

Table of Contents

Preface	11
Prerequisites	12
Basic machining practice experience	12
Controls covered.....	12
Limitations	13
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 Turning Centers	17
Lesson 1: Basic Machining Practices Required For CNC Turning 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.....	22
Behavior.....	22
Turning-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.....	24
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 2: What Does A CNC Turning Center Do?	63
Comparing a CNC machining center to other types of machines	63
Machining action of any lathe	66
The importance of basic machining practice to CNC people	67
The three general turning applications	67
Shaft work.....	67
Chucking work.....	69
Bar work.....	70
How a cutting tool works	71
External versus internal work.....	72
Roughing versus finishing operations	73
Roughing.....	73
How much finishing stock should be left for finishing?.....	74
Deterioration of roughing tools	75
When is finishing done?	75
Finishing.....	76
Some examples of rough and finish turning.....	77
Machining operations performed by single point cutting tools.....	80
Components of a single point cutting tool.....	82
The holder.....	82
Right hand versus left hand cutting tools	84
The insert.....	86

More about the geometry of a single point cutting tool.....	86
Lead angle, nose angle, clearance angle.....	87
Nose radius.....	87
Rake and relief angle.....	87
Other attributes of single point cutting tools.....	88
Chip breaker.....	88
Clamp and/or retaining pin.....	88
The difference between indexing and replacing an insert.....	88
Tool life.....	89
Chatter (vibration).....	89
Part finish.....	89
Seat.....	90
How carbide inserts are specified.....	90
First letter (shape):.....	90
Second letter (relief angle):.....	90
Third letter (tolerances):.....	90
Fourth letter (type of insert).....	91
First digit (size of insert).....	91
Second digit (thickness).....	91
Third digit (tool nose radius).....	92
Understanding wear on single point cutting tools.....	92
Introduction to cutting conditions.....	94
How cutting conditions are recommended.....	94
Spindle speed: Converting from surface feet per minute (sfm) to revolutions per minute (rpm).....	94
Spindle speed: Converting from meters per minute (mpm) to revolutions per minute (rpm).....	95
Feedrate: Converting inches per revolution (ipr) to inches per minute (ipm).....	95
Feedrate: Converting millimeters per revolution (mmpr) to millimeters per minute (mmpm).....	95
Getting recommendations for speed and feed.....	95
Other cutting conditions related to machining operations.....	95
Cutting tool materials.....	96
Getting recommendations for cutting conditions.....	97
The machining process.....	97
More on depth-of-cut.....	99
Roughing facing all external raw material.....	99
Finishing stock – on diameters and faces.....	100
Cutting conditions for rough facing.....	100
Cutting tools used for rough facing.....	101
Sizing adjustments needed during the life of the rough facing tool.....	102
Rough Turning.....	102
More about depth-of-cut.....	103
Finishing stock – on diameters and faces.....	103
Cutting conditions for rough turning.....	103
Cutting tools used for rough turning.....	104
A note about back turning.....	105
Sizing adjustments needed during the life of the rough turning tool.....	106
Drilling.....	106
Cutting conditions for drilling.....	107
Other types of drills.....	107
Spade drills.....	107
Carbide insert drills.....	108
Sizing adjustments needed during the life of a drill.....	108
Rough boring.....	109
More on depth-of-cut.....	109
About length-to-diameter ratio.....	109
Finishing stock – on diameters and faces.....	110
Cutting conditions for rough boring.....	110
Cutting tools used for rough boring.....	111
Sizing adjustments needed during the life of the rough boring bar.....	112
Where we stand in the process.....	112
Finish facing and turning.....	113
A reminder about the finishing stock left on faces.....	113

Understanding surface finish requirements	114
Cutting conditions for finish facing and turning.....	115
Cutting tools used for finish facing and turning.....	116
Sizing adjustments needed during the life of the finish facing and turning tool.....	116
Finish boring.....	117
A note about the finishing stock left on faces	118
Understanding surface finish requirements	118
Cutting conditions for finish facing and turning.....	118
Cutting tools used for finish boring.....	119
Sizing adjustments needed during the life of the finish boring bar.....	119
Other common machining operations	119
Necking (also called grooving.....)	119
Cutting conditions for necking.....	121
Parting (also called cut-off)	121
Threading.....	122
Cutting conditions for threading	123
Cutting tools for threading.....	123
Measuring threads.....	124
Machining internal threads.....	125
Sizing adjustments needed during the life of a threading tool.....	125
Workholding and support devices	125
Three jaw chucks	126
Hard jaws	127
Soft jaws	128
Programmable and/or manual features of three jaw chucks	129
Chuck jaws open and close.....	129
Chucking pressure.....	129
Chucking direction.....	130
Collet chucks	130
Bar feeding with collet chucks	131
Work support devices.....	132
Tailstocks	132
The tailstock body.....	133
Tailstock quill.....	133
Center.....	134
Tailstock alignment.....	134
Steady rests.....	134
Summary of key points	135
Conclusion to Key Concept Number One	135
Key Concept 2: Know Your Machine From An Operator's Viewpoint	139
The need for hands-on experience	139
Lesson 3: General Flow Of The CNC Process	141
Companies that use CNC turning centers.....	141
What will you be doing?	142
Flow of the CNC process	143
Study the workpiece drawing.....	143
Decision is made as to which CNC machine to use.....	143
The machining process is developed.....	143
Tooling is ordered and checked.....	143
The program is developed	143
Setup and production run documentation is made	143
Program is loaded into the CNC control's memory.....	143
The setup is made.....	144
The program is cautiously verified (still part of setup).....	144
Production is run.....	144
Corrected version of the program is stored for future use.....	144
Lesson 4: Machine Configurations	149
Types of CNC turning centers	149

Universal style slant bed turning center.....	149
Directions of motion (axes) for a universal style slant bed turning center.....	150
Live tooling for a universal style slant bed turning center.....	151
Other types of CNC turning centers.....	152
Chucking style slant bed turning center.....	153
Twin spindle horizontal bed turning centers.....	153
Sub-spindle style turning centers.....	154
Vertical single spindle turning centers.....	155
Twin spindle vertical turning centers.....	156
Gang style turning centers.....	156
Swiss-type CNC turning centers (also called sliding headstock turning centers).....	157
Programmable functions of turning centers.....	158
Spindle.....	158
Spindle speed.....	158
Spindle activation and direction.....	158
Spindle range.....	159
Feedrate.....	160
Turret indexing (tool changing).....	160
Turret station and offset selection.....	161
Coolant.....	162
Tailstock.....	162
What else might be programmable?.....	163
Key points for Lesson Four:.....	164
Lesson 5: Buttons And Switches On The Operation Panels.....	167
The two most important operation panels.....	167
The control panel buttons and switches.....	168
Display screen control keys (soft keys).....	168
Position display pages.....	169
Program display pages.....	170
Offset display pages.....	170
Program check display pages.....	170
The keyboard.....	171
Letter Keys.....	171
The slash key (/).....	171
Number keys.....	171
Decimal point key.....	171
The input key.....	172
Cursor control keys.....	172
Program Editing Keys.....	172
Reset key.....	172
Buttons and switches on the machine panel.....	173
Mode switch.....	173
Cycle start button.....	173
Feed hold button.....	174
Feedrate override switch.....	174
Rapid traverse override switch.....	174
Emergency stop button.....	175
Conditional switches.....	175
Dry run on/off switch.....	175
Single block on/off switch.....	175
Block delete on/off switch (also called optional block skip).....	175
Optional stop on/off switch.....	175
Buttons and switches for manual functions.....	176
Axis jogging controls.....	176
Handwheel controls.....	176
Spindle control.....	176
Turret index control.....	177
Indicator lights and meters.....	177
Spindle rpm and horsepower meters.....	177
Axis drive-motor horsepower meter.....	177

Cycle indicator lights	177
Zero return position indicator lights	177
Optional stop indicator light.....	177
Other buttons and switches on the machine panel.....	177
Other operation panels on your turning center	178
Lesson 6: The Key Operation Procedures	181
key Concept 3: Know The Compensation Types	189
Lesson 7: Introduction To Compensation	191
What is compensation and why is it needed?	191
The initial setting for compensation.....	192
When is trial machining required?.....	192
Understanding offsets.....	193
Offset organization.....	194
Offset pages on the display screen.....	194
How offsets are instated.....	196
Key points for Lesson Seven:	196
Lesson 8: Geometry Offsets And Assigning Program Zero	199
How is the program zero point determined?	199
Assigning program zero	201
Program zero must be assigned independently for <i>each</i> cutting tool.....	201
Understanding program zero assignment values.....	202
What is the zero return position?.....	202
Program zero assignment values	203
Understanding geometry offsets	205
The three most common ways to assign program zero.....	207
Using a tool touch-off probe to assign program zero – also called a <i>tool setter</i> (your 1st choice).....	207
Stylus use	208
What about center cutting tools?	209
Procedure to use a tool touch off probe	210
How does it work?.....	212
What about the wear offset for a cutting tool that has been probed?.....	212
What about tool pressure?.....	212
Determining and entering the work shift value.....	213
A note about the polarity of the work shift value	215
Using geometry offsets with work shift to assign program zero (your 2nd choice).....	216
Understanding the measure function.....	217
What if my machine doesn't have the measure function?.....	219
Using geometry offsets without work shift to assign program zero (your 3rd choice).....	220
Other kinds of cutting tools	222
Other considerations that affect program zero assignment values	224
How accurate are the program zero assignment values?.....	224
Trial machining.....	225
An advantage of tool touch off probes	225
Tool wear	225
Replacing dull tools during a production run.....	226
Using cutting tools from one job to the next	226
Key points for Lesson Eight:	227
Lesson 9: Wear Offsets	231
Which dimension do you choose for sizing?.....	232
How wear offsets are programmed.....	232
How wear offsets are entered.....	232
Sizing in a tool after it has just been placed in the turret	235
Sizing in a new tool with trial machining	236
What causes the initial deviation?.....	237
Dealing with deviations caused by tool wear	237

After a dull tool is replaced.....	238
What if my machine has a tool touch off probe?.....	239
Consistently replacing inserts	239
Consistently indexing inserts.....	240
Minimizing the need for trial machining.....	241
Going from job to job	241
When do you clear wear offsets?.....	242
A more complex example	242
Running the first workpiece – one tool at a time.....	243
Tool number one: Rough face and turn tool.....	243
Tool number Two: Finish face and turn tool.....	243
Tool number Three: 0.125 grooving tool.....	244
What about the Z position of the groove?.....	244
Completing the production run.....	245
Tool number one: Rough face and turn tool.....	245
Tool number Two: Finish face and turn tool.....	245
Tool number Three: 0.125 grooving tool.....	245
A reminder about up-coming jobs	245
A reminder about target values	246
Key points for Lesson Thirteen:	246
Lesson 10: Tool Nose Radius Compensation	249
Keeping the cutting edge flush with the work surface at all times	251
When tool nose radius compensation is used.....	252
Tool nose radius compensation from a setup person's point of view	252
What if my machine does not have geometry offsets?.....	254
What if I forget to enter tool nose radius compensation values?.....	254
What if I enter tool nose radius compensation values into wear offsets?.....	254
What if I enter tool nose radius compensation values into both the geometry and wear offsets?.....	254
Key points for Lesson Ten:	254
Conclusion to Key Concept Number Three:.....	254
Key Concept 4: Master Tasks To Run A CNC Turning Center	259
Lesson 11: Tasks Required To Setup A CNC Turning Center	261
A CNC job from start to finish.....	262
Setup documentation	263
Tear down the previous setup and put everything away.....	264
Gather the components needed to make the setup.....	264
Make the workholding setup.....	264
How To Mount Jaws In The Correct Serrations	265
The diameter at which to mount jaws	265
Using a long boring bar.....	265
Actually mounting jaws	266
A note about soft jaws	266
Machining soft jaws during setup.....	267
Other devices related to work holding setup.....	269
Assemble cutting tools	269
A reminder about mounting inserts.....	269
Load cutting tools into the turret and adjust coolant lines	270
Assign program zero for each new tool.....	270
Enter tool nose radius compensation values (if the programmer uses this feature).....	270
Load the CNC program/s	270
The physical tasks related to setup are now completed	271
Verify the correctness of a new or modified program	271
Verify the correctness of the setup.....	272
A tip that will save a crash some day	272
Dry running our example program	272
What if you do find a problem?.....	273

<i>Canceling the CNC cycle</i>	274
To cancel the cycle.....	274
When do you want the machine to stop?	276
Procedure to restart the program from the beginning of a specific cutting tool.....	276
Cautiously run the first workpiece.....	276
The most dangerous time.....	277
Making sure the first workpiece is a good one.....	277
Machining the first workpiece in our example job	278
Move through the program one tool at a time.....	280
Upcoming jobs	281
A note about our example job.....	281
First workpiece inspection.....	281
Program optimizing.....	281
Saving corrected version of the program.....	281
Verifying a job that contains mistakes	282
The dry run to check for setup mistakes.....	285
Cautiously running the first workpiece.....	286
Key points for Lesson Eleven:	287
Lesson 12: Tasks Required To Complete A Production Run	291
Production run documentation	292
Remove the previous workpiece.....	293
Load the next workpiece.....	294
Activate the cycle.....	294
Monitor the cycle.....	294
Clean and de-burr the just-completed workpiece.....	294
Perform specified measurements	296
Which takes longer, the CNC cycle or the off-line tasks an operator must perform?.....	297
Make offset adjustments to maintain size for critical dimensions (sizing).....	297
Replace worn tools	297
Clean the machine.....	298
Preventive maintenance.....	298
Machine warm-ups	298
Anything else you'll be expected to do?.....	298
Key points for Lesson Twelve:	298
Answers to Exercises	303
Exercise 1: Basic Machining Practices	303
Exercise 2: What Does A CNC Turning Center Do?	303
Exercise 3: General Flow Of The CNC Process.....	303
Exercise 4: Machine Configurations	304
Exercise 5: Understand The Control Panels	304
Exercise 6: The Key Operation Procedures	305
Exercise 7: Introduction To Compensation	305
Exercise 8: Fixture Offsets And Assigning Program Zero.....	306
Exercise 9: Tool Length Compensation.....	306
Exercise 10: Cutter Radius Compensation	307
Exercise 11: Tasks Required To Setup A CNC Turning Center.....	307
Exercise 12: Tasks Related To Completing A Production Run	307