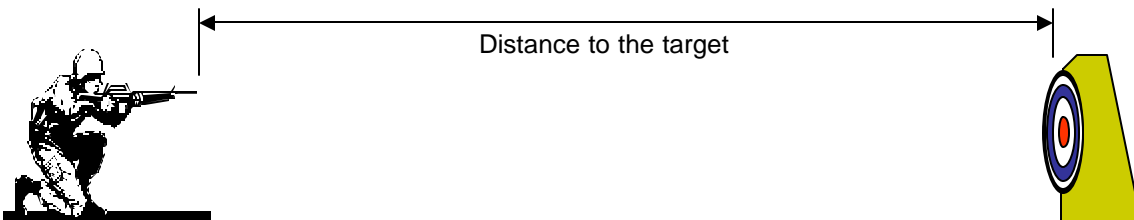
- To power up the machine
- To do a manual zero return
- To manually start the spindle
- To manually jog the axes
- To use the handwheel
- To manually load and remove cutting tools in the spindle

Lesson 7**Introduction To Compensation**

An airplane pilot must compensate for wind direction and velocity when setting a heading. A race-car driver must compensate for track conditions as they negotiate a turn. A marksman must compensate for the distance to the target when firing a rifle. And a CNC programmer must compensate for certain tooling-related concerns as programs are written. Setup people and operators must address these compensations – which in turn – will marry the CNC program to the physical setup that is being made.

What is compensation and why is it needed?

When you compensate for something, you are allowing for some unpredictable (or nearly unpredictable) variation. A *race car driver* must compensate for the condition of the race track before a turn can be negotiated. In this case, the variation is the condition of the track. An *airplane pilot* must compensate for the wind direction and velocity before a heading can be set. For them, wind direction and velocity are the variations. A *marksman* must compensate for the distance to the target before a shot can be fired – and the distance to the target is the variation. The marksman analogy is remarkably similar to what happens with most forms of CNC compensation. Let's take it further...



Before a marksman can fire a rifle, they must judge the distance to the target. If the target is judged to be one-hundred-fifty yards away, the sight on the rifle will be adjusted accordingly. When the marksman adjusts the sight, they are *compensating for the distance to the target*. But even after this preliminary adjustment and before the first shot is fired, the marksman cannot be *absolutely* sure that the sight is perfectly adjusted. If they've incorrectly judged the distance – or if some other variation (like wind) affects the adjustment – the first shot will not be perfectly in the center of the target.

After the first shot is fired, the marksman will know more. If the shot is not perfectly centered, another adjustment will be needed. And the second shot will be closer to the center of the target than the first. Depending upon the skill of the marksman, it might be necessary to repeat this process until the sight is perfectly adjusted.

With *all* forms of CNC compensation, the setup person will do their best to determine and enter certain *compensation values* needed to perfectly machine the workpiece (just as the marksman does their best in judging the distance to the target and adjusting the sight). But until machining actually occurs, the setup person cannot be absolutely sure that their initial compensation values are correct. After machining, they may find some imperfections in their initial entries. Or they may find that another variation (like tool pressure) is causing the initial adjustment to be incorrect. Depending upon the tolerances for the surfaces being machined, a second adjustment will probably be required. After this second adjustment, machining will be more precise.

Lesson 8

Fixture Offsets And Assigning Program Zero

The programmer will choose an origin for the program – which is called the program zero point. The use of a program zero point simplifies the task of programming; it is usually the setup person who must determine where the program zero point is located in the setup – and who must enter this position in fixture offsets.

You know that a CNC machining center has three linear axes: X, Y, and Z. It may additionally have one or two rotary axes. The CNC program tells the machine how to move these axes in a way that causes a cutting tool to machine the workpiece. To this end, the programmer specifies a series of *coordinates* through which the tool will move. Coordinates are simply positions specified within the program.

Think of the global positioning satellite (GPS) system in an automobile. It works by monitoring the car’s current position on the earth. And this position is specified with coordinates – using longitude and latitude.

With CNC machines, coordinates are specified in each axis to cause a positioning movement. Consider these X and Y coordinates:

X0.5 Y0.5

These coordinates may be specifying the position at which a hole must be drilled – and they send the machine to a position of 0.5 in each axis (X and Y). When this command is completed, the center of the spindle will be positioned directly above the hole-location. But in order to cause such a motion, the machine must know the *origin* for the coordinates – that is, the location from which the coordinates are taken. In CNC terms, we call this origin location the *program zero point*.

How is the program zero point determined?

It is the programmer who determines the position of the program zero point – and most programmers use a logical method for deciding where it is placed. The method is based upon how the blueprint is dimensioned. They will pick the location in each direction (axis) from which the dimensions begin. Consider the drawing shown in Figure 8.1 that shows an example of how program zero is determined in the X and Y axes.

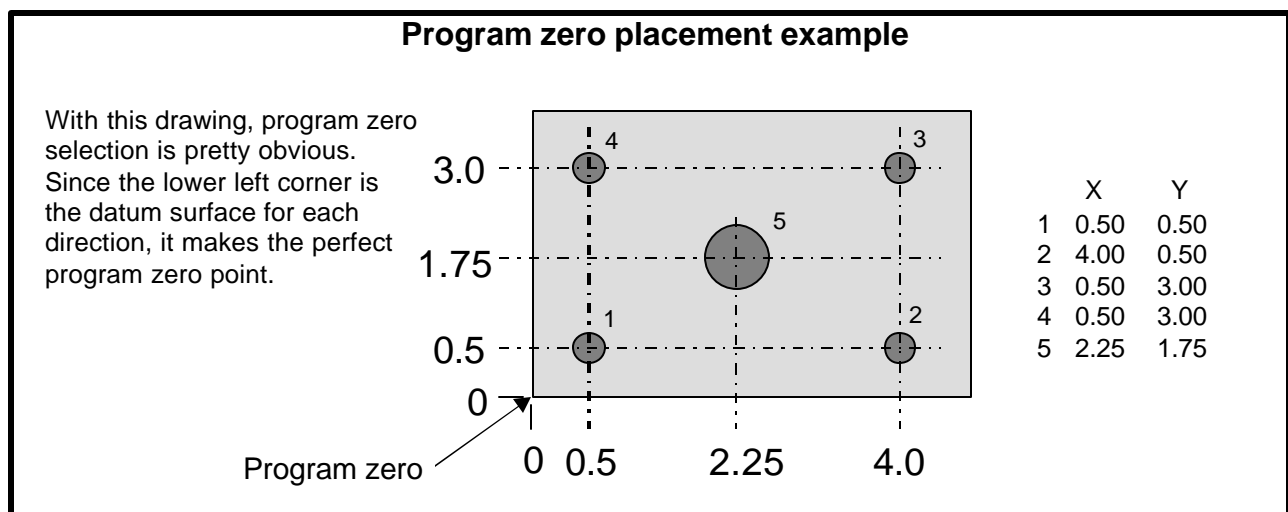


Figure 8.1 – An example of program zero point placement in the X and Y axes

Lesson 9**Tool Length Compensation**

Tool length compensation allows a programmer to ignore the precise length of each tool as a program is written. Each tool's length will be determined during (or just prior to) setup. The setup person is often responsible for measuring each tool's length and entering the length value into the machine's offset table. The operator must be able to repeat this process when dull cutting tools are replaced. Additionally, tool length compensation allows setup people and operators to make sizing adjustments to control the precise depth to which cutting tools machine.

You know from Lesson Seven that tool length compensation is one of the three compensation types – and that it is used for every cutting tool in every job. You also know that the tool length compensation value for each tool is placed in a tool offset. And you know that tool length compensation will allow you to make sizing adjustments to control the depth to which a cutting tool will machine. In this lesson, we will be taking an even closer look at this very important CNC feature.

The reasons why tool length compensation is needed

Cutting tools used on machining centers differ from one to the next. For one thing, and as you know from Lesson Two, there are a variety of cutting tool types that are used on machining centers, including (among others) center drills, spot drills, drills, taps, reamers, boring bars, end mills, and face mills. Each type of cutting tool requires its own type of tool holder. Most straight-shank tools use a collet holder. Others (like end mills) use a tool holder having a set-screw to clamp the cutting tool in place. Yet others (like face mills and taps) require a very special style of tool holder – designed specifically for the cutting tool type.

No two tools will have exactly the same length

Given the assortment of cutting tools available for use on CNC machining centers, it is unlikely that any two tools used by a program will be exactly the same length. And as the program is written, the programmer will not know the precise length of any given tool. Figure 9.1 illustrates this point with five different tools.

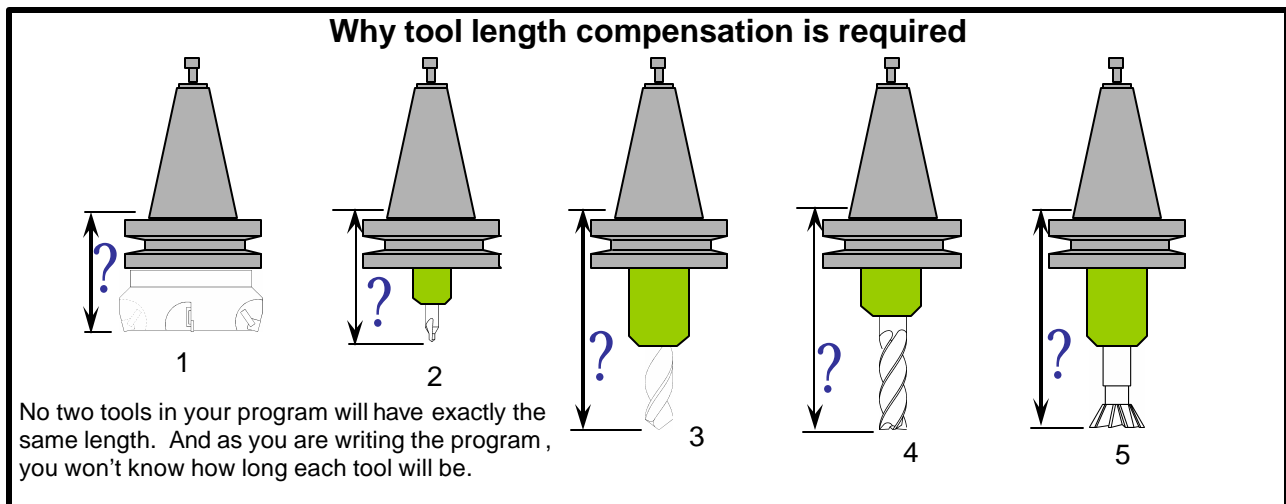


Figure 9.1 – Five cutting tools that might be used in by a CNC machining center program

The most important reason for tool length compensation has nothing to do with setup and operation. Tool length compensation will allow a programmer to write programs even though they don't know how long the cutting tools will be. But there are other reasons why tool length compensation is required – and most do apply to setup people and operators.

Lesson 10**Cutter Radius Compensation**

Cutter radius compensation is only used for milling cutters. Just as tool length compensation lets the programmer ignore the precise length of cutting tools as they write programs, cutter radius compensation allows them to ignore the precise diameter of milling cutters used for contour milling. And also like tool length compensation, setup people and operators must enter and manipulate the related offsets – during the tools initial setup, and for trial machining and sizing.

You know from Lesson One that milling cutters (like end mills) can be used for *side-milling* operations. With a side-milling operation, the milling cutter is machining on its *periphery* (around its diameter). Figure 10.1 shows some examples.

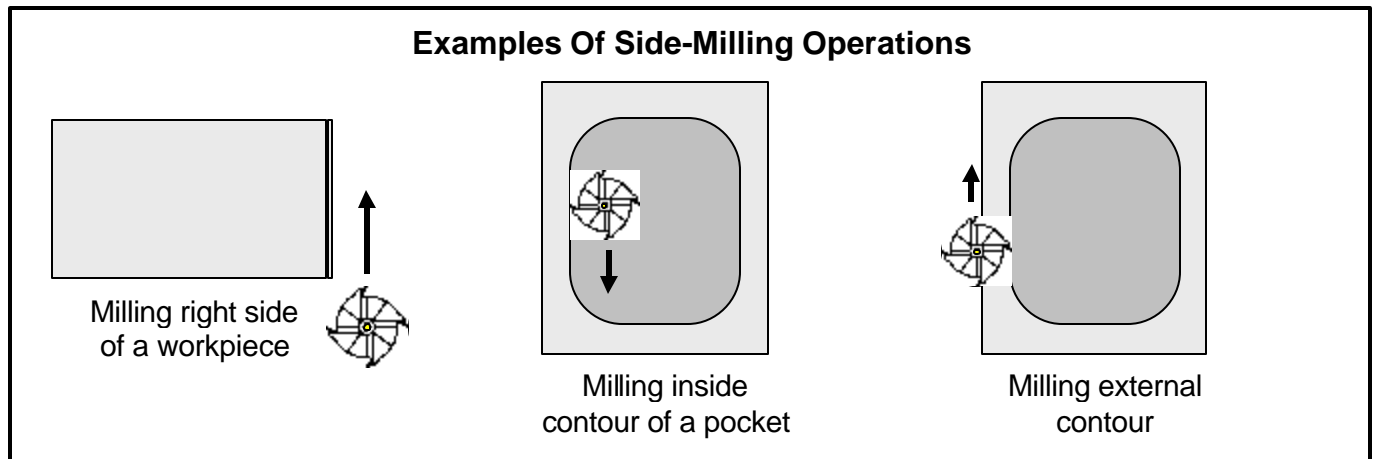


Figure 10.1 – Side-milling with an end mill

Some people refer to these side-milling operations as *contour milling* operations. But as you can see from Figure 10.1, a milling cutter may be milling but one straight surface – and it is still be considered to be a side-milling operation.

Do you need to learn about cutter radius compensation?

Unlike tool length compensation – which is used for every cutting tool in every program – cutter radius compensation is only used for milling cutters, and only when side-milling. Some companies do not perform side-milling operations – they may, for instance, only perform hole-machining operations on their machining centers. In this case, there will not be much of a need to learn about cutter radius compensation.

Even if you work for such a company, you will still want to know the reasons *why* cutter radius compensation is required for side milling. So at the very least, skim this lesson to make sure you know what is included. If the need ever arises, you can always come back to this lesson and dig in.

Lesson 11**Tasks Required To Setup A CNC Machining Center**

We define setup time as the total time a machine is down between production runs. We define cycle time as the total time it takes to complete a production run divided by the number of good workpieces produced. If you think about it, there are only two general tasks that occur on CNC machining centers – machines are either in setup or they are running production. In this lesson, we will address those things that occur while the machine is down between production runs.

It is important to understand the distinction between making a setup and running production. The tasks you perform during setup are *getting the machine ready* to run production. Only when the setup is completed and after a workpiece has passed inspection is it possible to run production. (Note that that we include the running of the first good workpiece – and its inspection – as part of setup.) The person making the setup is called the *setup person* – the person completing the production run is called the *CNC operator*. In many companies – especially many tooling- and workpiece-producing companies – the same person makes the setup *and* completes the production run.

Here is a general list of tasks that are related to setup. We will be presenting them in detail during this lesson. Note the order we use to list the tasks. This is the approximate order by which a setup is made (though each setup person will have his or her own way of doing things).

Setup-related tasks

- ✍ Tear down previous setup and put everything away
- ✍ Gather the components needed to make the setup
- ✍ Make the workholding setup
- ✍ Assign program zero point/s
- ✍ Assemble cutting tools
- ✍ Measure tool length and cutter radius compensation values
- ✍ Enter tool length and cutter radius compensation values
- ✍ Load cutting tools into the machine's automatic tool changer
- ✍ Load the CNC program/s
- ✍ Verify the correctness of the program and setup
- ✍ Cautiously run the first workpiece – ensure that it passes inspection
- ✍ If necessary, optimize the program for better efficiency (new programs only)
- ✍ If changes to the program have been made, save the corrected version of the program

Some of these tasks should sound familiar. We have been addressing bits and pieces of what is required to make setups as part of our building-blocks approach throughout this text. Many of these general tasks require more specific tasks to be done. *Assign program zero point/s*, for example, will often require the setup person to first measure the program zero assignment values and then enter them into the appropriate fixture offset. In this lesson, we intend to *put-it-all-together*, providing you with a logical flow for making setups. Some of the material in this lesson has been presented before – especially material that is related to the more complicated topics. Consider repeated material as review. It should help to solidify your understanding.

Lesson 12**Tasks Required To Complete A Production Run**

Once a job is set up and the first good workpiece is efficiently machined, the rest of the workpieces must be run. Completing a production run is the job of a CNC operator – though the same person who sets up the machine is often the person who completes the production run. A misconception exists about the difficulties related to completing a production run. Many manufacturing people feel that running out a job simply requires part loading, cycle activation, and workpiece removal. In reality, there is usually a lot more to it.

With an understanding of what it takes to set up a CNC machining center, let's turn our attention to what it takes to run the rest of the workpieces in the job. Here is a list of the related tasks:

Done during every cycle:

- ✍ Load a workpiece
- ✍ Activate the cycle
- ✍ Monitor the cycle to ensure that cutting tools are machining properly (first few workpieces only)
- ✍ Remove the workpiece
- ✍ Clean/de-burr the workpiece
- ✍ Perform specified measurements (if required)
- ✍ Report measurement results to statistical process control (SPC) system

Not required in every cycle:

- ✍ Make sizing adjustments for critical workpiece attributes
- ✍ Replace worn tools
- ✍ Remove chips from work area (if required)
- ✍ Machine maintenance

At first glance, the tasks related to completing a production run probably look pretty simple. And as long as everything is going smoothly, keeping the machine running good workpieces may be as simple as loading the workpiece, pressing the cycle start button, and removing the completed workpiece when the cycle is completed. This may be the case for the first few workpieces machined in a proven job (one run before).

It is quite common, however, that operators must make adjustments during the production run to ensure that workpiece are produced consistently. The more workpieces in the production run, the tighter the tolerances, and the more abrasive the workpiece material, the more likely it will be that adjustments must be made. The frequency of these adjustments (how often they must be made) is also related to these factors.

In this lesson, we will be taking a close look at what it takes to complete a production run once the setup is completed. As we did in Lesson Eleven, we will present the related tasks in the approximate order that a production run is completed.

Remember from Lesson Eleven that there are only two things that occur on a CNC machining center. The machine is either in setup or running production. In this lesson, of course, we will be addressing thing that occur during the production run. And this includes just about anything that happens between setups.

- 1) All coordinates in a CNC program (positions through which a cutting tool will move) will have a polarity. An *origin* (zero point) will be designated for each motion direction (axis). Any position on one side of this origin point will be positive. Any position on the other side of the origin point will be negative.
- 2) Adjustments made to keep a cutting tool machining correctly will also have a polarity. If more material must be removed from a workpiece surface in order to make it on size, the adjustment polarity will probably be negative. If less material must be removed, the adjustment will probably be positive.

Summary of shop math

As you've seen, the only arithmetic you'll be doing on a regular basis— as a CNC operator or setup person — is addition, subtraction, multiplication, and division. It's not overly difficult. The trick is in knowing the appropriate formula to apply for your particular problem/s. We'll be discussing each formula as it's needed. If you'll be programming CNC machining centers, there may be the additional need to learn *right angle trigonometry*. We do include some trigonometry discussions in our CNC Machining Center Programming manual (sold separately).

Shop Math Quiz

- 1) All that is required of an electronic calculator for most CNC operators is addition, subtraction, multiplication and division.
 True False
- 2) Which of these has the highest priority (must be done first) in an arithmetic expression?
 a. Addition and subtraction
 b. Multiplication and division
 c. Anything in parentheses
 d. equality
- 3) Provide answers for the following expressions:
 a. $(3 + 4) \times 5 =$ _____
 b. $3.82 \times 300 / 2.5 =$ _____
 c. $4 \times 0.002 \times 500 =$ _____
 d. $1.0 / 2 + 4.5 =$ _____
- 4) Round these values to four places:
 a. 0.234375 _____
 b. 0.296875 _____
 c. 0.71875 _____
 d. 0.140625 _____
- 5) Given the formulae below, provide answers to the questions that follow:

$$\text{rpm} = 3.82 \times \text{sfm} / \text{tool diameter}$$

$$\text{inches} = \text{millimeters} / 25.4$$

$$\text{millimeters} = \text{inches} \times 25.4$$
 a. You look in the tooling manufacturer's manual, and they say to run a 0.75 diameter drill at 80 sfm for the material you must machine. What will the required rpm be?

 b. Convert 300.0 mm to inches.

 c. Convert 4.5 inches to millimeters.

- 6) Convert these fractions to decimal format:
 a. $3/16 =$ _____
 b. $21/64 =$ _____
 c. $15/16 =$ _____
 d. $1/2 =$ _____

Answers: 1: True, 2: c, 3a: 35, 3b: 458.4, 3c: 4.0, 3d: 5.0, 4a: 0.2344, 4b: 0.2969, 4c: 0.7187, 4d: 0.1406, 5a: 407 rpm, 5b: 11.8110 inches, 5c: 114.3 millimeters, 6a: 0.1875, 6b: 0.328125, 6c: 0.9375, 6d: 0.5

Basic Machining Practices

Exercise 1

Name: _____

Date: _____

Score (100 possible): _____

5 points each (answers begin on page 273)

Use this drawing to answer the questions below.

Technical drawing of a top mounting plate. The front view shows a rectangular plate with a total width of 3.00 and a total height of 5.00. A rectangular pocket is located in the center, with a width of 1.50 and a depth of 0.87. The distance from the left edge to the start of the pocket is 0.75, and the distance from the end of the pocket to the right edge is 1.250. The pocket has a depth of 0.750. Two holes, each with a diameter of 0.50, are located in the lower portion of the plate, with a center-to-center distance of 1.50. The distance from the bottom edge to the center of the holes is 0.75. The corners of the plate are rounded with a radius of 0.37 R (4). The side view shows a total height of 4.00 and a top width of 1.00. The distance from the top edge to the start of the pocket is 0.3750. The distance from the top edge to the center of the holes is 0.750. The distance from the center of the holes to the bottom edge is 0.875 with a tolerance of +0.001, -0.002. A feature control symbol (A) is located near the bottom edge.

Revisions	
Rev	Description
A	Was +/- 0.001

Notes:
 1) Break all sharp edges.
 2) Similar part: A -37625.

Implied tolerances:
 x.x: +/-0.01
 x.xx: +/-0.005
 x.xxx: +/-0.001
 x.xxxx: +/-0.0005

Top Mounting Plate	A-37625
Drawn: MCH	Checked: LDA
Date: 4/03/08	Assembly: none
Material: 316 stainless	Scale: none

1) Specify the overall size of this workpiece (thickness, width, and height).

2) What is the distance between the two 0.5 diameter holes?

3) What is the size of the rectangular pocket (width, length, and depth)?

4) What material will this workpiece be made from?

5) Do the 0.5 diameter holes go all the way through the workpiece?

6) All tolerances but one are implied.

- true
- false

7) What is the tolerance for the rectangular pocket's width and length (not depth)?

8) For the slot width (0.750 dimension), specify the mean value, high limit, and low limit.

mv: _____ hl: _____ ll: _____

9) For the hole diameters (0.50 dia. dimension), specify the mean value, high limit, and low limit.

mv: _____ hl: _____ ll: _____

10) For the rectangular pocket depth (0.875 dimension), specify the mean value, high limit, and low limit.

mv: _____ hl: _____ ll: _____

11) Unless told otherwise, you should use the mean value as your target value when making adjustments.

- true
- false

12) What is the current revision (letter) for this drawing?

13) After removing a completed workpiece from the machine, you measure the width of the 0.750 slot and find it to be 0.7503.

- a. Is it acceptable? _____
- b. Is an adjustment necessary? _____
- c. If so, how much? _____

14) After removing a completed workpiece from the machine, you measure the depth of the 0.750 slot and find it to be 0.3743.

- a. Is it acceptable? _____
- b. Is an adjustment necessary? _____
- c. If so, how much? _____

15) After removing a completed workpiece from the machine, you measure the width of the rectangular pocket and find it to be 1.252.

- a. Is it acceptable? _____
- b. Is an adjustment necessary? _____
- c. If so, how much? _____

16) After removing a completed workpiece from the machine, you measure the length of the rectangular pocket and find it to be 2.002.

- a. Is it acceptable? _____
- b. Is an adjustment necessary? _____
- c. If so, how much? _____

17) After removing a completed workpiece from the machine, you measure the depth of the rectangular pocket and find it to be 1.8737.

- a. Is it acceptable? _____

b. Is an adjustment necessary? _____

c. If so, how much? _____

18) After removing a completed workpiece from the machine, you measure the distance between the two 0.5 diameter holes and find it to be 1.503.

a. Is it acceptable? _____

b. Is an adjustment necessary? _____

c. If so, how much? _____

19) When adjustments are necessary, adjustment polarity (plus or minus) is determined by whether or not more material must be removed from the workpiece. If more material must be removed, the polarity for the adjustment will usually be negative.

- true
- false

20) When a workpiece attribute is out of tolerance, it can usually be saved if more material must be machined in order to bring the attribute within its tolerance band.

- true
- false

Answers to Exercises

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

Exercise 1: Basic Machining Practices

- | | |
|--|---|
| 1) 1.00 by 3.00 by 5.00 inches (or 1 by 3 by 5 inches) | 14) a: Yes, b: Not immediately, but it is getting close to it's low limit – so if you said yes, you are also correct, c: deviation from target (mean) value is 0.0007 |
| 2) 1.50 | 15) a: Yes, b: No, c: deviation from target (mean) value is 0.002 |
| 3) 1.25 by 2.0 by 0.875 | 16) a: No, b: Yes, c: 0.002 |
| 4) 316 stainless steel | 17) a: No, b: Yes, c: 0.0013 |
| 5) yes | 18) a: Yes, b: No, c: deviation from target (mean) value is 0.003 (note that there is nothing a setup person or operator could do (without changing the program) to adjust the distance between the two holes). |
| 6) true | 19) true |
| 7) Plus or minus 0.001 | 20) true |
| 8) mv: 0.75, hl: 0.751, ll: 0.749 | |
| 9) mv: 0.50, hl: 0.505, ll: 0.495 | |
| 10) mv: 0.8745, hl: 0.876, ll: 0.873 | |
| 11) true | |
| 12) Revision “A” | |
| 13) a: Yes, b: No, c: deviation from target (mean) value is 0.0003 | |

Exercise 2: What Does A CNC Machining Center Do?

- | | |
|-------------------------------|--|
| 1) true | 11) b. center drill it |
| 2) b. milling machine | 12) d. spot drill |
| 3) true | 13) false |
| 4) true | 14) c. ream or bore it |
| 5) false | 15) b. tap |
| 6) a. spindle speed | 16) true |
| 7) b. feedrate | 17) true |
| 8) c. radial depth-of-cut | 18) false (they recommend feedrate in per-tooth fashion) |
| 9) b. surface feet per minute | 19) d. all of the above |
| 10) true | 20) true |