Table of Contents

Preface 9

Prerequisites 9

Basic machining practice experience 9

Controls covered 10

Limitations 10

Programming method 10

The need for hands-on practice 10

Instruction method 11

Scope 11

Key Concepts approach 11

Lesson structure 12

Practice makes perfect 12

Key Concepts and lessons 12

Enjoy! 12

Key Concept 1: Know Your Machine From A Programmer's Viewpoint 13

Lesson 1: Machine Configurations 15

Types of CNC turning centers 15

Universal style slant bed turning center 15

Directions of motion (axes) for a universal style slant bed turning center 16

Live tooling for a universal style slant bed turning center 17

Other types of CNC turning centers 19

Chucking style slant bed turning center 19

Twin spindle horizontal bed turning centers 20

Sub-spindle style turning centers 21

Vertical single spindle turning centers 22

Twin spindle vertical turning centers 23

Gang style turning centers 23

Swiss-type CNC turning centers (also called sliding headstock turning centers) 24

Programmable functions of turning centers 24

Spindle 24

Spindle speed 24

Spindle activation and direction 25

Spindle range 25

Feedrate 26

Turret indexing (tool changing) 27

Turret station and offset selection 28

Coolant 28

Other possible programmable functions 28

Tailstock 28

Programmable steady rest 29

Bar feeders and chuck activation 29

Part catcher 29

Tool touch off probe 29

Automatic tool changing systems 30

Exceptions to X axis 30

A quick fix 30

Gang style turning centers with cutting tools on both sides of the spindle centerline 30

Center cutting axis 31

What else might be programmable? 31

Key points for Lesson One: 31

Lesson 2: Understanding Turning Center Speeds and Feeds 33

The machining operation to be performed 33

The material to be machined 33

The material of the cutting tool's cutting edge 33

The two ways to select spindle speed 34

When to use constant surface speed mode 35

When to use rpm mode 36

How fast will the spindle be running when constant surface speed is used? 37

How fast can the spindle rotate? 37

How to specify a maximum speed for the constant surface speed mode 38

A potential limitation of constant surface speed 38

The two ways to specify feedrate 39

When to use the feed per revolution mode 39

When to use the feed per minute feedrate mode 40

An example of speed and feed usage 40

Key points for Lesson Two: 41

Lesson 3: Visualizing The Execution Of A CNC Program 43

Program make-up 44

Method of program execution 44

An example of program execution 44

The CNC program to machine the 2.875 diameter 45

Sequence numbers 46

A note about decimal point programming 46

A decimal point tip 47

Other mistakes of omission 47

Modal words 47

Initialized words 47

Letter O or number zero? 47

Word order in a command 48

Key points for Lesson Three: 48

Lesson 4: Program Zero And The Rectangular Coordinate System 49

Graph analogy 49

More about polarity 50

Wisely choosing the program zero point location 51

In X 51

In Z 51

Absolute versus incremental positioning movements 56

Another way to specify absolute and incremental positioning 56

A decimal point reminder 57

Key points for Lesson Four: 58

Lesson 5: Introduction To Programming Words 61

Words allowing a decimal point 61

O 62

N 62

G 62

X 62
Y 62
Z 62
U 63
W 63
C 63
R 63
I & K 63
F 63
E 63
S 64
T 64
M 64
P 64
Q 64
L 64
EOB (end of block character) 65
/ (slash code) 65
G and M codes 65
G codes 65
G code limitation: 65
Option G codes 65
What does initialized mean? 66
What does modal mean? 66
The most popular G codes 66
Common M codes used on a CNC turning center 67
Key points for Lesson Five: 68
Key points: Key Concept one, Know your machine from a programmer's viewpoint 69

Key Concept 2: You Must Prepare To Write Programs 71

Preparation and time 71
Preparation and safety 72
Typical mistakes 73
Syntax mistakes 73
Motion mistakes 73
Mistakes of omission 73
Process mistakes 74

Lesson 6: Preparation Steps For Programming 75

Prepare the machining process 75
Develop the needed cutting conditions 77
An example 78
Roughing tools 78
Drilling 79
Finishing tools 79
Chasing threads 79
Cutting conditions can be subjective 79
Do the required math and mark-up the print 80
Other ways to come up with coordinates 83
Marking up the print 83
Doing the math 84
Check the required tooling 86
Plan the work holding set-up 87
Other documentation needed for the job 88
Production run documentation 89
Program listing 89
Is it all worth it? 89
Key points for Lesson Six: 89

Key Concept 3: Understand The Motion Types 93

What is interpolation? 93

Lesson 7: Programming The Three Most Basic Motion Types 97

Motion commonalties 97
Understanding the programmed point of each cutting tool 97
G00 Rapid motion (also called positioning) 101
What is a safe approach distance? 102
What about feed-off distance? 103
G01 linear interpolation (straight line motion) 104
Using G01 for a fast feed approach 106
G02 and G03 Circular motion commands 107
Specifying a circular motion with the radius word 107
Circular motion with directional vectors (I and K) 109
What's wrong with this picture? 110
Key points for Lesson Six: 113

Key Concept 4: Know The Compensation Types 115

More on interpreting tolerances 115
What if a measured dimension is not on-size (not acceptable)? 116
The target value 117
Another consideration – tool wear 117
Do you really want to target the mean value? 118

Lesson 8: Introduction To Compensation 119

What is compensation and why is it needed? 119
The initial setting for compensation 120
When is trial machining required? 120
What happens as tools begin to wear? 121
What do you shoot for? 121
Why do programmers have to know this? 121
Understanding offsets 121
Offset organization 122
Offset pages on the display screen 122
Key points for Lesson Eight: 124

Lesson 9: Geomety Offsets And Wear Offsets 125

Review of reasons for using geometry offsets to assign program zero 125
How geometry offsets work 125
The total program zero assignment value 126
Warning about the machine lock feature: 127
Minimizing program zero assignment effort from job to job 127
So when do you clear geometry offsets? 129
Wear offsets 129
Which dimension do you choose for sizing? 130
How wear offsets are programmed 130
What about wear offset cancellation? 131
Secondary wear offset applications 131
Flip jobs 132

 © CNC Concepts, Inc

Two or more critical diameters 132
Unwanted taper 132
Grooving into different areas of the workpiece 132
Key points for Lesson Nine: 132

Lesson 10: Tool Nose Radius Compensation 137

Keeping the cutting edge flush with the work surface at all times 139
When to use tool nose radius compensation 140
Steps to programming tool nose radius compensation 140
Instating tool nose radius compensation 140
Programming motion commands to machine the workpiece 141
Canceling tool nose radius compensation 142
An example program 142
Tool nose radius compensation from a setup person's point of view 143
What if my machine does not have geometry offsets? 145
What if I forget to enter tool nose radius compensation values? 145
What if I enter tool nose radius compensation values into wear offsets? 145
What if I enter tool nose radius compensation values into both the geometry and wear offsets? 145
Programming tool nose radius compensation value entries 145
Another example program showing tool nose radius compensation 145
Key points for Lesson Ten: 147

Key Concept 5: You Must Provide Structure To Your CNC Programs 149

Lesson 11: Introduction To Program Structure 151

Objectives of your program structure 151
Reasons for structuring programs with a strict and consistent format 151
Familiarization 151
Consistency 152
Re-running tools in the program 152
Efficiency limitations 154
Machine variations that affect program structure 155
M code differences 155
G code numbering differences 156
Turret variations 157
How do you determine a safe yet efficient index position? 157
What if my machine doesn't have geometry offsets? 158
A few spindle concerns 159
Spindle limiting 159
Choosing the appropriate s pindle range 160
Which direction do you run the spindle? 160
How do you check what each tool has done? 160
Safety commands 160
How to use our given formats 161
Key points for Lesson Eleven: 162

Lesson 12: Four Types Of Program Format 163

Format for assigning program zero with geometry offsets 164
Format for assigning program zero in the program with G50 165
A reminder about the program's starting point and tool change position 165
Program startup format (assigning program zero in the program) 165
Tool ending format (assigning program zero in the program) 166
Tool startup format (assigning program zero in the program) 167
Program ending format (assigning program zero in the program) 167
Understanding the formats 167
G Words 167
M Words 168
Other M Words Related to Turning Centers 168
Other Words In The Format 169
Example programs showing format for turning centers 169
Where are the restart commands? 171
Example program when assigning program zero in the program 171
Where are the restart commands? 172
Suggestions for cycle time improvements. 173
Combine M codes in motion commands 173
Minimize spindle dead time 173
Efficiently programming spindle range changes 176
Minimize spindle reversals 177
Key points for Lesson Twelve: 178

Key Concept 6: Special Features That Help With Programming 181

Control series differences 182

Lesson 13: One-Pass Canned Cycles 183

Cycle consistencies 183
G90 - One pass turning and boring cycle 184
G92 - One pass threading cycle 185
G94 - One pass facing command 186
Example of G90 and G94 186
Example of G92 command 187
Key points for Lesson Thirteen: 188

Lesson 14: G71 And G70 – Rough Turning And Boring Followed By Finishing 189

G71 - Rough turning and boring 189
The two phases of G71 190
Understanding G71 command words 192
P word 192
Q word 192
U word 192
W word 192
D word 192
F word 193

What about finishing? 193
Example showing G71 for rough turning and G70 for finish turning 193
Using G71 for rough boring 195
Limitations of the G71 command 197
Different format for some controls (with two-line multiple repetitive cycle) 198
Key points for Lesson Fourteen: 198

Lesson 15: G72-G75 – Other Multiple Repetitive Cycles 201
G72 - Rough facing 201
Different format for two-command multiple repetitive cylces 203
G73 - pattern repeating 203
D word 204
I word 204
K word 204
Example of G73 pattern repeating 205
Can you use G73 for castings and forgings? 206
Different format for two-command multiple repetitive cycles 207
G74 - peck drilling 207
What if I must clear chips between pecks? 208
Different format for two-command multiple repetitive cycles 209
G75 - grooving cycle 209
Key points for Lesson Fifteen: 210

Lesson 16: G76 Threading Command 211
X word 211
Z word 212
What is thread chamfering? 212
K word 212
D word 212
A word 213
F word and E word 213
I word 213
Q word 214
Example program for threading 214
Other tips on threading 215
When possible, thread in the lowest spindle range 215
Thread in the rpm mode (G97) 215
Watch out for maximum allowable feedrate 215
Thread with thread chamfering turned off 215
Finish the thread before removing workpiece 216
Right hand threads versus left hand threads 216
Offsetting for threading tools 216
Start the tool far enough away from the thread being machined 216
Minimum depth-of-cut, final pass depth, and number of spring passes 216
Disabled or modified control functions during threading 216
Tapered threads 217
Multiple start threads 218
Different format for two-command multiple repetitive cycles 220
What about tapping? 220
Tap in the rpm mode 220
Tap in the low range 220
Use G32 as the motion command for tapping 221
Use a tension/compression tap holder 221
Keep the approach position 0.2 inch from the Z surface to tap 221
Key points for Lesson Sixteen: 222

Lesson 17: G76 Working With Subprograms 223
The difference between main- and sub- programs 223
Loading multiple programs 223
Words used with subprograms 224
A quick example 224
Nesting subprograms 226
Applications for subprograms 226
Repeated machining operations 226
Control programs 226
Utility applications 227
Example for repeating machining operations - multiple identical grooves 227
Example for control program applications - flip jobs 229
Example for utility applications - bar feeder activation 230
Special notes about M99 232
Ending a main program with M99 232
Changing the order of program execution with M99 232
What is parametric programming (custom macro B)? 235
Part families 235
User defined canned cycles 236
Utilities 236
Complex motions and shapes 237
Key points for Lesson Seventeen: 238

Lesson 18: Other Special Programming Features 239
Block delete (also called optional block skip) 239
Applications for block delete 240
Another optional stop 240
Trial machining 240
Warning about block delete applications 242
Sequence number (N word) techniques) 242
Eliminating sequence numbers 242
Using special sequence numbers in program restart commands 242
Documenting your programs with messages in parentheses 244
General information about the job 244
Tool information 245
At every program stop 245
To document anything out of the ordinary 245
For changes made after a dispute 246
Automatic corner rounding and chamfering 246
Other G codes of interest 248
G04 - Dwell command 248

 © CNC Concepts, Inc

Other G codes 249
Key points for Lesson Eighteen: 250

Lesson 19: Special Machine Types And Accessories 251

Work holding and work support devices 251
Work holding devices 251
Three jaw chucks 252
Programmable features of three jaw chucks 254
Collet chucks 255
Work support devices 257
Tailstocks 257
The tailstock body 257
Tailstock quill 258
Tailstock center 258
Tailstock alignment problems 258
Programming considerations 258
Steady rests 259
Bar feeders 260
How a bar feeder works 260
Workholding considerations 260
Styles of bar feeders 260
How to program for bar feeders 261
Determining how much to feed the bar 261
The steps to bar feeding 262
The redundancy of bar feed programming 264
When to program the bar feed 264
Ending a bar feed program 264
An example bar feeding program 265
Part catchers 267
Live tooling 267
Features of live tooling turning centers (also called mill/turn machines) 267
Rotating tools 267
Special tool holders 267
Precise control of main spindle rotation 268
Only one way to specify speed and feedrate 268
Hole machining canned cycles 268
Polar coordinate interpolation 268
Selecting the main spindle mode 269
Programming an indexer 269
Example program for an indexer 270
Programming a rotary axis (C axis) 272
Angular values 272
Zero return position 272
Rapid versus straight line motion 272
Program zero assignment 272
Absolute versus incremental 273
Canned cycles for hole machining 274
How do you specify the machining direction? 274
Canned cycle types 274
Words used in canned cycles 275
Understanding polar coordinate interpolation 276
Other machine types 280
Twin spindle turning centers 281
Sub-spindle turning centers 281
Swiss-type turning centers (also called sliding headstock turning centers) 281

Practice exercises 283

Programming Activities 319

Answers to Exercises 367

Answers to programming activities 376