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

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
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

Lesson 7: Programming The Three Most Basic Motion Types 97
Motion commonalities 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
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 spindle range 160
Which direction do you run the spindle? 160
How do you check what each tool has done? 160
Safety commands 160
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

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

Key points for Lesson Fourteen: 198

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

Key points for Lesson Fifteen: 210

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 Sixteen: 222

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 |
Lesson 19: Special Machine Types And Accessories

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