Table of Contents

Preface 7

Prerequisites 7

Basic machining practice experience 8

Controls covered 8

Limitations 8

Programming method 8

The need for hands-on practice 9

Instruction method 9

Scope 9

Key Concepts approach 9

Lesson structure 10

Practice makes perfect 10

Key Concepts and lessons 10

Key concept 1: Know Your Machine From A Programmer's Viewpoint 11

Lesson 1: Machine Configurations 13

Vertical machining centers 13

C-frame style 14

Directions of motion (axes) for a C-frame style vertical machining center 14

Axis polarity 15

Knee style vertical CNC milling machines 16

Bridge-style vertical machining center (also called gantry-style) 17

Horizontal machining centers 17

Directions of motion (axes) for a horizontal machining center 18

Axis polarity 19

Programmable functions of machining centers 20

Spindle 20

Spindle speed 20

Spindle activation and direction 20

Spindle range 21

Feedrate 21

Coolant 22

Automatic tool changer 22

Measurement system mode (inch or metric) 23

What else might be programmable? 24

Key points for Lesson One: 24

Lesson 2: Visualizing The Execution Of A CNC Program 27

Program make-up 28

Method of program execution 28

An example of program execution 28

Manual milling machine procedure: 29

CNC program: 29

Sequence numbers 30

A note about decimal point programming 30

A decimal point tip 31

Other mistakes of omission 31

Modal words 31

Initialized words 31

Letter O or number zero? 31

Word order in a command 32

Lesson 3: Program Zero And The Rectangular Coordinate System 33

Graph analogy 33

What about the Z axis? 34

Understanding polarity 35

Wisely choosing the program zero point location 37

In X and Y 38

Reminder about axis movement 39

In Z 39

Absolute versus incremental positioning modes 41

A decimal point reminder 42

How the program zero location is specified in the program 42

Setup-related tasks for program zero assignment 43

Lesson 4: Introduction To Programming Words 47

Words allowing a decimal point 47

O 48

N 48

G 48

X 48

Y 48

Z 48

A 48

B 49

C 49

Note about rotary axis designators and indexer activators 49

R 49

I, J, K 50

Q 50

P 50

L 50

F 50

S 51

T 51

M 51

D 51

H 51

EOB (end of block character) 51

/ (slash code) 52

G and M codes 52

G codes 52

G code limitation: 52

Option G codes 52

What does initialized mean? 52

What does modal mean? 53

The most popular G codes 53

Common M codes used on a CNC machining center 55

Other M Codes for your machine (found in your machine tool builder's manuals) 55

Lesson one – Machine configurations: 56

Lesson two – Visualizing program execution: 56
Lesson three – Program zero and the rectangular coordinate system: 56
Lesson four – Introduction to CNC words: 56

Key Concept 2: You Must Prepare To Write Programs 57

Preparation and time 57
Preparation and safety 58
Typical mistakes 59
Syntax mistakes 59
Motion mistakes 59
Mistakes of omission 59
Process mistakes 60

Lesson 5: Preparation Steps For Programming 61

Prepare the machining process 61
Develop the needed cutting conditions 63
An example 64
Cutting conditions can be subjective 64
Do the required math and mark-up the print 65
Marking up the print 67
Doing the math 68
What about milling operations? 69
Check the required tooling 70
Plan the work holding set-up 71
Other documentation needed for the job 72
Production run documentation 72
Program listing 73
Is it all worth it? 73

Key Concept 3: Understand The Motion Types 75

What is interpolation? 75

Lesson 6: Programming The Three Most Basic Motion Types 79

Motion commonalties 79
Understanding the programmed point of each cutting tool 79
Center drill 80
Spot drill (not shown above) 80
Drill 80
Reamer 80
Tap 80
Boring bar 81
What about milling cutters? 81
G00 – Rapid motion (also called positioning) 82
How many axes can be included in a rapid motion command? 83
About the dog-leg motion… 83
When do you use rapid motion? 84
What is a safe approach distance? 84
G01 – Linear interpolation (straight-line motion) 85
Using G01 for a fast-feed approach 86
A milling example 87
Drill holes with G01? 87
G02 and G03 – Circular interpolation (circular motion) 88
Which positions to program 88
Specifying arc size with the R word 88
The R word is not modal 90
Circular motion with directional vectors (I, J, and K) 91
Arc limitations 92
Full circle in one command 93
Planning your own tool paths 95

Key Concept 4: Know The Compensation Types 97

Lesson 7: Introduction To Compensation 99

What is compensation and why is it needed? 99
More on tolerances 100
The initial setting for compensation 100
When is trial machining required? 100
What happens as tools begin to wear? 101
What do you shoot for? 101
Why do programmers have to know this? 101
Understanding offsets 101
Offset organization 102
Offsets related to cutting tools 102
Offsets related to program zero assignment 103
How offsets are instated 103

Lesson 8: Tool Length Compensation 105

The reasons why tool length compensation is needed 105
No two tools will have exactly the same length 105
A given tool's length will vary from one time it is assembled to the next 106
Tool data is entered separately from the program 106
Sizing and trial machining must often be done 106
What about interference and reach? 106
Programming tool length compensation 106
Choosing the offset number to be used with each tool 106
An example program 107
The setup person's responsibilities with tool length compensation 108
Typical mistakes with tool length compensation 109
Forgetting to instate tool length compensation 109
Forgetting to enter the tool length compensation value 109
Mismatching offsets 109

Lesson 9: Cutter Radius Compensation 113

Will you need to learn this feature? 113
Reasons why cutter radius compensation is required 113
Calculations are simplified for manual programmers 113
Do you have a CAM system? 115
Range of cutter sizes 115
Do you use sharpened (re-ground) cutters? 116
Trial machining and sizing 117

Rough and finish milling with the same set of coordinates 117
Do you have a CAM system? 118
How cutter radius compensation works 118
Steps to programming cutter radius compensation 119
Step one: Instate cutter radius compensation 119
The XY motion to the prior position 119
The Z motion/s to the Z axis work surface 121
The command instating cutter radius compensation that positions the cutting tool to the first surface to mill 121
The offset used with cutter radius compensation 122
The motion to the first work surface 123
Step two: Program the tool path to be machined 124
Step three: Cancel cutter radius com pensation 126
What if I have more than one contour to mill? 127
Examples 128
What if I use a computer aided manufacturing (CAM) system to prepare programs? 130
The setup person's responsibilities with cutter radius compensation 130
Rough and finish milling with the same set of tool path coordinates 131
A warning 132
Lesson 10: Fixture Offsets 137
Do you need to learn about fixture offsets? 137
Assigning multiple program zero points 138
Programming with multiple program zero points 140
The potential trade-off with this method 141
Reminder about tool length compensation values 141
Shifting the point of reference for fixture offset entries 142
Programming fixture offset entries 144
Some other applications for the common fixture offset 145
Allowing for variations in pallet changers 145
Allowing for variations after a mishap 145
Differences in spindle gap from one machine to another 145
To enhance safety during dry-runs 145
Key Concept 5: You Must Provide Structure To Your CNC Programs 147
Lesson 11: Introduction To Program Structure 149
Objectives of your chosen program structure 149
Reasons for structuring programs with a strict and consistent format 149
Familiarization 149
Consistency 150
Re-running tools in the program 150
Efficiency limitations 151
Machine variations that affect program structure 151
M code differences 151
Automatic tool changer variations 152
T word brings a tool to the ready station, M06 commands the tool change 152
Do you have a double-arm tool changer? 152
T word does everything 154
Tool change at beginning or end? 155
Does the machine even have an automatic tool changer? 155
Understanding the G28 command 155
What about G53? 156
A possible problem with initialized modes 157
How to use our given formats 157
Lesson 12: Four Types Of Program Format 159
Format for vertical machining centers 159
A note about documentation 162
Example program for vertical machining centers 163
A few questions about the program: 164
More on the optional stop word (M01) 164
Where is the restart command for each tool? 164
What if my machine doesn't have fixture offsets? 165
Format for horizontal machining centers 167
Key Concept 6: Special Features That Help With Programming 173
Lesson 13: Hole-Machining Canned Cycles 175
Canned cycle commonalities 176
Description of each canned cycle 176
G80 – Cancel the canned cycle mode 176
G81 –Standard drilling cycle 176
G73 – Chip-breaking peck drilling cycle 176
G83 – Deep-hole drilling cycle (full retract between pecks) 176
G84 – Right-hand tapping cycle 177
Feedrate for tapping 178
Tapping can be a little scary 178
Coolant for tapping? 178
When to tap 179
G74 – Left-hand tapping cycle 179
G82 – Counter-boring cycle 179
G89 – Counter-boring cycle for a boring bar 179
G86 – Standard boring cycle (leaves drag line witness mark) 180
Controlling move-over at hole-bottom 180
A tip for boring bar tip pointing 181
G85 – Reaming cycle (most programmers use G81 for reaming) 182
G87 and G88 – Manual cycles (not recommended) 182
Words used in canned cycles 182
A simple example 183
Understanding G98 and G99 185
Canned cycles and the Z axis 187
Extended example showing canned cycle usage 189
Using canned cycles in the incremental positioning mode 193
Lesson 14: Working With Subprograms 199

The difference between main- and sub- programs 199
Loading multiple programs 201
Words used with subprograms 201
Nesting subprograms 202
Machining multiple identical pockets 202
Understanding G52 – temporary shift of program zero 204
Multiple hole-machining operations on a series of holes 204
Do you want to include all of the hole-locations in the subprogram? 206
Rough and finish contour milling 207
Two utility applications for subprograms 208
Control programs 208
A trick to get more fixture offsets 209
What is parametric programming (custom macro B)? 211
Part families 211
User defined canned cycles 211
Utilities 211
Complex motions and shapes 211
Driving accessory devices 212

Lesson 15: Other Special Programming Features 215

Block delete (also called optional block skip) 215
Applications for block delete 216
Another optional stop 216
Trial machining 216
Trial boring 217
Sequence number (N word) techniques) 219
Eliminating sequence numbers 219
Using special sequence numbers in program restart commands 219
Using sequence numbers as statement labels 220
Using block delete to exit a series of commands 220
Using statement labels to change machining order 221
Other G codes of interest 222
Thread milling, G02 & G03 222
G04 - Dwell command 224
G09 and G61 - Exact stop check 225
G10 - Offset setting by programmed command 226
Applications for G10 226
Polar coordinates (G15 and G16) 228
Plane selection commands (G17, G18, and G19) 229
Inch/metric mode selection G20 and G21 230
Secondary reference position, G30 231
Scaling commands, G50 and G51 231
G50.1 and G51.1 - Mirror image commands 231
Applications for mirror image 231
The two ways to activate mirror image 232
Motion relative to zero return position, G53 233
Single direction positioning mode, G60 233
Coordinate rotation G68 and G69 234

Lesson 16: Programming Rotary Devices 239

The difference between an indexer and a rotary axis 239
A note to horizontal machining center programmers 239
Benefits of rotary devices 240
Indexers 240
Programming indexer rotation 240
90 degree and 45 degree indexers 240
Five degree indexers 241
One degree indexer 241
Rotary axes 241
How to program a rotary axis departure 242
Comparison to other axes 242
Zero return position 242
Polarity 243
Designation of program zero 244
Absolute positioning mode 244
Incremental positioning mode 247
Clamping the rotary axis for machining after rotation 249
Rapid and straight line motion 249
Canned cycle usage 250
Approaching rotary device applications 250
Program zero point selection 250
Assigning one program zero point per side 252
Using one central program zero point 252
Example program using rotary device 254

Practice exercises and programming activities 261

Programming Activities 293

Answers to Exercises 333

Answers to programming activities 343

Index 357

 © CNC Concepts, Inc