Table of contents

Preface 11

Prerequisites 11
Basic machining practice experience 11
Math 12
Motivation 12
Controls covered 12
What about conversational controls? 13
Controls other than Fanuc 13
Limitations 14
Instruction method 14
Scope 14
Key Concepts approach 15
Lesson structure 15
Practice, practice, practice! 15
Key Concepts and lessons 15
1: Know your machine from a programmer's viewpoint 15
2: You must prepare to write programs 15
3: Understand the motion types 16
4: Know the compensation types 16
5: You must provide structure to your CNC programs 16
6: Special features that help with programming 16
7: Know your machine from a setup person or operator's viewpoint 16
8: Know the three basic modes of operation 16
9: Understand the importance of procedures 16
10: You must know how to safely verify programs 16
Enjoy! 16

Key Concept 1: Know Your Machine From A Programmer's Viewpoint 17

Lesson 1: Machine Configurations 19
Types of CNC turning centers 19
Universal style slant bed turning center 19
Directions of motion (axes) for a universal style slant bed turning center 20
Live tooling for a universal style slant bed turning center 21
Other types of CNC turning centers 23
Chucking style slant bed turning center 23
Twin spindle horizontal bed turning centers 23
Sub-spindle style turning centers 24
Vertical single spindle turning centers 25
Twin spindle vertical turning centers 26
Gang style turning centers 26
Swiss-type CNC turning centers (also called sliding headstock turning centers) 27
Programmable functions of turning centers 28
Spindle 28
Spindle speed 28
Spindle activation and direction 28
Spindle range 29
Feedrate 30
Turret indexing (tool changing) 30
Turret station and offset selection 31
Coolant 32
Other possible programmable functions 32
Tailstock 32
Programmable steady rest 32
Bar feeders and chuck activation 32
Part catcher 33
Tool touch off probe 33
Automatic tool changing systems 33
Exceptions to X axis 33
A quick fix 33
Gang type turning centers with cutting tools on both sides of the spindle centerline 34
Center cutting axis 34
What else might be programmable? 34
Key points for Lesson One: 35

Lesson 2: Understanding Turning Center Speeds and Feeds 36
The two ways to select spindle speed 37
When to use constant surface speed mode 38
When to use rpm mode 39
How to specify a maximum speed for the constant surface speed mode 41
A potential limitation of constant surface speed 41
The two ways to specify feedrate 42
When to use the feed per revolution mode 42
When to use the feed per minute feedrate mode 43
An example of speed and feed usage 43
Key points for Lesson Two: 44

Lesson 3: General Flow Of The CNC Process 45
Companies that use CNC turning centers 45
What will you be doing? 46
Flow of the CNC process 46
Study the workpiece drawing 46
Decision is made as to which CNC machine to use 46
The machining process is developed 47
Tooling is ordered and checked 47
The program is developed 47
Setup and production run documentation is made 47
Program is loaded into the CNC control's memory 47
The setup is made 47
The program is cautiously verified 48
Production is run 48
Corrected version of the program is stored for future use 48
Key points for Lesson Three: 48

Lesson 4: Visualizing The Execution Of A CNC Program 49
Program make-up 50
Method of program execution 50
An example of program execution 50
The CNC program to machine the 2.875 diameter 51
Sequence numbers 52
A note about decimal point programming 52
A decimal point tip 53
Other mistakes of omission 53
Modal words 53
Initialized words 53
Letter O or number zero? 53
Word order in a command 53
Key points for Lesson Four: 54

Lesson 5: Program Zero And The Rectangular Coordinate System 55
Graph analogy 55
More about polarity 56
Wisely choosing the program zero point location 57
In X 57
In Z 57
62
Absolute versus incremental positioning movements 62
Another way to specify absolute and incremental positioning 62
A decimal point reminder 63
Key points for Lesson Five: 64

Lesson 6: Determining Program Zero Assignment Values 65
Program zero must be assigned independently for *each* cutting tool 65
Understanding program zero assignment values 67
What is the zero return position? 67
Program zero assignment values 67
How do you determine program zero assignment values? 69
Will you really have to determine all of these values? 72
Other considerations that affect program zero assignment values 72
How accurate are the program zero assignment values? 72
Trial machining 73
Another advantage of tool touch off probes 73
Tool wear 73
Replacing dull tools during a production run 73
Using cutting tools from one job to the next 74
Key points for Lesson six: 74

Lesson 7: Four Ways To Assign Program Zero 77
Understanding geometry offsets 77
How geometry offsets are instated 78
The four most common ways to assign program zero 79
Using a tool touch off probe to assign program zero – also called a *tool setter* (your 1st choice) 80
Stylus use 81
Procedure to use a tool touch off probe 82
Determining and entering the work shift value 85
Using geometry offsets with work shift to assign program zero (your 2nd choice) 87
Understanding the measure function 88
Using geometry offsets without work shift to assign program zero (your 3rd choice) 91
Assigning program zero in the program with G50 or G92(your last resort) 93
Determining the program zero assignment values for each tool 94
Programming format when program zero is assigned in the program 95
Making the machine more efficient when assigning program zero in the program 97
Key points for Lesson Seven: 98

Lesson 8: Introduction To Programming Words 101
Words allowing a decimal point 101
O 102
N 102
G 102
X 102
Y 102
For turning centers equipped with a Y axis (not many turning centers have this function) this letter address specifies absolute positioning along the Y axis. A decimal point is allowed with the Y word. 102
Z 102
U 103
W 103
C 103
R 103
I & K 103
F 103
E 103
S 104
T 104
M 104
P 104
Q 104
L 105
EOB (end of block character) 105
/ (slash code) 105
G and M codes 106
G codes 106
G code limitation: 106
Option G codes 106
What does initialized mean? 106
What does modal mean? 106
The most popular G codes 106
Common M codes used on a CNC turning center 108
Key points for Eight Seven: 109
Key points: Key Concept one 110

Key Concept 2: You Must Prepare To Write Programs 113

Preparation and time 113
Preparation and safety 114
Typical mistakes 115
Syntax mistakes 115
Motion mistakes 115
Mistakes of omission 115
Process mistakes 116

Lesson 9: Preparation Steps For Programming 117
Prepare the machining process 117
Develop the needed cutting conditions 119
An example 120
Cutting conditions can be subjective 121
Do the required math and mark-up the print 122
Marking up the print 125
Doing the math 126
Check the required tooling 129
Plan the work holding set-up 130
Other documentation needed for the job 131
Production run documentation 131
Program listing 131
Is it all worth it? 131
Key points for Lesson Eight: 131

Key Concept 3: Understand The Motion Types 135

What is interpolation? 135

Lesson 10: Programming The Three Most Basic Motion Types 139
Motion commonalties 139
Understanding the programmed point of each cutting tool 139
G00 Rapid motion (also called positioning) 142
What is a safe approach distance? 144
What about feed-off distance? 145
G01 linear interpolation (straight line motion) 145
Using G01 for a fast feed approach 148
G02 and G03 Circular motion commands 149
Specifying a circular motion with the radius word 149
Circular motion with directional vectors (I and K) 151
What's wrong with this picture? 152
Key points for Lesson Ten: 155

Key Concept 4: Know The Compensation Types 157

More on interpreting tolerances 157
How design engineers specify tolerances 158
High and low limit specified 158
Plus or minus tolerance 158
Plus one value, minus another 159
Is a measured workpiece attribute on-size (acceptable)? 159
What if a measured dimension is not on-size (not acceptable)? 160
More on the target value 160
One more consideration – tool wear 161

Lesson 11: Introduction To Compensation 163
What is compensation and why is it needed? 163
The initial setting for compensation 164
When is trial machining required? 164
What happens as tools begin to wear? 165
What do you shoot for? 165
Why do programmers have to know this? 165
Understanding offsets 165
Offset organization 166
Offset pages on the display screen 166
How offsets are instated 168
Key points for Lesson Eleven: 168

Lesson 12: Geometry Offsets 169
Review of reasons for using geometry offsets to assign program zero 169
How geometry offsets work 169
The total program zero assignment value 170
Warning about the machine lock feature: 171
Minimizing program zero assignment effort from job to job 171
So when do you clear geometry offsets? 173
Key points for Lesson Twelve: 173

Lesson 13: Wear offsets 177
Which dimension do you choose for sizing? 178
How wear offsets are programmed 178
What if my machine doesn't have geometry offsets? 178
What about wear offset cancellation? 179
How wear offsets are entered 180
Which is better, INPUT or +INPUT? 182
What if my machine doesn't have a +INPUT soft key? 182
Sizing in a tool after it has just been placed in the turret 182
Sizing in a new tool with trial machining 184
What causes the initial deviation? 184
Dealing with deviations caused by tool wear 185
After a dull tool is replaced 186
What if my machine has a tool touch off probe? 187
Consistently replacing inserts 187
Consistently indexing inserts 188
Minimizing the need for trial machining 189
Going from job to job 189
When do you clear wear offsets? 189
A more complex example 189
Running the first workpiece – one tool at a time 190
Tool number one: Rough face and turn tool 190
Tool number Two: Finish face and turn tool 191
Tool number Three: 0.125 grooving tool 192
Completing the production run 192
Tool number one: Rough face and turn tool 192

Tool number Two: Finish face and turn tool 192
Tool number Three: 0.125 grooving tool 193
A reminder about up-coming jobs 193
A reminder about target values 193
Secondary wear offset applications 193
Flip jobs 193
Two or more critical diameters 194
Unwanted taper 194
Grooving into different areas of the workpiece 194
Key points for Lesson Thirteen: 194

Lesson 14: Tool Nose Radius Compensation 195
Keeping the cutting edge flush with the work surface at all times 197
When to use tool nose radius compensation 198
Steps to programming tool nose radius compensation 198
Instating tool nose radius compensation 198
Programming motion commands to machine the workpiece 199
Canceling tool nose radius compensation 200
An example program 200
Tool nose radius compensation from a setup person's point of view 201
What if my machine does not have geometry offsets? 203
What if I forget to enter tool nose radius compensation values? 203
What if I enter tool nose radius compensation values into wear offsets? 203
What if I enter tool nose radius compensation values into both the geometry and wear offsets? 203
Programming tool nose radius compensation value entries 203
Another example program showing tool nose radius compensation 203
Key points for Lesson Fourteen: 205

Key Concept 5: You Must Provide Structure To Your CNC Programs 207

Lesson 15: Introduction To Program Structure 209
Objectives of your program structure 209
Reasons for structuring programs with a strict and consistent format 209
Familiarization 209
Consistency 210
Re-running tools in the program 210
Efficiency limitations 212
Machine variations that affect program structure 213
M code differences 213
G code numbering differences 214
Turret variations 215
How do you determine a safe yet efficient index position? 215
What if my machine doesn't have geometry offsets? 216
A reminder about spindle limiting 217
Choosing the appropriate spindle range 218
Which direction do you run the spindle? 218
How do you check what each tool has done? 218
A possible problem with initialized modes 218
How to use our given formats 219
Key points for Lesson Fifteen: 220

Lesson 16: Four Types Of Program Format 221
Format for assigning program zero with geometry offsets 222
Program start-up format (using geometry offsets) 222
Tool ending format (using geometry offsets) 223
Tool start-up format (using geometry offsets) 223
Program ending format (using geometry offsets) 223
Format for assigning program zero in the program with G50 224
A reminder about the program's starting point and tool change position 224
Program start-up format (assigning program zero in the program) 224
Tool ending format (assigning program zero in the program) 225
Tool start-up format (assigning program zero in the program) 225
Program ending format (assigning program zero in the program) 225
Understanding the formats 225
G Words 226
M Words 226
Other M Words Related to Turning Centers 227
Other Words In The Format 227
Example programs showing format for turning centers 228
Example when assigning program zero with geometry offsets 228
Where are the restart commands? 229
Example when assigning program zero in the program 230
Where are the restart commands? 231
Suggestions for cycle time improvements. 231
Combine M codes in motion commands 231
Minimize spindle dead time 232
Efficiently programming spindle range changes 235
Minimize spindle reversals 236
Key points for Lesson Sixteen: 236

Key Concept 6: Special Features That Help With Programming 239

Control series differences 240

Lesson 17: One-Pass Canned Cycles 241
Cycle consistencies 241
G90 - One pass turning and boring cycle 242
G92 - One pass threading cycle 243
G94 - One pass facing command 244

Example of G90 and G94 244
Example of G92 command 245
Key points for Lesson Seventeen: 246

Lesson 18: G71 And G70 – Rough Turning And Boring Followed By Finishing 247
G71 - Rough turning and boring 247
The two phases of G71 248
Understanding G71 command words 250
P word 250
Q word 250
U word 250
W word 250
D word 250
F word 251
What about finishing? 251
Example showing G71 for rough turning and G70 for finish turning 251
Using G71 for rough boring 253
Limitations of the G71 command 255
Different format for 0TA, some 0TBs, 3T, and 18T controls 256
Key points for Lesson Eighteen: 256

Lesson 19: G72: Rough Facing, G73: Pattern Repeating, G74: Peck Drilling, And G75: Grooving 259
G72 - Rough facing 259
Different format for 0TA, some 0TBs, 3T, and 18T controls 261
G73 - pattern repeating 261
D word 262
I word 262
K word 262
Example of G73 pattern repeating 263
Can you use G73 for castings and forgings? 264
Different format for 0TA, some 0TBs, 3T, and 18T controls 265
G74 - peck drilling 265
What if I must clear chips between pecks? 266
Different format for 0TA, some 0TBs, 3T, and 18T controls 267
G75 - grooving cycle 267
Key points for Lesson Nineteen: 268

Lesson 20: G76 – Threading Command 269
X word 269
Z word 270
What is thread chamfering? 270
K word 270
D word 270
A word 271
F word and E word 271
I word 271
Q word 271
Example program for threading 272
Other tips on threading 273
When possible, thread in the lowest spindle range 273
Thread in the rpm mode (G97) 273
Watch out for maximum allowable feedrate 273
Thread with thread chamfering turned off 273
Finish the thread before removing workpiece 273
Right hand threads versus left hand threads 273
Offsetting for threading tools 274
Start the tool far enough away from the thread being machined 274
Minimum depth-of-cut, final pass depth, and number of spring passes 274
Disabled or modified control functions during threading 274
Tapered threads 275
Multiple start threads 276
Different format for 0TA, some 0TBs, and 3T controls 278
What about tapping? 278
Tap in the rpm mode 278
Tap in the low range 278
Use G32 as the motion command for tapping 278
Use a tension/compression tap holder 279
Keep the approach position 0.2 from the Z surface to tap 279
Key points for Lesson Twenty: 279

Lesson 21: Working With Subprograms 281
The difference between main- and sub- programs 281
Loading multiple programs 281
Words used with subprograms 282
A quick example 282
Nesting subprograms 283
Applications for subprograms 284
Repeated machining operations 284
Control programs 284
Utility applications 284
Example for repeating machining operations - multiple identical grooves 285
Example for control program applications - flip jobs 287
Example for utility applications - bar feeder activation 288
Special notes about M99 290
Ending a main program with M99 290
Changing the order of program execution with M99 290
What is parametric programming (custom macro B)? 293
Part families 293
User defined canned cycles 293
Utilities 293
Complex motions and shapes 295
Key points for Lesson Seventeen: 295

Lesson 22: Differences To 0TA, 0TB, 3T, And 18T Controls 297
G71 297
G72 297
G73 297
G74 298
G75 298
G76 298

Subprograms 298
What about controls that are not made by Fanuc? 299
G code differences 299

Lesson 23: Other Special Programming Features 301
Block delete (also called optional block skip) 301
Applications for block delete 302
Another optional stop 302
Trial machining 302
Warning about optional block skip applications 304
Sequence number (N word) techniques) 304
Eliminating sequence numbers 304
Using special sequence numbers in program restart commands 304
Documenting your programs with messages in parentheses 306
General information about the job 306
Tool information 307
At every program stop 307
To document anything out of the ordinary 307
For changes made after a dispute 308
Automatic corner rounding and chamfering 308
Other G codes of interest 310
G04 - Dwell command 310
Other G codes 311
Key points for Lesson Twenty-Three: 312

Appendix: Special Machine Types And Accessories 313
Work holding and work support devices 313
Work holding devices 313
Three jaw chucks 314
Collet chucks 317
Work support devices 319
Tailstocks 319
The tailstock body 319
Tailstock quill 319
Center 320
Tailstock alignment problems 320
Programming considerations 320
Steady rests 321
Bar feeders 321
How a bar feeder works 322
Workholding considerations 322
Styles of bar feeders 322
How to program for bar feeders 323
Determining how much to feed the bar 323
The steps to bar feeding 324
The redundancy of bar feed programming 325
When to program the bar feed 325
Ending a bar feed program 326
An example bar feeding program 326
Part catchers 328
Live tooling 328
Features of live tooling turning centers (also called mill/turn machines) 329
Rotating tools 329
Special tool holders 329
Precise control of main spindle rotation 329
Only one way to specify speed and feedrate 329
Hole machining canned cycles 330
Polar coordinate interpolation 330
Selecting the main spindle mode 330
Programming an indexer 330
Example program for an indexer 331
Programming a rotary axis (C axis) 333
Angular values 333
Zero return position 333
Rapid versus straight line motion 333
Program zero assignment 333
Absolute versus incremental 334
Canned cycles for hole machining 335
How do you specify the machining direction? 335
Canned cycle types 335
Words used in canned cycles 336
An example program 336
Understanding polar coordinate interpolation 337
Other machine types 341
Twin spindle turning centers 341
Sub-spindle turning centers 341
Swiss-type turning centers (also called sliding headstock turning centers) 342

Key Concept 7: Know Your Machine From An Operator's Viewpoint 345

The need for hands-on experience 346
About Key Concept Number Seven 347
Procedures you must know about 347
Machine power-up and power-down 347
Sending the machine to its zero return position 347
Manually moving each axis 347
Manually starting the spindle 348
Manually indexing the turret 348
Manipulating the display screen 348
The position display screen 348
The program display screen 349
The offset display screen 349
The program check display screen 349
Loading and saving CNC programs 349

Lesson 24: Tasks Related To Setup And Running Production 350
A CNC job from start to finish 352
Setup documentation 353
Tear down the previous setup and put everything away 354
Gather the components needed to make the setup 354
Make the workholding setup 354
How To Mount Jaws In The Correct Serrations 355
Machining soft jaws during setup 357
Other devices related to work holding setup 359
Assemble cutting tools 359

A reminder about mounting inserts 359
Load cutting tools into the turret and adjust coolant lines 360
Assign program zero for each new tool 360
Enter tool nose radius compensation values (if the programmer uses this feature) 360
Load the CNC program/s 360
Verify the correctness of a new or modified program 361
Verify the correctness of the setup 362
Dry running our example program 362
Cautiously run the first workpiece 363
The most dangerous time 363
Making sure the first workpiece is a good one 364
Machining the first workpiece in our example job 365
Move through the program one tool at a time 367
Upcoming jobs 367
A note about our example job 368
First workpiece inspection 368
Program optimizing 368
Saving corrected version of the program 368
What will you be doing? 368
Production run documentation 368
Remove the previous workpiece 370
Load the next workpiece 371
Activate the cycle 371
Monitor the cycle 371
Clean and de-burr the workpiece 371
Perform specified measurements 372
Make offset adjustments to maintain size for critical dimensions (sizing) 372
Replace worn tools 373
Clean the machine 373
Preventive maintenance 373
Machine warm-ups 373
Key points for Lesson Twenty-Four: 373

Lesson 25: Buttons And Switches On The Operation Panels 375
The two most important operation panels 375
The control panel buttons and switches 375
Display screen control keys (soft keys) 376
Position display pages 377
Program display pages 377
Offset display pages 377
Program check display pages 378
Other display screen modes 378
The keyboard 379
Letter Keys 379
The slash key (/) 379
Number keys 379
Decimal point key 379
The input key 379
Cursor control keys 379
Program Editing Keys 379
Reset key 379
The machine panel 380
Mode switch 380
Cycle start button 380
Feed hold button 380
Feedrate override switch 381
Rapid traverse override switch 381
Emergency stop button 381
Conditional switches 382
Dry run on/off switch 382
Single block on/off switch 382
Block delete on/off switch (also called optional block skip) 382
Optional stop on/off switch 382
Buttons and switches for manual functions 382
Axis jogging controls 383
Handwheel controls 383
Spindle control 383
Turret index control 383
Indicator lights and meters 384
Spindle rpm and horsepower meters 384
Axis drive-motor horsepower meter 384
Cycle indicator lights 384
Zero return position indicator lights 384
Optional stop indicator light 384
Other buttons and switches on the machine panel 384
Other operation panels on your turning center 384

Key Concept 8: Know The Three Basic Modes Of Operation 385

Lesson 26: The Three Modes Of Operation 385
The manual mode 385
The manual data input mode 386
The manual data input (MDI) mode switch position 386
Commanding an MDI zero return 386
The complete procedure to give an MDI command 387
Commanding an MDI tool change 387
Commanding spindle activation with MDI 387
Other times when MDI is used 388
Can you make motion commands with MDI? 388
The edit mode switch position 388
To make a program in memory the active program (to call up a program) 389
To enter a new program 389
The program operation mode 390
To run the active program from the beginning 390
Key points for Lesson Twenty-Six: 390

Key Concept 9: Understand The Importance Of Procedures 391

Lesson 27: The Key Operation Procedures 392
Key points for Lesson Twenty-Seven: 402

Key Concept 10: You Must Know How To Safely Verify Programs 403

Safety priorities 403

Operator safety 403
Machine tool safety 404
Workpiece safety 404

Lesson 28: Program Verification 405
Two more procedures 405
Canceling the CNC cycle 405
To cancel the cycle 405
To re-run a tool 406
Verifying a job that contains mistakes 408
Loading the program 410
The dry run to check for setup mistakes 411
Cautiously running the first workpiece 412

INDEX 415