

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