Table of Contents

PREFACE 11
Prerequisites 11
Basic machining practice experience 12
Math 12
Motivation 12
Controls covered 12
Limitations 12
The need for hands-on practice 12
Instruction method 13
Scope 13
Key Concepts approach 13
Lesson structure 14
Practice makes perfect 14
Key Concepts and lessons 14
Enjoy! 14

GET READY TO LEARN ABOUT CNC TURNING CENTERS 15

Basic Machining Practices Required for CNC Turning Centers 17
Shop Safety 17
Safety equipment 17
Safety practices 18
Handling raw material 19
Finished workpiece handling 19
Tightening and loosening fasteners 19
Getting around the shop 20
Behavior 20
Turning-center-specific safety practices 20
Shop math 21
A few suggestions for choosing your calculator 21
Keep it simple 21
Watch out for trick functions 21
Avoid solar powered calculators 22
Big buttons – big display 22
Be sure it has clear entry button 22
Calculator basics 23
What is an arithmetic expression? 23
Practice makes perfect 24
The most basic arithmetic functions 24
Addition 24
Subtraction 25
Multiplication 25
Division 25
More on the priority of arithmetic operators 25
The use of parentheses in an expression 25
Does your calculator have parentheses? 26
Does your calculator have one or more memories? 26
What is a formula? 28
Who creates a formula? 29
More about numbers 30
Integers 30
Real numbers 30
Decimal format 30
Fractional format 30
Converting fractional format to decimal format 31
Decimal equivalents for common fractions 32
Decimal places 32
Rounding to a given number of decimal places 32
Measurement systems 33
Saying numeric values out loud in a machine shop 34
Saying values in the Metric system 35
Understanding polarity 35
Summary of shop math 35
Interpreting an engineering drawing 37
Orthographic projection 37
Line appearance 38
Non-standard views 39
Isometric views 39
Angular views 40
Detail views 40
Section views 41
Multiple page engineering drawings 42
Do you have a sample workpiece? 42
Drawing scale 42
Dimensioning 43
Measurement systems 44
Other descriptive notes 44
Title block 44
Revisions 44
Introduction to tolerance interpretation 46
Who specifies tolerance? 47
Dimensional tolerances 48
Tolerance terminology 49
Judging acceptability 50
What are you shooting for? 50
Calculating the deviation (adjustment) amount 50
Determining the deviation (sizing adjustment) polarity 50
More on dimensional tolerance specifications 50
High/low limits 51
Uneven +/- 51
You may get some help 52
Implied tolerances 52
Four conditions of a measured surface 52
If the surface is not within low and high limits, can the workpiece be saved? 52
Tool wear affects acceptability 53

Cutting tool type affects how quickly tool wear affects machined surfaces 53
Dealing with deviations..... 54
Choosing the surface to measure 54
More on determining deviation amount ... 55
Setup sizing versus production run sizing..... 55
During setup..... 55
During production runs..... 56
Why targeting the mean value may not be best 56
Geometric (surface relationship) tolerances 57
Measuring devices 60
Checking calibration..... 60
Measurement systems affect gauges 60
Gauge resolution 61
Common gauge resolutions in each measurement system 61
Environments for measuring 61
Minimizing the impact temperature changes 63
Measurement types..... 63
Gauge types 65
Fixed gauges..... 66
Go/No-go fixed gauges 66
Fixed gauges with dial displays 67
Importance of the secondary dial..... 71
More on needle rotation direction and internal versus external measurements..... 71
Variable gauges..... 74
Display types..... 76
Electronic digital display 76
Dial display..... 77
Dial indicators 79
Vernier display 81
Display type demonstrations 82
Accuracy of the measuring device 82
Checking calibration..... 82
A general suggestion for getting started with any variable gauge 84
Interpreting (reading) the displayed value of a Vernier micrometer 85
General use of a micrometer that has a clamp and ratchet..... 86
Steps to reading the displayed value for a micrometer that measures in inches 86
Steps to reading the displayed value for a micrometer that measures in millimeters . 88
Interpreting (reading) the displayed value of a dial caliper..... 91
Steps to reading the displayed value for a micrometer that measures in inches 91
Steps to reading the displayed value for a micrometer that measures in millimeters . 92
Interpreting (reading) the displayed value of a Vernier caliper..... 93
Using a depth gauge 95
Testing for calibration..... 96
Summary 96

What Does A CNC Turning Center Do? 99
Comparing a CNC turning center to other types of machines 99
Machining action of any lathe 101
The importance of basic machining practice to CNC people 102
The three general turning applications 103
Shaft work..... 103
Chucking work..... 104
Bar work..... 105
How a cutting tool works 106
External versus internal work 107
Roughing versus finishing operations 108
Roughing 108
How much finishing stock should be left for finishing?..... 109
Deterioration of roughing tools 109
When is finishing done?..... 110
Finishing 110
Some examples of rough and finish turning 111
Machining operations performed by single point cutting tools 114
Components of single point cutting tools..... 116
The holder..... 116
Right hand versus left hand cutting tools 118
The insert 120
More about the geometry of a single point cutting tool..... 121
Insert shape..... 121
Lead angle, nose angle, clearance angle..... 121
Nose radius 122
Rake and relief angle..... 122
Other attributes of single point cutting tools .. 122
Chip breaker..... 123
Clamp screw/pin, clamp, and/or retaining pin 123
The difference between indexing and replacing an insert 123
Tool life 124
Chatter (vibration) 124
Part finish 124
Shim 124
How carbide inserts are specified..... 124
First letter (shape):..... 125
Second letter (relief angle): 125
Third letter (tolerances):..... 125
Fourth letter (type of insert)..... 126
First digit (size of insert)..... 126
Second digit (thickness) 126
Third digit (tool nose radius)..... 126
Understanding wear on single point cutting tools 126
Introduction to cutting conditions 128

 © CNC Concepts, Inc.

How cutting conditions are recommended.....128
Spindle speed: Converting from surface feet per minute (sfm) to revolutions per minute (rpm).....129
Spindle speed: Converting from meters per minute (mpm) to revolutions per minute (rpm).....129
Feedrate: Converting inches per revolution (ipr) to inches per minute (ipm).....129
Feedrate: Converting millimeters per revolution (mmpr) to millimeters per minute (mmpm).....129
Getting recommendations for speed and feed.....129
Other cutting conditions related to machining operations.....130
Cutting tool materials.....131
Getting recommendations for cutting conditions.....131
The machining process 131
Rough facing.....132
More on depth-of-cut.....133
Roughing facing all external raw material.....133
Finishing stock – on diameters and faces.....134
Cutting conditions for rough facing.....134
Cutting tools used for rough facing.....135
Sizing adjustments needed during the life of the rough facing tool.....136
Rough Turning.....136
More about depth-of-cut.....137
Finishing stock – on diameters and faces.....137
Cutting conditions for rough turning.....137
Cutting tools used for rough turning.....138
A note about back turning.....139
Sizing adjustments needed during the life of the rough turning tool.....140
Drilling.....140
Cutting conditions for drilling.....141
Other types of drills.....141
Spade drills.....141
Carbide insert drills.....142
Sizing adjustments needed during the life of a drill.....143
Rough boring.....143
More on depth-of-cut.....143
About length-to-diameter ratio.....143
Finishing stock – on diameters and faces.....144
Cutting conditions for rough boring.....144
Cutting tools used for rough boring.....145
Sizing adjustments needed during the life of the rough boring bar.....146
Where we stand in the process.....146
Finish facing and turning.....147
A reminder about the finishing stock left on faces.....147
Understanding surface finish requirements.148
Cutting conditions for finish facing and turning.....149
Cutting tools used for finish facing and turning.....150
Sizing adjustments needed during the life of the finish facing and turning tool.....150
Finish boring.....151
About the finishing stock left on faces.....152
Understanding surface finish requirements.152
Cutting conditions for finish facing and turning.....152
Cutting tools used for finish boring.....153
Sizing adjustments needed during the life of the finish boring bar.....153
Other common lathe machining operations 153
Necking (also called grooving.....153
Cutting conditions for necking.....154
Parting (also called cut-off).....155
Threading.....155
Cutting conditions for threading.....156
Cutting tools for threading.....157
Measuring threads.....157
Machining internal threads.....158
Sizing adjustments needed during the life of a threading tool.....158
Workholding and support devices 158
Three jaw chucks.....159
Top tooling.....160
Hard jaws.....160
Soft jaws.....161
Programmable and/or manual features of three jaw chucks.....162
Chuck jaws open and close.....162
Chucking pressure.....162
Chucking direction.....163
Collet chucks.....163
Bar feeding with collet chucks.....164
Work support devices.....165
Tailstocks.....165
The tailstock body.....166
Spindle considerations.....166
Tailstock quill.....166
Center.....166
Tailstock alignment.....167
Steady rests.....167
Summary of key points.....168
Conclusion to Key Concept Number One 168

KNOW YOUR MACHINE FROM AN OPERATOR'S VIEWPOINT 171
The need for hands-on experience 171

General Flow of the CNC Process 173
Companies that use CNC turning centers 173
What will you be doing?.....174

Flow of the CNC process 174
Study the workpiece engineering drawing 175
Decision is made as to which CNC machine to use 175
The machining process is developed 175
Tooling is ordered and checked 175
The program is developed 175
Setup and production run documentation is made 175
Program is loaded into the CNC control's memory 175
The setup is made 176
The program is cautiously verified (still considered part of setup) 176
Production is run 176
Corrected version of the program is stored for future use 176

Machine Configurations 179
Types of CNC turning centers 179
Universal style slant bed turning center 179
Directions of motion (axes) for a universal style slant bed turning center 180
X is specified in diameter 181
Live tooling for a universal style slant bed turning center 181
Other types of CNC turning centers 182
Chucking style slant bed turning center 183
Twin spindle horizontal bed turning centers 183
Sub-spindle style turning centers 184
Vertical single spindle turning centers 185
Twin spindle vertical turning centers 186
Gang style turning centers 186
Swiss-type CNC turning centers (also called sliding headstock turning centers) 187
Programmable functions of turning centers 188
Spindle 188
Spindle speed 188
Spindle activation and direction 188
Spindle range 189
Feedrate 189
Turret indexing (tool changing) 190
Turret station and offset selection 191
Coolant 192
Other possible programmable functions 192
Tailstock 192
Programmable steady rest 192
Bar feeders and chuck activation 192
Part catcher 192
Tool touch off probe 193
Key points for Lesson Four: 193

Buttons And Switches on the Operation Panels 197
The Most Important Operation Panels 197
Control Panel Functions 198
Display Screen Function Keys 199
Program Display Pages 200
Offset Display Pages 201
Graph Display Pages 202
Other Display Screen Modes 202
The Keyboard 202
Letter Keys 203
Slash Key (/) 203
Number Keys 203
Decimal Point Key 203
CAN Key 203
EOB Key 203
Input Key 203
Cursor Control Keys 203
Program Editing Keys 203
Reset Key 203
The Machine Panel 204
Power Buttons 204
MODE Switch 204
CYCLE START Button 204
FEED HOLD Button 205
FEEDRATE OVERRIDE Switch 205
RAPID OVERRIDE Switch 205
EMERGENCY STOP Button 205
Conditional Switches 206
DRY RUN On/Off Switch 206
SINGLE BLOCK On/Off Switch 206
BLOCK DELETE On/Off Switch (also called optional block skip) 206
OPTIONAL STOP On/Off Switch 206
Buttons and Switches for Manual Functions 206
Axis Jogging Controls (continuous jog) 206
Axis Jogging Controls (incremental jog) ... 207
Hand wheel Controls 207
Spindle Control 207
Turret Index Control 207
Indicator Lights and Meters 207
Spindle RPM and Horsepower Meters 207
Axis Drive-Motor Horsepower Meter 208
Cycle Indicator Lights 208
Reference Position Indicator Lights 208
Optional Stop Indicator Light 208
Other Buttons and Switches on the Machine Panel 208
Other Operation Panels on your Turning Center 208
More About the Mode Switch 208
The Manual Mode Switch Positions 208
MDI Mode Switch Position 209
Commanding an MDI Reference Return 209
The Complete Procedure to Execute an MDI Command 210
Commanding an MDI Turret Index 210
Commanding Spindle Activation with MDI .. 210
Other Times when MDI is used 211
Commanding Motion with MDI 211
EDIT Mode Switch Position 211

 © CNC Concepts, Inc.

To Make a Program in Memory the Active Program (to call up a program) 212
To Enter a New Program 212
What if I Make a Mistake when Typing? 212
Program Operation Mode 212
To Run the Active Program from the Beginning 213

Important Operation Procedures 217
Develop Your Own Operation Handbooks 217
Manual Procedures: 217
MDI Procedures 217
Program Manipulation Procedures 217
Setup Procedures 218
Program Running Procedures 218
Blank Procedure Form 218
Sample Operation Handbook (for a machine with a FANUC 0iF control) 219
Manual Procedures 219
To Power-Up the Machine 219
To Do a Reference Return (send each axis to its reference position) 219
To Index the Turret 219
To Start the Spindle 219
To Jog the Axes (using continuous jog) 219
To Jog the Axes (using incremental jog) 219
To Use the Hand Wheel 220
To Set Axis Displays 220
Origin 220
Preset 220
To Enter Wear Offsets 220
+INPUT (Modifies current offset value) ... 220
INPUT (Overwrites current offset value) .. 220
MDI Procedures 220
To Execute an MDI Command 220
Program Manipulation Procedures 221
To Get Ready to Edit Programs 221
To Show a Directory of Programs 221
To Call up a Program from within the CNC Memory (make it the active program) 221
Method 1: (from the program page, assumes you know the program number) 221
Method 2: (from the directory/folder page) 221
To Load a Program 221
To Delete a Program 222
To Search within a Program 222
Search with Cursor Keys 222
Find the Next Word Occurrence Using Only the Letter Address 222
Find the Next Word Occurrence Using the Whole Word 222
To Alter, Insert, & Delete 222
Alter 222
Insert 222
Delete 222
To Save Programs to an External Device 223
To Use Background Edit 223
Setup Procedures 223
To Mount Jaws 223
To Measure and Enter Program zero Assignment Values 223
For the X-Axis Geometry Offset 223
For the Z-Axis Geometry Offset 224
To Measure and Enter the Work Shift Value 224
Program Running Procedures 224
To Run the Program (in normal production - no verification techniques) 224
To Rerun Tools 225
To Do a Free-Flowing Dry Run 225
To Do a Normal Air Cutting Run 225
To Run the First Workpiece 225
To Cancel the Cycle 226
To Clear an Alarm 226

KNOW THE COMPENSATION TYPES 229

Introduction To Compensation 231
What is compensation and why is it needed? 231
Tolerance review 232
What if a Measured Attribute is not On-Size (not acceptable)? 232
The Target Value 232
One More Consideration – Tool Wear 233
Do You Really Want to Target the Mean Value? 233
The Initial Setting for Compensation 234
When is Trial Machining Required? 234
What happens as tools begin to wear? 234
Never make sizing adjustments by changing the CNC program! 235
Understanding Offsets 235
Offset Organization 236
Offset Pages on the Display Screen 236
How Offsets are Instated 238
Key points for Lesson Seven: 238

Geometry Offsets and Assigning Program Zero 241
How is the program zero point determined? 241
In X 241
In Z 241
Determining the Geometry Offset Values 243
Understanding Program zero Assignment Values 244
What is the Reference Position? 244
Program zero Assignment Values 245
The Relative Position Display Page 246
Other Kinds of Cutting Tools 247
Other Considerations that Affect Program Zero Assignment Values 248

How Accurate are the Program Zero Assignment Values? 249
Understanding Geometry Offsets 249
How Geometry Offsets are Instated 251
Three Common Ways to Assign Program zero 251
Using a Tool Touch-Off Probe (1st choice) 252
Stylus Use 253
How it Works 256
What About the Wear Offset for a Cutting Tool that has been Probed? 256
A Note about Tool Pressure 256
Determining and Entering the Work Shift Value 257
A Note about the Polarity of the Work Shift Value 259
Using Geometry Offsets with Work Shift to Assign Program Zero (2nd choice) 259
The Measure Function 260
Using Geometry Offsets Without Work Shift to Assign Program zero (3rd choice) 262
What if a Machine Does Not Have the Measure Function? 263

Wear Offsets 267
Review of the T-Word 268
About Wear Offset Cancellation 268
How to Enter Wear Offsets 268
Which is Better, [INPUT] or [+INPUT]? 270
What if my Machine Does Not Have a [+INPUT] Soft Key? 270
Sizing in a Tool After it has Just Been Placed in the Turret 271
Sizing in a New Cutting Tool with Trial Machining 272
Causes for the Initial Deviation 272
Dealing with Deviations Caused by Tool Wear 273
After a Dull tool is Replaced 274
If the Machine has a Tool Touch-Off Probe 274
Consistently Replacing Inserts 275
Consistently Indexing Inserts 275
Minimizing the Need for Trial Machining 276
Going from Job to Job 276
When do you Clear Wear Offsets? 277
A More Complex Example 277
Running the First Workpiece – One Tool at a Time 278
Tool #1: Rough Face and Turn Tool 278
Tool #2: Finish Face and Turn Tool 278
Tool #3: 3.0-mm (0.125-in) Grooving Tool... 279
What About the Z Position of the Groove? 279
Completing the Production Run 280
Tool #1: Rough Face and Turn Tool 280
Tool #2: Finish Face and Turn Tool 280
Tool #3: 3.0-mm (or 0.125-in) Wide Grooving Tool 280
A Reminder About Up-Coming Jobs 280
A Reminder About Target Values 281

Tool Nose Radius Compensation 285
Why Tool Nose Radius Compensation is Required 285
+The Size of the Deviation 287
Keeping the Cutting-Edge Flush with the Work Surface 287
When to use Tool Nose Radius Compensation 288
Tool Nose Radius Compensation from a Setup Person's Viewpoint 288
If You Forget to Enter Tool Nose Radius Compensation Values 290
Conclusion to Key Concept Number Three: 290

MASTER TASKS TO RUN A CNC TURNING CENTER 293

Tasks Required to Setup a CNC Turning center..... 295
A CNC job from start to finish 295
Setup documentation 297
Tear down the previous setup and put everything away 298
Gather the components needed to make the setup 298
Make the workholding setup 298
How to mount jaws in the correct serrations 298
The diameter at which to mount jaws 299
Using a long boring bar 299
Actually mounting jaws 299
A note about soft jaws 300
Machining soft jaws during setup 300
Other devices related to work holding setup 302
Assemble cutting tools 302
A reminder about mounting inserts 303
Load cutting tools into the turret and adjust coolant nozzles 303
Assign program zero for each new tool 303
Enter tool nose radius compensation values (if the programmer uses this feature) 303
Load the CNC program/s 304
The physical tasks related to setup are now completed 304
Verify the correctness of a new or modified program 304
Verify the correctness of the setup 305
A tip that will save a crash some day 305
Dry running our example program 305

What if you do find a problem? 306
Canceling the CNC cycle 306
To cancel the cycle 307
Rerunning a tool in the program 307
When do you want the machine to stop? 308
Procedure to restart the program from the beginning of a specific cutting tool 308
Cautiously run the first workpiece 309
The most dangerous time 309
Making sure the first workpiece is a good one 310
Machining the first workpiece in our example job 310
Tool one: 310
Tool two 311
Tool three 311
Tool four 311
Tool five 312
Move through the program one tool at a time 312
Upcoming jobs 313
A note about our example job 313
First workpiece inspection 313
Program optimizing 313
Saving corrected version of the program 313
Verifying a Job that Contains Mistakes 314
Dry Run to Check for Setup Mistakes 318
Cautiously Running the First Workpiece 319

Tasks Required to Complete a Production Run 323
Production run documentation 324
A note to programmers: 324
Remove the previous workpiece 325
Load the next workpiece 326
Activate the cycle 326
Monitor the cycle 326
Clean and de-burr the just-completed workpiece 326
Perform specified measurements 328
Which takes longer, the CNC cycle or the off-line tasks an operator must perform? 328
Make offset adjustments to maintain size for critical dimensions (sizing) 328
Replace worn tools 329
Clean the machine 329
Preventive maintenance 329
Machine warm-ups 330
Anything else you'll be expected to do? 330
Key points for Lesson Twelve: 330

Answers to Exercises
Exercise 1: Basic Machining Practices 333
Exercise 2: What Does a CNC Turning center Do? 333
Exercise 3: General Flow of the CNC Process 333
Exercise 4: Machine Configurations 334
Exercise 5: Buttons and Switches on the Control Panels 334
Exercise 6: The Key Operation Procedures 335
Exercise 7: Introduction to Compensation 335
Exercise 8: Workpiece coordinate system offsets and Assigning Program Zero 335
Exercise 9: Tool Length Compensation 336
Exercise 10: Cutter Radius Compensation 337
Exercise 11: Tasks Required to Setup a CNC Turning center 337
Exercise 12: Tasks Related to Completing a Production Run 338

Index: 341