Preface 13
Prerequisites 13
Basic Machining Practice Experience 13
Math 14
Motivation 14
Controls Covered 14
Controls other than FANUC 15
Limitations 15
Scope 15
Key Concepts approach 15
Lesson structure 15
Practice, practice, practice! 16
Key Concepts and lessons 16
Enjoy! 17

KNOW THE MACHINING CENTER FROM A PROGRAMMER'S VIEWPOINT 19
Experienced with Conventional (non-CNC) Machine Tools? 19

Lesson 1.1 - Machine Configurations 21
Objectives 21
Introduction 21
Types of CNC turning centers 21
Universal style slant bed turning center 21
Directions of motion (axes) for a universal style slant bed turning center 23
Live tooling for a universal style slant bed turning center 24
Other types of CNC turning centers 25
Chucking style slant bed turning center 25
Twin spindle horizontal bed turning centers 26
Sub-spindle style turning centers 27
Vertical single spindle turning centers 28
Twin spindle vertical turning centers 28
Gang style turning centers 29
Swiss-type CNC turning centers (also called sliding headstock turning centers) 30
Programmable functions of turning centers 30
Spindle 30
Spindle speed 30
Spindle activation and direction 31
Spindle range 31
Feedrate 32
Turret indexing (tool changing) 33
Turret station and offset selection 34
Coolant 34
Other possible programmable functions 34
Tailstock 34
Programmable steady rest 34
Bar feeders and chuck activation 35
Part catcher 35
Tool touch-off probe 35
Automatic tool changing systems 35
Exceptions to X-axis 36
A quick fix 36
Gang style turning centers with cutting tools on both sides of the spindle centerline 36
Center cutting axis 36
What else might be programmable? 36
Key points for lesson 1.1: 37

Lesson 1.2 - Turning Center Speeds and Feeds 39
Objectives 39
Introduction 39
The machining operation to be performed 39
The material to be machined 40
The material of the cutting tool's cutting edge 40
The two ways to select spindle speed 41
When to use constant surface speed mode 42
When to use rpm mode 42
How fast will the spindle be running when constant surface speed is used? 43
How fast can the spindle rotate? 43
How to specify a maximum speed for the constant surface speed mode 44
A potential limitation of constant surface speed 44
The two ways to specify feedrate 45
When to use the feed per revolution mode 45
When to use the feed per minute feedrate mode 45
An example of speed and feed usage 46
Key points for lesson 1.2: 47

Lesson 1.3 - CNC Job Work Flow 49
Objectives 49
Introduction 49
Companies that use CNC turning centers 49
What will you be doing? 50
Flow of the CNC process 50
Study the workpiece drawing 50
Decision is made as to which CNC machine to use 50
The machining process is developed 51
Tooling is ordered and checked 51
The program is developed 51
Setup and production run documentation is made 51
Program is loaded into the CNC control's memory 51
The setup is made 51
The program is cautiously verified 51
Production is run 52
Corrected version of the program is stored for future use 52
Key points for lesson 1.3: 52

Lesson 1.4 - Visualizing CNC Program Execution 53
Objectives 53

Introduction 53
Program make-up 54
Method of program execution 54
An example of program execution 55
A CNC program to machine the 2.875 diameter 55
Sequence numbers 56
A note about decimal point programming 56
A decimal point tip 57
Other mistakes of omission 57
Modal words 57
Initialized words 57
Letter O or number zero? 57
Word order in a command 58
Key points for lesson 1.4: 58

Lesson 1.5 - Understanding The Workpiece Coordinate System 59
Objectives 59
Introduction 59
Graph analogy 59
More about polarity 61
Wisely choosing the program zero point location 62
In X 62
In Z 62
Absolute versus incremental positioning movements 66
Another way to specify absolute and incremental positioning 67
A decimal point reminder 67
We repeat our suggestion about how to specify any real number value: 68
Key points for lesson 1.5: 68

Lesson 1.6 - Geometry Offset and Work Shift Values 69
Objectives 69
Introduction 69
Program zero must be assigned independently for *each* cutting tool 70
Understanding program zero assignment values 71
What is the zero return position? 71
Program zero assignment values 72
How do you determine program zero assignment values? 73
Other kinds of cutting tools 75
Will you really have to determine all of these values? 76
Other considerations that affect program zero assignment values 76
How accurate are the program zero assignment values? 76
Trial machining 77
An advantage of tool touch-off probes 77
Tool wear 77
Replacing dull tools during a production run 77
Using cutting tools from one job to the next 78
Key points for lesson 1.6: 78

Lesson 1.7 - Setting Geometry Offsets 81
Objectives 81
Introduction 81
Understanding geometry offsets 81
How geometry offsets are instated 83
The three most common ways to assign program zero 83
Using a tool touch-off probe to assign program zero – also called a *tool setter* (your 1st choice) 84
Stylus use 85
What about center cutting tools? 86
Procedure to use a tool touch-off probe 86
How does it work? 88
What about the wear offset for a cutting tool that has been probed? 88
What about tool pressure? 88
Determining and entering the work shift value 89
A note about the polarity of the work shift value 91
Using geometry offsets with work shift to assign program zero (your 2nd choice) 91
Understanding the measure function 92
What if my machine doesn't have the measure function? 94
Using geometry offsets without work shift to assign program zero (your 3rd choice) 96
What if my machine doesn't have the measure function? 97
Key points for lesson 1.7: 99

Lesson 1.8 - Introduction to Programming Words 101
Objectives 101
Introduction 101
Words allowing a decimal point 101
O (letter O, not zero) 102
N 102
G 102
X 102
Y 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 104

EOB (end of block character) 104
/ (slash code) 105
G and M-codes 105
G-codes 105
G-code limitation: 105
Option G-codes 105
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 107
Other M-codes for your machine (found in your machine tool builder's manuals) 108
Key points for lesson 1.8: 108

YOU MUST PREPARE TO WRITE PROGRAMS 109
Preparation and time 109
Preparation and safety 110
Typical mistakes 111
Syntax mistakes 111
Motion mistakes 111
Mistakes of omission 111
Process mistakes 111

Lesson 2.1 - Preparation for Programming 113
Objectives 113
Introduction 113
Prepare the machining process 113
Develop the needed cutting conditions 115
Speed and feed formulae: 115
An example 116
Roughing tools 116
Drilling 116
Finishing tools 117
Chasing threads 117
Cutting conditions can be subjective 117
Do the required math and mark-up the print 118
Other ways to come up with coordinates 120
Marking up the print 121
Doing the math 121
Check the required tooling 124
Plan the work holding set-up 125
Other documentation needed for the job 126
Production run documentation 126
Program listing 127
Is it all worth it? 127
Key points for lesson 2.1: 127

YOU MUST UNDERSTAND THE MOTION TYPES 131
What is interpolation? 131

Lesson 3.1 - Motion Commands 135
Objectives 135
Introduction 135
Motion commonalties 135
Understanding the programmed point of each cutting tool 135
G00 Rapid motion (also called positioning) 139
What is a safe approach distance? 140
What about feed-off distance? 141
G01 linear interpolation (straight line motion) 141
Using G01 for a fast feed approach 143
G02 and G03 circular motion commands 144
Specifying a circular motion with the radius word 144
Circular motion with directional vectors (I and K) 146
What's wrong with this picture? 147
Key points for lesson 3.1: 149

YOU MUST KNOW THE COMPENSATION TYPES 151
More on interpreting tolerances 151
How design engineers specify tolerances 152
High and low limit specified 152
Plus or minus tolerance 152
Plus one value, minus another 152
Is a measured workpiece attribute on-size (acceptable)? 153
What if a measured dimension is not on-size (not acceptable)? 153
More on the target value 154
One more consideration – tool wear 154

Lesson 4.1 - Introduction to Compensation 157
Objectives 157
Introduction: What is compensation and why is it needed? 157
The initial setting for compensation 158
When is trial machining required? 158
What happens as tools begin to wear? 159
What do you shoot for? 159
Why do programmers have to know this? 159
Understanding offsets 159
Offset organization 159
Offset pages on the display screen 160
How offsets are instated 162
Key points for lesson 4.1: 162

Lesson 4.2 - Geometry Offsets 163
Objectives 163
Introduction 163
Benefits of geometry offsets 163
How geometry offsets work 163
The total program zero assignment value 164
Warning about the machine lock feature: 165

Minimizing program zero assignment effort from job to job 166
So when do you clear geometry offsets? 167
Key points for lesson 4.2: 167

Lesson 4.3 - Wear Offsets 171
Objectives 171
Introduction 171
Which dimension do you choose for sizing? 172
How wear offsets are programmed 172
What about wear offset cancellation? 172
How wear offsets are entered 173
Which is better, INPUT or +INPUT? 175
What if my machine doesn't have a +INPUT soft key? 175
Sizing in a tool after it has just been placed in the turret 176
Sizing in a new tool with trial machining 177
What causes the initial deviation? 177
Dealing with deviations caused by tool wear 178
After a dull tool is replaced 179
What if my machine has a tool touch-off probe? 179
Consistently replacing inserts 179
Consistently indexing inserts 180
Minimizing the need for trial machining 181
Going from job to job 181
When do you clear wear offsets? 182
A more complex example 182
Running the first workpiece – one tool at a time 183
Tool number one: Rough face and turn tool 183
Tool number Two: Finish face and turn tool 183
Tool number Three: 0.125 grooving tool 184
What about the Z position of the groove? 184
Completing the production run 184
Tool number one: Rough face and turn tool 185
Tool number Two: Finish face and turn tool 185
Tool number Three: 0.125 grooving tool 185
A reminder about up-coming jobs 185
A reminder about target values 185
Secondary wear offset applications 185
Flip jobs 186
Two or more critical diameters 186
Unwanted taper 186
Grooving into different areas of the workpiece 186
Key points for lesson 4.3: 186

Lesson 4.4 - Tool Nose Radius Compensation 187
Objectives 187
Introduction 187
Why tool nose radius compensation is required 187
How much deviation are we talking about? 189
Keeping the cutting edge flush with the work surface at all times 190
When to use tool nose radius compensation 190
Steps to programming tool nose radius compensation 191
Instating tool nose radius compensation 191
Programming motion commands to machine the workpiece 191
Canceling tool nose radius compensation 192
An example program 192
Tool nose radius compensation from a setup person's point of view 193
What if my machine does not have geometry offsets? 195
What if I forget to enter tool nose radius compensation values? 195
What if I enter tool nose radius compensation values into wear offsets? 195
What if I enter tool nose radius compensation values into both the geometry and wear offsets? 195
Programming tool nose radius compensation value entries 195
Another example program showing tool nose radius compensation 196
Key points for lesson 4.4: 198

YOU MUST PROVIDE STRUCTURE TO YOUR PROGRAMS 201

Lesson 5.1 - Introduction to Program Structure 203
Objectives 203
Introduction: 203
Objectives of your program structure 203
Reasons for structuring programs with a strict and consistent format 204
Familiarization 204
Consistency 204
Re-running tools in the program 204
Efficiency limitations 206
Machine variations that affect program structure 207
M-code differences 207
G-code numbering differences 208
Turret variations 208
How do you determine a safe yet efficient index position? 209
What if my machine doesn't have geometry offsets? 210
A reminder about spindle limiting 210
Choosing the appropriate spindle range 211
Which direction do you run the spindle? 211
How do you check what each tool has done? 211
A possible problem with initialized modes 212
How to use our given formats 212
Key points for lesson 5.1: 213

Lesson 5.2 - Structured Program Format 215
Objectives 215
Introduction 215
Program structure 216
Program startup structure 216
Tool startup structure 216
Tool ending structure 216
Program ending structure 217
Understanding the structures 217
G Words 217
M Words 217
Other M Words Related to Turning Centers 218
Other words In The Structure 218
Example program showing structure for turning centers 219
Where are the restart commands? 220
Suggestions for cycle time improvements 220
Combine M-codes in motion commands 221
Minimize spindle dead time 221
Efficiently programming spindle range changes 224
Minimize spindle reversals 225
Key points for lesson 5.2: 225

MASTER SPECIAL FEATURES THAT HELP WITH PROGRAMING 229
Control series differences 230

Lesson 6.1 - One-Pass Canned Cycles 231
Objectives 231
Introduction: 231
Cycle consistencies 231
G90 - One pass turning and boring cycle 232
G92 - One pass threading cycle 233
G94 - One pass facing command 234
Example of G90 and G94 234
Example of G92 command 235
Key points for lesson 6.1: 236

Lesson 6.2 - Rough and Finish Turing and Boring (G70 and G71) 237
Objectives 237
Introduction 237
G71 - Rough turning and boring 237
The two phases of G71 238
Understanding G71 command words 240
P-word (P-word in second line) 240
Q-word (Q-word in second line) 240
U-word (U-word in second line) 241
W-word (W-word in second line) 241
D word (U-word in first line) 241
F-word (F-word in second line) 241
(R-word in second line) 241
What about finishing? 241
Example showing G71 for rough turning and G70 for finish turning 241
Using G71 for rough boring 244
Limitations of the G71 command 245
No recesses 245
Efficiency 246
Format for two-line multiple repetitive cycles 246
Key points for lesson 6.2: 246

Lesson 6.3 - Other Multiple Repetitive Cycles (G72-G75) 249
Objectives 249
Introduction 249
G72 - Rough facing 249
Format for two-line multiple repetitive cycles 251
G73 - pattern repeating 251
D word (R word in first G73 command) 252
I-word (U-word in second G73 command) 252
K-word (W-word in second G73 command) 252
Example of G73 pattern repeating 253
Can you use G73 for castings and forgings? 254
Format for two-line multiple repetitive cycles 255
G74 - peck drilling 255
What if I must clear chips between pecks? 256
Format for two-line multiple repetitive cycles 256
G75 - grooving cycle 257
Key points for lesson 6.3: 258

Lesson 6.4 - Threading Multiple Repetitive Cycle (G76) 259
Objectives 259
Introduction 259
Programming G76 259
X-word (X-word in second G76) 260
Z-word (Z-word in second G76) 260
What is thread chamfering? 260
K-word (P-word in second G76 - does not allow decimal point) 260
D-word (Q-word in second G76) 261
A-word (last two digits of P word in first G76) 261
F-word and E-word (F word in second G76) 261
I-word (R-word in second G76) 262
Q-word 262
(P-word in first G76) 262
(Q-word in first G76) 262
(R-word in first G76) 262
Example program for threading 263
Other notes about threading 264
When possible, thread in the lowest spindle range 264
Thread in the rpm mode (G97) 264
Watch out for maximum allowable feedrate 264
Thread with thread chamfering turned off 264
Finish the thread before removing workpiece 264
Right hand threads versus left hand threads 264

Offsetting for threading tools 265
Start the tool far enough away from the thread being machined 265
Minimum depth-of-cut, final pass depth, and number of spring passes 265
Disabled or modified control functions during threading 265
Tapered threads 265
Multiple start threads 267
Format for two-line multiple repetitive cycles 268
What about tapping? 269
Tap in the rpm mode 269
Tap in the low range 269
Use G32 as the motion command for tapping 269
Use a tension/compression tap holder 269
Keep the approach position 0.2 inch from the Z surface to tap 270
Key points for lesson 6.4: 270

Lesson 6.5 - Sub-Programming Techniques 271
Objectives 271
Introduction 271
The difference between main- and sub- programs 271
Loading multiple programs 272
Words used with subprograms 272
A quick example 272
Nesting subprograms 274
Applications for subprograms 274
Repeated machining operations 274
Control programs 274
Utility applications 275
Example for repeating machining operations - multiple identical grooves 275
Example for control program applications - flip jobs 277
Example for utility applications - bar feeder activation 278
Special notes about M99 280
Ending a main program with M99 280
Changing the order of program execution with M99 280
What is parametric programming (Custom Macro)? 282
Part families 283
User defined canned cycles 283
Utilities 283
Complex motions and shapes 284
Key points for lesson 6.5: 285

Lesson 6.6 - Control Model Differences 287
Objectives 287
Introduction 287
G71 287
G72 287
G73 288
G74 288
G75 288
G76 288
Sub-programming differences 289
What about controls that are not made by FANUC? 289
G-code differences 289

Lesson 6.7 - Other Special Programming Features 291
Objectives 291
Introduction 291
Block delete (also called optional block skip) 291
Applications for block delete 292
Another optional stop 292
Trial machining 293
Warning about block delete applications 294
Sequence number (N-word techniques) 294
Eliminating sequence numbers 294
Using special sequence numbers in program restart commands 294
Documenting your programs with messages in parentheses 296
General information about the job 296
Tool information 297
At every program stop 297
To document anything out of the ordinary 297
For changes made after a dispute 298
Automatic corner rounding and chamfering 298
Other G-codes of interest 300
G04 - Dwell command 300
Other G-codes 301
Key points for lesson 6.7: 301

Appendix: Special Machine Types and Accessories 303
Introduction 303
Work holding and work support devices 303
Work holding devices 303
Three jaw chucks 303
Programmable features of three jaw chucks 306
Collet chucks 307
Work support devices 308
Tailstocks 308
The tailstock body 309
Tailstock quill 309
Tailstock center 309
Tailstock alignment problems 310
Programming considerations 310
Steady rests 310
Bar feeders 311
How a bar feeder works 311
Workholding considerations 312
Styles of bar feeders 312
How to program for bar feeders 312

Determining how much to feed the bar 313
The steps to bar feeding 313
The redundancy of bar feed programming 315
When to program the bar feed 315
Ending a bar feed program 315
An example bar feeding program 315
Part catchers 317
Live tooling 318
Features of live tooling turning centers (also called mill/turn or turn/mill machines) 318
Rotating tools 318
Special tool holders 318
Precise control of main spindle rotation 319
Only one way to specify speed and feedrate 319
Hole-machining canned cycles 319
Polar coordinate interpolation 319
Selecting the main spindle mode 319
Programming an indexer 320
Example program for an indexer 321
Programming a rotary axis (C-axis) 322
Angular values 322
Zero return position 322
Rapid versus straight line motion 322
Program zero assignment 323
Absolute versus incremental 323
Canned cycles for hole machining 324
Canned cycle types 324
Words used in canned cycles 325
An example program 325
Understanding polar coordinate interpolation 326
Other machine types 330
Twin spindle turning centers 330
Sub-spindle turning centers 331
Swiss-type turning centers (also called sliding headstock turning centers) 331

KNOW YOUR MACHINE FROM AN OPERATOR'S VIEWPOINT 333
Are you only interested in setup and operation? 333
The need for hands-on experience 334

Lesson 7.1 - Tasks Related to Setup and Running Production 335
Objectives 335
Introduction: 335
A CNC job from start to finish 337
Setup documentation 338
Tear down the previous setup and put everything away 339
Gather the components needed to make the setup 339
Make the workholding setup 339
How To Mount Jaws In The Correct Serrations 339
Machining soft jaws during setup 341
Other devices related to work holding setup 344
Assemble cutting tools 344
A reminder about mounting inserts 344
Load cutting tools into the turret and adjust coolant lines 344
Assign program zero for each new tool 344
Enter tool nose radius compensation values (if the programmer uses this feature) 345
Load the CNC program/s 345
Verify the correctness of a new or modified program 345
Verify the correctness of the setup 346
Dry running our example program 347
Cautiously run the first workpiece 347
The most dangerous time 348
Making sure the first workpiece is a good one 348
Machining the first workpiece in our example job 349
Move through the program one tool at a time 351
Upcoming jobs 351
A note about our example job 351
First workpiece inspection 351
Program optimizing 352
Saving corrected version of the program 352
What will you be doing? 352
Production run documentation 353
Remove the previous workpiece 354
Load the next workpiece 355
Activate the cycle 355
Monitor the cycle 355
Clean and de-burr the workpiece 355
Perform specified measurements 356
Make offset adjustments to maintain size for critical dimensions (sizing) 356
Replace worn tools 356
Clean the machine 357
Preventive maintenance 357
Machine warm-ups 357
Key points for lesson 7.1: 357

Lesson 7.2 - Buttons and Switches on the Operation Panels 359
Objectives 359
Introduction 359
The two most important operation panels 359
The control panel buttons and switches 360
Display screen mode keys 361
Position display pages 362
Program display pages 362
Offset display pages 363
Graph display pages 364
Other display screen modes 364

The keyboard 364
Letter Keys 365
The slash key (/) 365
Number keys 365
Decimal point key 365
CAN key 365
EOB key 365
The input key 365
Cursor control keys 365
Program Editing Keys 365
Reset key 365
The machine panel 366
Power buttons 366
MODE switch 366
CYCLE START button 366
FEED HOLD button 366
FEEDRATE OVERRIDE switch 367
RAPID OVERRIDE switch 367
EMERGENCY STOP button 367
Conditional switches 367
DRY RUN on/off switch 368
SINGLE BLOCK on/off switch 368
BLOCK DELETE on/off switch (also called optional block skip) 368
OPTIONAL STOP on/off switch 368
Buttons and switches for manual functions 368
Axis jogging controls (continuous jog) 368
Axis jogging controls (incremental jog) 369
Handwheel controls 369
Spindle control 369
Turret index control 369
Indicator lights and meters 369
Spindle rpm and horsepower meters 369
Axis drive-motor horsepower meter 369
Cycle indicator lights 370
Zero return position indicator lights 370
Optional stop indicator light 370
Other buttons and switches on the machine panel 370
Other operation panels on your turning center 370

KNOW THE MODES OF OPERATION 371

Lesson 8.1 - Operation Modes 373
Objectives 373
Introduction: 373
The manual mode switch positions 373
The MDI mode switch position 374
Commanding an MDI zero return 374
The complete procedure to give an MDI command 374
Commanding an MDI turret index 375
Commanding spindle activation with MDI 375
Other times when MDI is used 375
Can you make motion commands with MDI? 375
The EDIT mode switch position 375
To make a program in memory the active program (to call up a program) 376
To enter a new program 376
What if I make a mistake when typing? 377
The program operation mode 377
To run the active program from the beginning 377
Key points for lesson 8.1: 378

UNDERSTAND THE IMPORTANCE OF PROCEDURES 379

Lesson 9.1 - Operation Procedures 381
Objectives 381
Introduction 381
Develop your own operation handbooks 381
Manual procedures: 381
MDI Procedures 381
Program Manipulation Procedures 381
Setup Procedures 382
Program Running Procedures 382
Blank procedure form 383
Sample operation handbook (Levil certification cart - lathe) 384
Manual Procedures 384
To start the machine 384
To do a zero return 384
To index the turret 384
To start spindle 384
To jog axes (using continuous jog) 384
To jog axes (using incremental jog) 384
To use the handwheel 385
To set axis displays 385
To enter wear offsets 385
MDI Procedures 385
To execute an MDI command 385
Program Manipulation Procedures 386
To get ready to edit programs 386
To show a directory of programs 386
To call up a program from with the CNC memory (make it the active program) 386
To load programs 386
To delete programs 386
To search within a program 387
To alter, insert, & delete 387
To save programs 388
To use background edit 388
Setup Procedures 388
To mount jaws 388
To measure and enter program zero assignment values 388
To measure and enter work shift value 389

Program Running Procedures 389
To run the program (in normal production - no verification techniques) 389
To rerun tools 389
To do a free flowing dry run 390
To do an normal air cutting run 390
To run the first workpiece 390
To cancel a cycle 391
To clear an alarm 391
Key points for lesson 9.1: 391

YOU MUST KNOW HOW TO SAFELY VERIFY PROGRAMS 393
Safety priorities 393
Operator safety 393
Machine tool safety 394
Workpiece safety 394

Lesson 10.1 - Program Verification 395
Objectives 395
Introduction 395
Two more procedures 395
Canceling the CNC cycle 395
To cancel the cycle 396
To re-run a tool 396
Verifying a job that contains mistakes 398
Loading the program 400
The dry run to check for setup mistakes 401
Cautiously running the first workpiece 402
Review of program running procedures 403
Program Running Procedures 403
To do a free flowing dry run 403
To do an normal air cutting run 403
To run the first workpiece 403
To rerun tools 404
To run the program (in normal production - no verification techniques) 404
To cancel a cycle 404
To clear an alarm 404
Key points for lesson 10.1: 404

Index 405
Quick Reference Sheet 411