

Table of Contents

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 16
 - Lesson structure 16
 - 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
- Vertical Machining Centers 21
 - C-frame Style 21
 - Directions of motion (axes) for a C-frame-style vertical machining center 22
 - Axis polarity 22
 - Knee-style Vertical CNC Milling Machines 23
 - Bridge-style Vertical Machining Center (also called gantry-style) 24
- Horizontal Machining Centers 25
 - Directions of Motion (axes) for a Horizontal Machining Center 26
 - Axis polarity 26
- Programmable Functions of a Machining Center 27
 - Spindle 27
 - Spindle speed 27
 - Spindle activation and direction 28
 - Spindle range 28
 - Feedrate 29
 - Coolant 30
 - Automatic Tool Changer 30
 - Measurement System Mode (inch or metric) 31
- What Else Might be Programmable? 31
- Key Points for Lesson 1.1: 32
- Quiz 32

Lesson 1.1a – CNC Certification Cart Machine Configuration 33

- Objectives 33
- CNC Certification Cart Vertical Machining Center 33
 - Directions of motion (axes) for the CNC Certification Cart 34
- Basic Programmable Functions of the CNC Certification Cart 34
 - Spindle 34
 - Spindle speed 34
 - Spindle activation and direction 34
 - Spindle range 34
 - Feedrate 34
 - Coolant 35
 - Automatic Tool Changer 35
 - Measurement System Mode (inch or metric) 36
- Key Points for Lesson 1.1a: 36
- Quiz 36

Lesson 1.2 – CNC Job Workflow 37

- Objectives 37
- Companies that Use CNC Machining Centers 37
 - What Will You be Doing? 38
- The CNC Job Workflow 39
 - Study the Workpiece Drawing 39
 - Decide which CNC Machining Center to Use 39
 - Determine the Sequence of Operations and Tooling 39
 - Order and Check Tooling 39
 - Create the Part Program 39
 - Create Setup and Production Documentation 39
 - Load the Program into the CNC's Memory 40
 - Setup the Machine 40
 - Verify the Program 40
 - First Part Inspection 40
 - Production 40
 - Store the Corrected Version of the Program 40
- Key points for lesson 1.2: 40

Lesson 1.3 – Visualizing the Execution of a Program 43

- Objectives 43
- Introduction 43
- Program Structure 44
- Order of Program Execution 44
 - An Example of Program Execution 44
 - Manual milling machine procedure: 45
 - CNC program: 45
- Sequence Numbers 46
- Decimal Point Programming 46
- Modal Words 46
- Initialized Words 47
 - Word Order in a Block 47
- Common Beginner Mistakes 47

Omitting Decimal Points 47	Step 6: Touch off the workpiece edge in Y 70
Decimal point tip 47	Step 7: Set the relative position for the Y-axis to zero 71
Other Mistakes of Omission 48	Step 8: Back off in Z, move over -0.100 inches in Y 71
Letter O or Number Zero? 48	Step 9: Reset the Y-axis relative position again 72
Key points for lesson 1.3: 48	Step 10: Send the X and Y axes to the zero return position 72
Quiz 48	Measuring the X and Y Axes for a Circular-Shaped Workpiece 73
Lesson 1.4 – Understanding the Workpiece Coordinate System 49	Step 1: Load the workpiece and dial indicator 73
Objectives 49	Step 2: Indicate the center of the workpiece. 73
Introduction 49	Step 3: Zero the X and Y relative positions 73 73
A Graph Analogy 49	Step 4: Send the X and Y axes to the zero return position 74
What About the Z-Axis? 51	Measuring the Z-axis 74
Understanding Polarity 51	Step 1: Make the workholding setup and load a workpiece 74
Wisely Choosing the Workpiece Coordinate System Zero Location 54	Step 2: Touch the spindle nose to the gauge block 75
Selecting the X and Y Workpiece Coordinate System Zero 54	Step 3: Preset the Z-axis relative position to the height of the gauge block 75
A reminder about axis movement 55	Step 4: Send the Z-axis to the zero return position 76
Selecting the Z-Axis Workpiece Coordinate System Zero 56	Measuring the Workpiece Coordinate System Offsets with a Probe 76
Absolute Versus Incremental Modes 56 58	How the Program Zero Point Selection Affects the Setup Person 77
A Decimal Point Reminder 58	Key points for Lesson 1.5: 78
Key points for lesson 1.4: 58	Quiz 78
Quiz 59	Lesson 1.6–Setting Workpiece Coordinate System Offsets 79
Lesson 1.5 – Determining Workpiece Coordinate System Offsets 61	Objectives 79
Objectives 61	Introduction 79
Introduction 61	Understanding Workpiece Coordinate System Offsets 79
Determining the Workpiece Coordinate System Offset Values 61	A Full Example of Using the Workpiece Coordinate System 81
What is Machine Zero Position? 63	The Program 81
How to Determine the Workpiece Coordinate System Offset Values 63	What’s the G91 doing in block N090? 82
How Predictable is the Workholding? 63	The Setup 82
What is a Qualified Workholding Setup? 64	What is the External/Common Work Coordinate System Offset? 84
Machine, Workpiece, and Relative Coordinate Systems 64	What if My Machining Center Doesn’t Have Workpiece Coordinate Offsets? 84
Using the Relative Coordinate System and an Edge Finder for Measurement 65	Advantages of Workpiece Coordinate System Offsets versus G92 84
Manually Measuring the Workpiece Coordinate System Offset Values 67	Safety 84
Measuring the X and Y Axes for a Rectangular-Shaped Workpiece 68	Ease of use 84
Step 1: Load workpiece and edge finder 68	Efficiency 84
Step 2: Touch off the workpiece edge in X 68	
Step 3: Set the relative position for the X-axis to zero 69	
Step 4: Back off in Z, move over 0.100 inches in X 69	
Step 5: Zero the X-axis relative position again 70	

<ul style="list-style-type: none"> Reworking parts 84 Assigning Program Zero in the Program with G92 85 Key points for lesson 1.6: 85 Lesson 1.7—Introduction to Programming Words 87 <ul style="list-style-type: none"> Objectives 87 Introduction 87 <ul style="list-style-type: none"> Decimal Points in Words 88 O-Word 88 N-Word 88 G-Word 88 X-Word 89 Y-Word 89 Z-Word 89 A-Word 89 B-Word 89 C-Word 89 <ul style="list-style-type: none"> Note about rotary axis designators and indexer activators 90 R-Word 90 I, J, K-Words 90 Q-Word 90 P-Word 90 L-Word 90 F-Word 90 M-Word 91 S-Word 91 T-Word 91 H-Word 91 D-Word 91 EOB (end of block character) 91 / (slash code) 92 G-codes and M-codes 92 <ul style="list-style-type: none"> G-codes 92 <ul style="list-style-type: none"> G-code limitations: 92 Optional G-codes 92 What does initialized mean? 92 What does modal mean? 92 Commonly used G-codes 93 0i-MD G-codes 93 M-codes 95 <ul style="list-style-type: none"> Common M-codes used on a CNC machining center 95 Key points for lesson 1.7: 96 YOU MUST PREPARE TO WRITE PROGRAMS 97 <ul style="list-style-type: none"> Preparation and Time 97 Preparation and Safety 98 Typical Mistakes 99 <ul style="list-style-type: none"> Syntax Mistakes 99 Motion Mistakes 99 	<ul style="list-style-type: none"> Mistakes of Omission 99 Process Mistakes 99 Lesson 2.1—Preparation for Programming 101 <ul style="list-style-type: none"> Objectives 101 Introduction 101 Plan the Sequence of Machining Operations 101 Develop the Cutting Conditions 103 <ul style="list-style-type: none"> An example 104 Cutting Conditions can be Subjective 105 Do the Math and Mark-up the Print 106 <ul style="list-style-type: none"> Marking up the Workpiece Engineering Drawing 108 Doing the Math 109 What About Contour Milling Operations? 110 <ul style="list-style-type: none"> Planning the milling cutter’s centerline tool path 111 Check the Required Tooling 111 Plan the Workholding Set-up 112 Other Documentation Needed for the Job 113 <ul style="list-style-type: none"> Production Run Documentation 113 Program Listing 114 Is it all Worth It? 114 Key points for lesson 2.1: 114 UNDERSTANDING MOTION TYPES 117 <ul style="list-style-type: none"> What is Interpolation? 117 Lesson 3.1—Programming the Three Basic Motion Types 121 <ul style="list-style-type: none"> Objectives 121 Introduction 121 <ul style="list-style-type: none"> Motion Type Commonalities 121 Understanding the Point Programmed for each Cutting Tool 122 <ul style="list-style-type: none"> Hole-making Tools 122 <ul style="list-style-type: none"> Center drill 122 Spot drill (not shown above) 122 Drill 122 Reamer 122 Tap 122 Boring bar 123 Milling Cutting Tools 123 G00 Rapid Motion (also called positioning) 124 <ul style="list-style-type: none"> How Many Axes can be Included in a Rapid Motion Command? 125 About Dog-Leg Motion... 126 When do you Use Rapid Motion? 126 What is a Safe Approach Distance? 126 G01 Linear Interpolation (straight-line motion) 128 <ul style="list-style-type: none"> Using G01 for a Fast-Feed Approach 130 A Milling Example 130
--	---

<ul style="list-style-type: none"> Drill Holes with G01? 131 G02 and G03–Circular Interpolation (circular motion) 132 <ul style="list-style-type: none"> Which Positions to Program 132 Specifying Arc Size with the R-word 132 <ul style="list-style-type: none"> The R-word is not modal 134 More About Clockwise versus Counterclockwise Circular Interpolation 135 Specifying Circular Interpolation with Directional Vectors (I, J, and K) 135 Arc Limitations 137 <ul style="list-style-type: none"> Simultaneously specifying R with I, J or K 137 Specifying a semicircle 137 Specifying arcs greater than 180° using the R-word 137 Programming a Full Circle 137 <ul style="list-style-type: none"> Example using I, J and K 137 Example using R-word 138 Planning Your Own Tool Paths 139 Key Points for Lesson 3.1: 139 Exercise: 140 	<ul style="list-style-type: none"> Sizing and Trial Machining Must Often be Done 154 <ul style="list-style-type: none"> What about interference and reach? 154 Programming Tool Length Compensation 154 <ul style="list-style-type: none"> Choosing the offset number to be used with each tool 155 An example program 155 The Setup Person’s Responsibilities with Tool Length Compensation 156 <ul style="list-style-type: none"> Using the Tool’s Length as the Tool Length Compensation (offset) Value 157 <ul style="list-style-type: none"> Determining tool length compensation values 157 Step 1: Make the workholding setup—or place some kind of flat block on the machine table 158 Step 2: Place a gauge block on the flat surface 158 Step 3: Without a tool in the spindle, make the spindle nose touch the block or offset gauge. 158 Step 4: Retract the Z-axis and load a cutting tool to be measured. Bring the tool tip to the same block. 158 Step 5: Repeat step four for all cutting tools that must be measured. 159 Measuring tool lengths off line—with a tool length measuring gauge 159 Entering tool length compensation offsets 160 Using the Distance from the Tool Tip to Workpiece Coordinate System Zero as the Tool Length Compensation (offset) Value 160 <ul style="list-style-type: none"> Measuring tool length compensation values with this second method 161 Step 1: Make the workholding setup and load a workpiece 161 Step 2: Send the machine to the Z-axis zero return position and reset (set to zero) the Z-axis relative display. 161 Step 3: Load a tool to be measured and manually bring the tip of the tool to the Z-axis program zero surface. 162 Reasons for Using the Recommended Method 162 Typical Mistakes with Tool Length Compensation 162 <ul style="list-style-type: none"> Forgetting to instate tool length compensation 162 Forgetting to enter the tool length compensation value 163 Mismatching offsets 163 What if a Machine Doesn’t have Workpiece Coordinate System Offsets? 163 Trial Machining with Tool Length Compensation 163
--	--

KNOW THE COMPENSATION TYPES 143

Lesson 4.1—Introduction to Compensation 145

- Objectives 145
- What are Compensations and Why are they Needed? 145
- More on Tolerances 146
 - The Initial Setting for Compensation 146
 - When is Trial Machining Required? 146
 - What Happens as Tools Begin to Wear? 147
 - What do You Shoot For? 147
 - Why do Programmers have to know about Compensations? 147
- Understanding Offsets 147
 - Offset Organization 148
 - Offsets Related to Cutting Tools 148
 - Offsets Related to Program Zero 149
 - How are Offsets Instated? 150
- Key points for Lesson 4.1: 150

Lesson 4.2—Tool Length Compensation 153

- Objectives 153
- Introduction 153
- The Reasons Why Tool Length Compensation is Needed 153
 - No Two Tools Will Have Exactly the Same Length 153
 - Tool’s Length Will Vary from Each Time it is Assembled 154
 - Tool Data is Entered Separately from the Program 154

When trial machining is not required 164
 Sizing with Tool Length Compensation 164
 A tip for remembering which way to adjust the offset 164
 What if I use the second method shown for tool length compensation? 164
 Do I have to make all these calculations when adjusting offsets? 164
 Why can't I just change the Z coordinate/s in the program to make sizing adjustments? 164
 Key points for lesson 4.2: 165

Lesson 4.3—Cutter Radius Compensation 169

Objectives 169
 Introduction 169
 Will You Need to Learn this Feature? 169
 Reasons Why Cutter Radius Compensation is Required 169
 Calculations are Simplified for Manual Programmers 169
 Range of Cutter Sizes 171
 Do you use re-sharpened (re-ground) cutters? 172
 Trial Machining and Sizing 173
 Rough and Finish Milling with the Same Coordinates 173
 Do you have a CAM system? 174
 How Cutter Radius Compensation Works 174
 Steps to Programming Cutter Radius Compensation 174
 Step 1: Instate Cutter Radius Compensation 175
 The XY motion to the prior position 175
 The Z motions to the Z-axis work surface 176
 Instate cutter radius compensation and positions the cutting tool to the first surface to mill 177
 G41 or G42? 177
 The offset used with cutter radius compensation 178
 The motion to the first work surface 179
 Step 2: Program the Tool Path to be Machined 180
 Step 3: Cancel Cutter Radius Compensation 181
 Don't forget to cancel! 182
 What if I have more than one contour to mill? 183
 Examples 183
 Example 1 183
 Example 2 184
 What if I use a CAM System to Prepare Programs? 185
 Setup Person's Cutter Radius Compensation Responsibilities 185
 Rough and Finish Milling with the Same Set of Tool Path Coordinates 186

A warning 187
 Trial Machining with Cutter Radius Compensation 188
 189
 What if the milling cutter machines on both sides of the workpiece? 189
 A tip for remembering which way to adjust the offset 189
 How important is it that you make your first workpiece a good one? 189
 When trial machining is not required 190
 Sizing with Cutter Radius Compensation 190
 What if I use a CAM system and the offset value is the deviation from the planned cutter size? 190
 Do I have to make all these calculations when adjusting offsets? 190
 Why can't I just change programmed coordinate/s in the program to make sizing adjustments? 190
 Key points for lesson 4.3: 191

Lesson 4.4—Workpiece Coordinate System Offsets 195

Objectives 195
 Introduction 195
 Do you Need to Learn More about Workpiece Coordinate System Offsets? 195
 Assigning Multiple Program Zero Points 196
 Programming with Multiple Program Zero Points 198
 The potential trade-off with this method 200
 Reminder about tool length compensation values 200
 Shifting the Workpiece Coordinate System Offsets Reference Point 200
 Programming Workpiece coordinate system setting offset Values 203
 Some other Applications for the External Work Offset 203
 Allowing for Variations in Pallet Changers 203
 Allowing for Variations after a Mishap 204
 Differences in Spindle Gap from One Machine to Another 204
 To enhance safety during dry-runs 204
 Key points for lesson 4.4: 204

YOU MUST PROVIDE STRUCTURE TO YOUR CNC PROGRAMS 205

Lesson 5.1—Introduction to Program Structure 207

Objectives 207
 Introduction 207
 Objectives of Your Chosen Program Structure 207

Reasons for Structuring Programs with Consistent Format 208	More on the optional stop word (M01) 223
Familiarization 208	Where is the restart command for each tool? 223
Consistency 208	Format for Horizontal Machining Centers 224
Re-running Tools in the Program 208	Program-Startup Structure: 224
Efficiency Limitations 209	Tool-Startup Structure: 224
Machine Variations that Affect Program Structure 209	Tool-End Structure: 224
M-code Differences 209	Program-End Structure: 224
Automatic Tool Changer Variations 210	Key points for lesson 5.2: 226
T-word brings a tool to the ready station, M06 commands the tool change 210	
Do you have a double-arm tool changer? 210	
Where is the tool change position? 211	
What is the M19 doing in block N100? 211	
Why is the T-word repeated in each M06 command? 212	
Do you have a single-arm or umbrella tool changer? 212	
T-word does everything 212	
Tool change at beginning or end? 213	
Does the machine even have an automatic tool changer? 213	
Understanding the G28 Command 214	
What about G53? 215	
A Possible Problem with Initialized Modes 215	
Key points for lesson 5.1: 216	
Lesson 5.2–Structured Program Format 217	
Objectives 217	
Introduction 217	
Structured Program Format for Vertical Machining Centers 218	
Program Start-Up Structure: 218	
Tool Start-Up Structure: 218	
Tool Ending Structure: 218	
Program Ending Structure: 218	
N010 219	
N020 219	
N030 219	
N100 219	
N110 219	
N120 219	
N130 219	
N140 219	
N150 219	
N200 220	
N210 220	
N220 220	
220	
N300 - N330 220	
A Note About Documentation Comments 220	
Example Program for Vertical Machining Centers 221	
A few questions about the program: 223	
	FEATURES THAT HELP SIMPLIFY PROGRAMMING 229
	Lesson 6.1–Canned Cycles For Drilling 231
	Objectives 231
	Introduction 231
	What Does “Canned” Mean? 231
	Canned Cycle Commonalities 232
	Description of each canned cycle 232
	G80 – Cancel the canned cycle mode 232
	G81 – Drilling cycle 232
	G73 – High-speed peck drilling cycle 232
	G83 – Peck drilling cycle (full retract between pecks) 233
	G84 – Right-hand tapping cycle (also used for rigid tapping) 234
	Rigid tapping 234
	Feedrate for tapping 234
	Rapid plane for tapping 234
	Tapping can be a little scary 234
	Coolant for tapping? 235
	When to tap 235
	G74 – Left-hand tapping cycle 235
	G82 – Counter-boring cycle 236
	G89 – Boring cycle with dwell 236
	G86 – Boring cycle (leaves drag line witness mark) 236
	G76 – Fine boring cycle (leaves no witness mark) 236
	Controlling move-over at hole-bottom 237
	A tip for boring bar tip pointing 237
	G85 – Boring cycle (Feeds in and feeds out) 238
	G87 and G88 – Manual cycles (not recommended) 238
	Words used in canned cycles 238
	A simple example 239
	Understanding G98 and G99 240
	Canned cycles and the Z-axis 242
	Extended example showing canned cycle usage 244
	N020, N105, N175 247
	N030, N115, N185, N255, N300, N345, N395 247
	N245, N290, N335 247

N255, N300, N345 247	Using sequence numbers as statement labels 272
N045, N050, N125, N130, N195, N200 247	Using block delete to exit a series of commands 272
N075, N145, N215, N260, N305, N355, N405 247	Using statement labels to change machining order 273
N255 247	Other G-codes of interest 274
N300 247	Thread milling, G02 & G03 274
N183 248	G04 - Dwell command 277
N185 248	G09 and G61 - Exact stop check 278
Using canned cycles in the incremental positioning mode 248	G10 - Offset setting by programmed command 279
Key points for Lesson 6.1: 250	Applications for G10 280
Lesson 6.2–Subprogram Techniques 253	Programming tool length and cutter radius compensation offset values 280
Objectives 253	Handling discrepancies among pallets 281
Introduction 253	Polar coordinates (G15 and G16) 282
The Difference Between a Main Program and a Subprogram 253	Plane selection commands (G17, G18, and G19) 282
Loading the Main Program and Subprograms 255	Inch/metric mode selection G20 and G21 283
Words used with Subprograms 255	Secondary reference position, G30 284
Nesting Subprograms 256	Scaling commands, G50 and G51 284
Machining Multiple Identical Pockets 256	G50.1 and G51.1 - Mirror image commands 284
Understanding G52–Local Coordinate System 257	Applications for mirror image 284
Multiple Hole-machining Operations on a Series of Holes 258	The two ways to activate mirror image 285
Want to include all of the hole-locations in the subprogram? 259	Manually activating mirror image 285
Rough and Finish Side milling 261	Activating mirror image from within a program 285
Utility Applications for Subprograms 262	Motion relative to zero return position, G53 286
Control Programs 262	Single direction positioning mode, G60 286
A Trick to get More Workpiece Offsets 263	Coordinate rotation G68 and G69 287
What is Parametric Programming (Custom Macro)? 265	Adjusting for workholding devices that are not square with the axes 289
Part Families 265	Key points for lesson 6.3: 290
User Defined Canned Cycles 265	
Utilities 265	Lesson 6.4–Programming Rotary Devices 291
Complex Motions and Shapes 266	Objectives 291
Driving Accessory Devices 266	Introduction 291
Key points for Lesson 6.2: 266	The difference between an indexer and a rotary axis 291
	A note to horizontal machining center programmers 291
Lesson 6.3–Other Special Programming Features 267	Benefits of rotary devices 292
Objectives 267	Indexers 292
Introduction 267	Programming indexer rotation 292
Block delete (also called optional block skip) 267	90 degree and 45 degree indexers 292
Applications for block delete 268	Five degree indexers 293
Another optional stop 268	One degree indexer 293
Trial machining 269	Rotary axes 293
Trial boring 269	How to program a rotary axis departure 294
A warning about block delete applications 271	Comparison to other axes 294
Sequence number (N-word) techniques) 271	Zero return position 294
Eliminating sequence numbers 271	Polarity 295
Using special sequence numbers in program restart commands 271	Designation of program zero 295
	Absolute positioning mode 296
	Incremental positioning mode 299

- Clamping the rotary axis for machining after rotation 301
- Rapid and straight line motion 301
- Canned cycle usage 302
- Approaching rotary device applications 302
 - Program zero point selection 302
 - Assigning one program zero point per side 304
 - Using one central program zero point 304
 - Calculating coordinates when center of rotation is the program zero point 305
- Example program using rotary device 306
- Key points for lesson 6.4: 313

KNOW YOUR MACHINE FROM AN OPERATOR’S POINT OF VIEW 315

- Are you only interested in setup and operation? 315
 - From Lesson 1.1: Machine configurations 315
 - From Lesson 1.2: CNC job vlow 315
 - From Lesson 1.4: Understanding the workpiece coordinate system 315
 - From Lesson 1.5: Determining workpiece coordinate system values 315
 - From Lesson 1.6: Setting workpiece coordinate system offset values 316
 - From Lesson 4.1: Introduction to compensation 316
 - From Lesson 4.2: Tool length compensation 316
 - From Lesson 4.3: Cutter radius compensation 316
 - From Lesson 4.4: Workpiece coordinate system setting offsets 316
 - From Lesson 5.1: Introduction to program structure 316
- The need for hands-on experience 316
- Key Concept Number Seven 317
 - More on axis polarity 317
- Procedures you must know about 319
 - Machine power-up 319
 - Sending the machine to its zero return position 319
 - Manually moving each axis 319
 - An example of manual axis movement 319
 - Manually starting the spindle 320
 - Manually making tool changes 320
 - Manipulating the display screen 320
 - The position display screen 320
 - The program display screen 321
 - The offset/setting display screen 321
 - The program check display screen 321
 - Loading programs 321

Lesson 7.1—Tasks Related to Setup and Running Production 323
Objectives 323

- Introduction 323
 - Setup-related tasks 323
 - Running production-related tasks 323
- CNC job from start to finish 325
- Setup documentation 326
 - Tear down the previous setup and put everything away 327
 - Gather the components needed to make the setup 327
 - Make the workholding setup 327
 - Assign the program zero point 328
 - Measure program zero assignment values 328
 - Enter program zero assignment values into workpiece coordinate system setting offset number one 329
 - Assemble the cutting tools needed for the job 330
 - Measure tool length- and cutter-radius-compensation values 331
 - The static nature of cutting tool measurements 331
 - Enter tool length and cutter radius compensation values into offsets 332
 - Load cutting tools into the machine’s automatic tool changer magazine 332
 - Load the CNC program 333
 - The physical tasks related to setup are now completed 333
 - Verify the correctness of a new or modified program 333
 - Verify the correctness of the setup 334
 - A tip that will save a crash some day 335
 - Dry running our example program 335
 - Cautiously run the first workpiece 336
 - The most dangerous time 336
 - Making sure the first workpiece is a good one 337
 - Machining the first workpiece in our example job 338
 - If necessary, optimize the program for better efficiency 339
 - If changes have been made to the program, save the corrected version of the program 339
- Production run documentation 340
 - A note to programmers: 340
 - Remove the previous workpiece 341
 - Load the next workpiece 341
 - Activate the cycle 341
 - Monitor the cycle 342
 - Clean and de-burr the workpiece 342
 - Perform specified measurements 342
 - Make offset adjustments to maintain size for critical dimensions (sizing) 343
 - Replace worn tools 344
 - Clean the machine 344
 - Preventive maintenance 344

Key points for lesson 7.1: 344

Lesson 7.2—Operation Panels 347

- Objectives 347
- Introduction 347
- The two most important operation panels 347
- The control panel buttons and switches 348
 - Display screen mode keys 349
 - Position display pages 351
 - Program display pages 351
 - Offset display pages 352
 - Graph display pages 353
 - Other display screen modes 353
 - The keyboard 353
 - Letter Keys 354
 - The slash key (/) 354
 - Number keys 354
 - Decimal point key 354
 - CAN key 354
 - EOB key 354
 - The input key 354
 - Cursor control keys 354
 - Program Editing Keys 354
 - Reset key 354
- The machine panel 355
 - Power buttons 356
 - MODE switch 356
 - CYCLE START button 356
 - FEED HOLD button 356
 - FEEDRATE OVERRIDE switch 356
 - RAPID OVERRIDE switch 357
 - EMERGENCY STOP button 357
 - Conditional switches 357
 - DRY RUN on/off switch 357
 - SINGLE BLOCK on/off switch 357
 - BLOCK DELETE on/off switch (also called optional block skip) 357
 - OPTIONAL STOP on/off switch 357
 - Buttons and switches for manual functions 358
 - Axis jogging controls 358
 - Handwheel controls 358
 - Spindle control 358
 - Automatic tool changer control 358
 - Indicator lights and meters 359
 - Spindle rpm and horsepower meters 359
 - Axis drive-motor horsepower meter 359
 - Cycle indicator lights 359
 - Zero return position indicator lights 359
 - Optional stop indicator light 359
 - Other buttons and switches on the machine panel 359
- Other operation panels on your machining center 359
- Key points for lesson 7.2: 360

KNOW THE BASIC MODES OF OPERATION 361

Lesson 8.1—Operation Modes 363

- Objectives 363
- Introduction 363
- The manual mode switch positions 363
- The manual data input (MDI) mode switch position 364
 - Commanding an MDI zero return 364
 - The complete procedure to give an MDI command 364
 - Commanding a tool change 365
 - Commanding spindle activation with MDI 365
 - Other times when MDI is used 365
 - Can you make motion commands with MDI? 365
- The edit mode switch position 366
 - To make a program in memory the active program (to call up a program) 366
 - To enter a new program 367
- The program operation mode 367
 - To run the active program from the beginning 367
- Key points for lesson 8.1: 368

THE IMPORTANCE OF PROCEDURES 369

Lesson 9.1—The Key Operation Procedures 371

- Objectives 371
- Introduction 371
- Develop your own operation handbooks 371
 - Manual procedures: 371
 - MDI Procedures 371
 - Program Manipulation Procedures 372
 - Setup Procedures 372
 - Program Running Procedures 372
 - Blank procedure form 372
- Sample operation handbook (Levil certification cart - mill) 374
 - Manual Procedures 374
 - To start the machine 374
 - To do a zero return 374
 - To start spindle 374
 - To jog axes (using continuous jog) 374
 - To jog axes (using incremental jog) 374
 - To use the handwheel 374
 - To set axis displays 375
 - To enter tool offsets (GEOM(H), WEAR(H), GEOM(D), WEAR(D) 375
 - To enter workpiece coordinate system offsets 375
 - MDI Procedures 376
 - To execute an MDI command 376
 - Program Manipulation Procedures 376

To get ready to edit programs 376
To show a directory of programs 376
To call up a program from with the CNC memory (make it the active program) 376
To load programs 376
To delete programs 377
To search within a program 377
To alter, insert, & delete 377
To save programs 378
To use background edit 378
Setup Procedures 378
To mount the workholding device on the table 378
To measure the program zero assignment values in the X and Y axes 378
To measure the program zero assignment values in the Z-axis 379
To measure and enter tool length compensation values 379
Program Running Procedures 379
To run the program (in normal production - no verification techniques) 379
To rerun tools 380
To do a free flowing dry run 380
To do an normal air cutting run 380
To run the first workpiece 380

To cancel a cycle 381
To clear an alarm 381
Key points for lesson 9.1: 381

KNOW HOW TO SAFELY VERIFY PROGRAMS

383

Safety priorities 383
Operator safety 383
Machine tool safety 384
Workpiece safety 384

Lesson 10.1–Program Verification 385

Objectives 385
Introduction 385
Two more procedures 385
Canceling the CNC cycle 385
To re-run a tool 386
Verifying a job that contains mistakes 388
391
Loading the program 391
The dry run to check for setup mistakes 392
Cautiously running the first workpiece 393
Index 395