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

Key Concept 1: Know your machine from a programmer's viewpoint 17

Lesson 1: ***Machine configurations 19***
Vertical machining centers 19
C-frame style 19
Directions of motion (axes) for a C-frame style vertical machining center 20
Axis polarity 20
Knee style vertical CNC milling machines 21
Bridge-style vertical machining center (also called gantry-style) 22
Horizontal machining centers 23
Directions of motion (axes) for a horizontal machining center 24
Axis polarity 24
Programmable functions of machining centers 26
Spindle 26
Spindle speed 26
Spindle activation and direction 26
Spindle range 26
Feedrate 27
Coolant 28
Automatic tool changer 28
Measurement system mode (inch or metric) 29
What else might be programmable? 30
Key points for Lesson One: 30

Lesson 2: ***General flow of the CNC process 31***
Companies that use CNC machining centers 31
What will you be doing? 32
Flow of the CNC process 32
Study the workpiece drawing 32
Decision is made as to which CNC machine to use 32
The machining process is developed 33
Tooling is ordered and checked 33
The program is developed 33
Setup and production run documentation is made 33
Program is loaded into the CNC control's memory 33
The setup is made 33
The program is cautiously verified 34
Production is run 34
Corrected version of the program is stored for future use 34
Key points for Lesson Two: 34

Lesson 3: ***Visualizing the execution of a CNC program 35***
Program make-up 36
Method of program execution 36
An example of program execution 36
Manual milling machine procedure: 37
CNC program: 37
Sequence numbers 38
A note about decimal point programming 38
A decimal point tip 39
Other mistakes of omission 39
Modal words 39
Initialized words 39
Letter O or number zero? 39
Word order in a command 40
Key points for Lesson Three: 40

Lesson 4: ***Program zero and the rectangular coordinate system 41***
Graph analogy 41
What about the Z axis? 42
Understanding polarity 43
Wisely choosing the program zero point location 46
In X and Y 46
Reminder about axis movement 47

In Z 47
Absolute versus incremental positioning modes 49
A decimal point reminder 50
Key points for Lesson Four: 50

Lesson 5: ***Determining program zero assignment values 51***
Program zero assignment values 51
What is zero return position? 52
Determining program zero assignment values 53
How to manually measure the program zero point location on the machine 53
In X and Y for a square or rectangular workpiece 53
In X and Y for a round workpiece 55
In Z 56
Measuring the program zero point location using a spindle probe 57
Again, do you have to measure the program zero point? 58
How program zero point selection can affect the setup person 58
Key points for Lesson Five: 59

Lesson 6: ***Assigning program zero 61***
Understanding fixture offsets 61
A full example of program zero assignment 62
The program 62
The setup 63
What is the common fixture offset? 64
What if my machining center doesn't have fixture offsets? 64
Advantages of assigning program zero with fixture offsets instead of G92 64
Safety 64
Ease of use 64
Efficiency 64
Rerunning tools 64
Assigning program zero in the program 65
Key points for Lesson Six: 65

Lesson 7: ***Introduction to programming words 67***
Words allowing a decimal point 67
O 68
N 68
G 68
X 68
Y 68
Z 68
A 68
B 69
C 69
R 69
I, J, K 69
Q 70
P 70
L 70
F 70
S 70
T 70
M 71
D 71
H 71
EOB (end of block character) 71
/ (slash code) 71
G and M codes 72
G codes 72
G code limitation: 72
Option G codes 72
What does initialized mean? 72
What does modal mean? 72
The most popular G codes 72
Common M codes used on a CNC machining center 74
Key points for Lesson Seven: 75
Key points for Key Concept one 76

Key Concept 2: You must prepare to write programs 77

Preparation and time 77
Preparation and safety 78
Typical mistakes 79
Syntax mistakes 79
Motion mistakes 79
Mistakes of omission 79
Process mistakes 79

Lesson 8: ***Preparation steps for programming 81***
Prepare the machining process 81
Develop the needed cutting conditions 83
An example 84
Cutting conditions can be subjective 84
Do the required math and mark-up the print 85
Marking up the print 87
Doing the math 88
What about milling operations? 89
Check the required tooling 90
Plan the work holding set-up 91
Other documentation needed for the job 92
Production run documentation 93

Program listing 93
Is it all worth it? 93
Key points for Lesson Eight: 93

Key Concept 3: Understand the motion types 95

What is interpolation? 95

Lesson 9: ***Programming the three most basic motion types 99***
Motion commonalties 99
Understanding the programmed point of each cutting tool 99
Center drill 100
Spot drill 100
Drill 100
Reamer 100
Tap 100
Boring bar 100
What about milling cutters? 101
G00 – Rapid motion (also called positioning) 102
How many axes can be included in a rapid motion command? 103
About the dog-leg motion... 103
When do you use rapid motion? 104
What is a safe approach distance? 104
G01 – Linear interpolation (straight-line motion) 105
Using G01 for a fast-feed approach 106
A milling example 107
Drill holes with G01? 107
G02 and G03 – Circular interpolation (circular motion) 108
Which positions to program 108
Specifying circular motion with the R word 108
The R word is not modal 110
More details about clockwise versus counter clockwise circular motion 110
Circular motion with directional vectors (I, J, and K) 111
Arc limitations 112
Full circle in one command 113
Planning your own tool paths 115
Key points for Lesson Nine: 115

Key Concept 4: Know the compensation types 117

Lesson 10: ***Introduction to compensation 119***
What is compensation and why is it needed? 119
More on tolerances 120
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
Offsets related to cutting tools 122
Offsets related to program zero assignment 123
How offsets are instated 123
Key points for Lesson Ten: 124

Lesson 11: ***Tool length compensation 125***
The reasons why tool length compensation is needed 125
Programming tool length compensation 126
Choosing the offset number to be used with each tool 126
An example program 127
The setup person's responsibilities with tool length compensation 128
Recommended method: Using the tool's length as the tool length compensation (offset) value 128
Determining tool length compensation values 129
Entering tool length compensation offsets 131
Using the distance from the tool tip to program zero as the tool length compensation (offset) value 132
Determining tool length compensation values with this second method 132
Reasons for using our recommended method 133
Typical mistakes with tool length compensation 133
Forgetting to instate tool length compensation 133
Forgetting to enter the tool length compensation value 134
Mismatching offsets 134
What if my machine doesn't have fixture offsets? 134
Trial machining with tool length compensation 134
When trial machining is not required 135
Sizing with tool length compensation 135
A tip for remembering which way to adjust the offset 135

What if I use the second method shown for tool length compensation? 135
Do I have to make all these calculations when adjusting offsets? 135
Why can't I just change the Z coordinate/s in the program to make sizing adjustments? 136
Key points for Lesson Eleven: 136

Lesson 12: ***Cutter radius compensation 139***
Reasons why cutter radius compensation is required 139
Calculations are simplified for manual programmers 139
Range of cutter sizes 141
Do you use re-sharpened (re-ground) cutters? 142
Trial machining and sizing 143
Rough and finish milling with the same set of coordinates 143
How cutter radius compensation works 144
Steps to programming cutter radius compensation 145
Step one: Instate cutter radius compensation 145
The XY motion to the prior position 145
The Z motion/s to the Z axis work surface 147
The command instating cutter radius compensation that positions the cutting tool to the first surface to mill 147
G41 or G42? 147
The offset used with cutter radius compensation 148
The motion to the first work surface 149
Step two: Program the tool path to be machined 150
Step three: Cancel cutter radius compensation 152
What if I have more than one contour to mill? 153
Examples 154
What if I use a computer aided manufacturing (CAM) system to prepare programs? 156
The setup person's responsibilities with cutter radius compensation 156
Rough and finish milling with the same set of tool path coordinates 156
A warning 158
Trial machining with cutter radius compensation 158
159
Is the milling cutter machining on both sides of the workpiece (are you measuring over two machined surfaces)? 159
A tip for remembering which way to adjust the offset 160
How important is it that you make your first workpiece a good one? 160
When trial machining is not required 160
Sizing with cutter radius compensation 160
What if I use a CAM system and the offset value is the deviation from the planned cutter size? 161
Do I have to make all these calculations when adjusting offsets? 161
Why can't I just change programmed coordinate/s in the program to make sizing adjustments? 161
Key points for Lesson Twelve: 161

Lesson 13: ***Fixture offsets 165***
Assigning multiple program zero points 166
Programming with multiple program zero points 168
The potential trade-off with this method 169
Reminder about tool length compensation values 169
Shifting the point of reference for fixture offset entries 170
Programming fixture offset entries 172
Some other applications for the common fixture offset 173
Allowing for variations in pallet changers 173
Allowing for variations after a mishap 173
Differences in spindle gap from one machine to another 173
To enhance safety during dry-runs 173
Key points for Lesson Thirteen: 173

Key Concept 5: You must provide structure to your CNC programs 175

Lesson 14: ***Introduction to program structure 177***
Objectives of your chosen program structure 177
Reasons for structuring programs with a strict and consistent format 177
Familiarization 177
Consistency 178

Re-running tools in the program 178
Efficiency limitations 179
Machine variations that affect program structure 179
M code differences 179
Automatic tool changer variations 180
T word brings a tool to the ready station, M06 commands the tool change 180
T word does everything 182
Tool change at beginning or end? 183
Does the machine even have an automatic tool changer? 183
Understanding the G28 command 183
What about G53? 184
A possible problem with initialized modes 185
How to use our given formats 185
Key points for Lesson Fourteen: 186

Lesson 15: ***Four types of program format 187***
Format for vertical machining centers 187
Program Start-Up Format: 188
Tool Ending Format: 188
Tool Start-Up Format: 188
Program Ending Format: 188
A note about documentation 190
Example program for vertical machining centers 191
More on the optional stop word (M01) 192
Where is the restart command for each tool? 192
What if my machine doesn't have fixture offsets? 193
Format for horizontal machining centers 194
Program Start-Up Format: 194
Tool Ending Format: 195
Tool Start-Up Format: 195
Program Ending Format: 195
Key points for Lesson Fifteen: 196

Key Concept 6: Special features that help with programming 199

Lesson 16: ***Hole-machining canned cycles 201***
Canned cycle commonalities 202
Description of each canned cycle 202
G80 – Cancel the canned cycle mode 202
G81 –Standard drilling cycle 202
G73 – Chip-breaking peck drilling cycle 202
G83 – Deep-hole drilling cycle (full retract between pecks) 202
G84 – Right-hand tapping cycle 203
G74 – Left-hand tapping cycle 205
G82 – Counter-boring cycle 205
G89 – Counter-boring cycle for a boring bar 205
G86 – Standard boring cycle (leaves drag line witness mark) 206
G76 – Fine boring cycle (leaves no witness mark) 206
G85 – Reaming cycle (most programmers use G81 for reaming) 208
G87 and G88 – Manual cycles (not recommended) 208
Words used in canned cycles 208
A simple example 208
Understanding G98 and G99 210
Canned cycles and the Z axis 212
Extended example showing canned cycle usage 214
Using canned cycles in the incremental positioning mode 218
Key points for Lesson Sixteen: 220

Lesson 17: ***Working with subprograms 223***
The difference between main- and sub-programs 223
Loading multiple programs 225
Words used with subprograms 225
Nesting subprograms 226
Machining multiple identical pockets 226
Understanding G52 – temporary shift of program zero 228
Multiple hole-machining operations on a series of holes 228
Rough and finish contour milling 230
Two utility applications for subprograms 231
Control programs 231
A trick to get more fixture offsets 232
What is parametric programming (custom macro B)? 234
Key points for Lesson Seventeen: 236

Lesson 18: ***Other special programming features 237***
Block delete (also called optional block skip) 237
Applications for block delete 238
Another optional stop 238
Trial machining 238
Sequence number (N word) techniques) 241
Eliminating sequence numbers 241

Using special sequence numbers for each tool start up 241
Using sequence numbers as statement labels 242
Using block delete to exit a series of commands 242
Using statement labels to change machining order 243
Other G codes of interest 244
Thread milling, G02 & G03 244
G04 - Dwell command 246
G09 and G61 - Exact stop check 247
G10 - Offset setting by programmed command 248
Applications for G10 248
Polar coordinates (G15 and G16) 250
Plane selection commands (G17, G18, and G19) 251
Inch/metric mode selection G20, G21 252
Secondary reference position, G30 253
Scaling commands, G50 & G51 253
G50.1 and G51.1 - Mirror image commands 253
Applications for mirror image 253
The two ways to activate mirror image 254
Motion relative to zero return position, G53 255
Single direction positioning mode, G60 255
Coordinate rotation G68 and G69 256
Key points for Lesson Eighteen: 258

Lesson 19: ***Programming rotary devices 259***
The difference between an indexer and a rotary axis 259
A note to horizontal machining center programmers 259
Benefits of rotary devices 260
Indexers 260
Programming indexer rotation 260
90 degree and 45 degree indexers 260
Five degree indexers 260
One degree indexer 261
Rotary axes 261
How to program a rotary axis departure 261
Comparison to other axes 262
Zero return position 262
Polarity 263
Designation of program zero 263
Absolute positioning mode 264
Incremental positioning mode 267
Rapid and straight line motion 269
Canned cycle usage 270
Approaching rotary device applications 270
Program zero point selection 270
Assigning one program zero point per side 272
Using one central program zero point 272
Example program using rotary device 274
Key points for Lesson Nineteen: 280

Key Concept 7: Know your machine from an operator's viewpoint 281

The need for hands-on experience 282
More on axis polarity 283
Procedures you must know about 285
Machine power-up 285
Sending the machine to its zero return position 285
Manually moving each axis 285
An example of manual axis movement 286
Manually starting the spindle 286
Manually making tool changes 287
Manipulating the display screen 287
The position display screen 287
The program display screen 287
The offset display screen 288
The program check display screen 288
Loading programs 288

Lesson 20: ***Tasks related to setup and running production 289***
Setup-related tasks 292
Tear down the previous setup and put everything away 293
Gather the components needed to make the setup 293
Make the workholding setup 293
Assign the program zero point 294
Measure program zero assignment values 294
Enter program zero assignment values into fixture offset number one 296
Assemble the cutting tools needed for the job 296
Measure tool length- and cutter-radius-compensation values 297
The static nature of cutting tool measurements 298
Enter tool length and cutter radius compensation values into offsets 298

Load cutting tools into the machine's automatic tool changer magazine 299
Load the CNC program 299
Verify the correctness of a new or modified program 299
Verify the correctness of the setup 301
Dry running our example program 301
Cautiously run the first workpiece 302
The most dangerous time 303
Making sure the first workpiece is a good one 304
If necessary, optimize the program for better efficiency 306
If changes have been made to the program, save the corrected version of the program 306
Production-run-tasks 307
Remove the previous workpiece 308
Load the next workpiece 308
Activate the cycle 308
Monitor the cycle 309
Clean and de-burr the workpiece 309
Perform specified measurements 310
Make offset adjustments to maintain size for critical dimensions 310
Replace worn tools 311
Clean the machine 311
Preventive maintenance 311
Key points for Lesson Twenty: 311

Lesson 21: ***Buttons and switches on the operation panels 313***
The two most important operation panels 313
The control panel buttons and switches 313
Display screen control keys (soft keys) 314
Position display pages 315
Program display pages 315
Offset display pages 315
Program check display pages 316
Other display screen modes 316
The keyboard 317
Letter Keys 317
The slash key (/) 317
Number keys 317
Decimal point key 317
The input key 317
Cursor control keys 317
Program Editing Keys 317
Reset key 317
The machine panel 318
Mode switch 318
Cycle start button 318
Feed hold button 318
Feedrate override switch 319
Rapid traverse override switch 319
Emergency stop button 319
Conditional switches 320
Dry run on/off switch 320
Single block on/off switch 320
Block delete on/off switch (also called optional block skip) 320
Optional stop on/off switch 320
Buttons and switches for manual functions 320
Axis jogging controls 321
Handwheel controls 321
Spindle control 321
Automatic tool changer control 321
Indicator lights and meters 322
Spindle rpm and horsepower meters 322
Axis drive-motor horsepower meter 322
Cycle indicator lights 322
Zero return position indicator lights 322
Optional stop indicator light 322
Other buttons and switches on the machine panel 322
Other operation panels on your machining center 322

Key Concept 8: Know the three basic modes of operation 323

Lesson 22: ***The three modes of operation 323***
The manual mode 323
The manual data input mode 324
The manual data input (MDI) mode switch position 324
Commanding an MDI zero return 324
The complete procedure to give an MDI command 325
Commanding an MDI tool change 325
Commanding spindle activation with MDI 325
Other times when MDI is used 326
Can you make motion commands with MDI? 326
The edit mode switch position 326
To make a program in memory the active program (to call up a program) 327
To enter a new program 327
The program operation mode 328
To run the active program from the beginning 328

Key Concept 9: Understand the importance of procedures 329

Lesson 23: ***The key operation procedures 330***

Key Concept 10: You must know how to safely verify programs 341

Safety priorities 341
Operator safety 341
Machine tool safety 342
Workpiece safety 342

Lesson 24: ***Program verification 343***
Two more procedures 343
Canceling the CNC cycle 343
To re-run a tool 344
Verifying a job that contains mistakes 346
Loading the program 349
The dry run to check for setup mistakes 350
Cautiously running the first workpiece 352

INDEX 353

 © CNC Concepts, Inc.