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
### Table of Contents

- 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 commonalities 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 compensation 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
<table>
<thead>
<tr>
<th>Page</th>
<th>Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>199</td>
<td>The difference between main- and sub- programs</td>
</tr>
<tr>
<td>201</td>
<td>Loading multiple programs</td>
</tr>
<tr>
<td>201</td>
<td>Words used with subprograms</td>
</tr>
<tr>
<td>202</td>
<td>Nesting subprograms</td>
</tr>
<tr>
<td>202</td>
<td>Machining multiple identical pockets</td>
</tr>
<tr>
<td>202</td>
<td>Understanding G52 – temporary shift of program zero</td>
</tr>
<tr>
<td>204</td>
<td>Multiple hole-machining operations on a series of holes</td>
</tr>
<tr>
<td>206</td>
<td>Do you want to include all of the hole-locations in the subprogram?</td>
</tr>
<tr>
<td>207</td>
<td>Rough and finish contour milling</td>
</tr>
<tr>
<td>208</td>
<td>Two utility applications for subprograms</td>
</tr>
<tr>
<td>208</td>
<td>Control programs</td>
</tr>
<tr>
<td>208</td>
<td>A trick to get more fixture offsets</td>
</tr>
<tr>
<td>211</td>
<td>What is parametric programming (custom macro B)?</td>
</tr>
<tr>
<td>211</td>
<td>Part families</td>
</tr>
<tr>
<td>211</td>
<td>User defined canned cycles</td>
</tr>
<tr>
<td>211</td>
<td>Utilities</td>
</tr>
<tr>
<td>211</td>
<td>Complex motions and shapes</td>
</tr>
<tr>
<td>212</td>
<td>Driving accessory devices</td>
</tr>
<tr>
<td>215</td>
<td>Lesson 15: Other Special Programming Features</td>
</tr>
<tr>
<td>215</td>
<td>Block delete (also called optional block skip)</td>
</tr>
<tr>
<td>216</td>
<td>Applications for block delete</td>
</tr>
<tr>
<td>216</td>
<td>Another optional stop</td>
</tr>
<tr>
<td>216</td>
<td>Trial machining</td>
</tr>
<tr>
<td>217</td>
<td>Trial boring</td>
</tr>
<tr>
<td>219</td>
<td>Sequence number (N word) techniques</td>
</tr>
<tr>
<td>219</td>
<td>Eliminating sequence numbers</td>
</tr>
<tr>
<td>219</td>
<td>Using special sequence numbers in program restart commands</td>
</tr>
<tr>
<td>220</td>
<td>Using sequence numbers as statement labels</td>
</tr>
<tr>
<td>220</td>
<td>Using block delete to exit a series of commands</td>
</tr>
<tr>
<td>221</td>
<td>Using statement labels to change machining order</td>
</tr>
<tr>
<td>222</td>
<td>Other G codes of interest</td>
</tr>
<tr>
<td>222</td>
<td>Thread milling, G02 &amp; G03</td>
</tr>
<tr>
<td>224</td>
<td>G04 - Dwell command</td>
</tr>
<tr>
<td>225</td>
<td>G09 and G61 - Exact stop check</td>
</tr>
<tr>
<td>226</td>
<td>G10 - Offset setting by programmed command</td>
</tr>
<tr>
<td>226</td>
<td>Applications for G10</td>
</tr>
<tr>
<td>228</td>
<td>Polar coordinates (G15 and G16)</td>
</tr>
<tr>
<td>229</td>
<td>Plane selection commands (G17, G18, and G19)</td>
</tr>
<tr>
<td>230</td>
<td>Inch/metric mode selection G20 and G21</td>
</tr>
<tr>
<td>231</td>
<td>Secondary reference position, G30</td>
</tr>
<tr>
<td>231</td>
<td>Scaling commands, G50 and G51</td>
</tr>
<tr>
<td>231</td>
<td>G50.1 and G51.1 - Mirror image commands</td>
</tr>
<tr>
<td>231</td>
<td>Applications for mirror image</td>
</tr>
<tr>
<td>232</td>
<td>The two ways to activate mirror image</td>
</tr>
<tr>
<td>233</td>
<td>Motion relative to zero return position, G53</td>
</tr>
<tr>
<td>233</td>
<td>Single direction positioning mode, G60</td>
</tr>
<tr>
<td>234</td>
<td>Coordinate rotation G68 and G69</td>
</tr>
<tr>
<td>239</td>
<td>Lesson 16: Programming Rotary Devices</td>
</tr>
<tr>
<td>239</td>
<td>The difference between an indexer and a rotary axis</td>
</tr>
<tr>
<td>239</td>
<td>A note to horizontal machining center programmers</td>
</tr>
<tr>
<td>240</td>
<td>Benefits of rotary devices</td>
</tr>
<tr>
<td>240</td>
<td>Indexers</td>
</tr>
<tr>
<td>240</td>
<td>Programming indexer rotation</td>
</tr>
<tr>
<td>240</td>
<td>90 degree and 45 degree indexers</td>
</tr>
<tr>
<td>241</td>
<td>Five degree indexers</td>
</tr>
<tr>
<td>241</td>
<td>One degree indexer</td>
</tr>
<tr>
<td>241</td>
<td>Rotary axes</td>
</tr>
<tr>
<td>242</td>
<td>How to program a rotary axis departure</td>
</tr>
<tr>
<td>242</td>
<td>Comparison to other axes</td>
</tr>
<tr>
<td>243</td>
<td>Zero return position</td>
</tr>
<tr>
<td>244</td>
<td>Polarity</td>
</tr>
<tr>
<td>244</td>
<td>Designation of program zero</td>
</tr>
<tr>
<td>244</td>
<td>Absolute positioning mode</td>
</tr>
<tr>
<td>247</td>
<td>Incremental positioning mode</td>
</tr>
<tr>
<td>249</td>
<td>Clamping the rotary axis for machining after rotation</td>
</tr>
<tr>
<td>249</td>
<td>Rapid and straight line motion</td>
</tr>
<tr>
<td>250</td>
<td>Canned cycle usage</td>
</tr>
<tr>
<td>250</td>
<td>Approaching rotary device applications</td>
</tr>
<tr>
<td>250</td>
<td>Program zero point selection</td>
</tr>
<tr>
<td>252</td>
<td>Assigning one program zero point per side</td>
</tr>
<tr>
<td>252</td>
<td>Using one central program zero point</td>
</tr>
<tr>
<td>254</td>
<td>Example program using rotary device</td>
</tr>
<tr>
<td>254</td>
<td>Practice exercises and programming activities</td>
</tr>
<tr>
<td>259</td>
<td>Programming Activities</td>
</tr>
<tr>
<td>333</td>
<td>Answers to Exercises</td>
</tr>
<tr>
<td>343</td>
<td>Answers to programming activities</td>
</tr>
<tr>
<td>357</td>
<td>Index</td>
</tr>
</tbody>
</table>

**Table of Contents**