# CONTENTS

## CHAPTER 1  *Introduction to Computer Numerical Control Manufacturing*

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1–1  Chapter Objectives</td>
<td>1</td>
</tr>
<tr>
<td>1–2  Introduction</td>
<td>1</td>
</tr>
<tr>
<td>1–3  Numerical Control Definition, Its Concepts and Advantages</td>
<td>1</td>
</tr>
<tr>
<td>1–4  Definition of Computer Numerical Control and Its Components</td>
<td>2</td>
</tr>
<tr>
<td>1–5  Advantages of CNC Compared with NC</td>
<td>4</td>
</tr>
<tr>
<td>1–6  Special Requirements for Utilizing CNC</td>
<td>4</td>
</tr>
<tr>
<td>1–7  Financial Rewards of CNC Investment</td>
<td>5</td>
</tr>
<tr>
<td>1–8  CNC Machining Centers and Turning Centers</td>
<td>6</td>
</tr>
<tr>
<td>1–9  Other Types of CNC Equipment</td>
<td>8</td>
</tr>
<tr>
<td>1–10 CNC Input and Storage Media</td>
<td>8</td>
</tr>
<tr>
<td>1–11 Chapter Summary</td>
<td>11</td>
</tr>
<tr>
<td>Review Exercises</td>
<td></td>
</tr>
</tbody>
</table>

## CHAPTER 2  *Modern Machine Tool Controls*

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2–1  Chapter Objectives</td>
<td>13</td>
</tr>
<tr>
<td>2–2  Introduction</td>
<td>13</td>
</tr>
<tr>
<td>2–3  Different Types of System Control</td>
<td>13</td>
</tr>
<tr>
<td>2–4  Loop Systems for Controlling Tool Movement</td>
<td>16</td>
</tr>
<tr>
<td>2–5  Establishing Locations via Cartesian Coordinates</td>
<td>19</td>
</tr>
<tr>
<td>2–6  CNC Machine Axes of Motion</td>
<td>20</td>
</tr>
<tr>
<td>2–7  Types of Tool Positioning Modes</td>
<td>24</td>
</tr>
<tr>
<td>2–8  Units Used for Positioning Coordinates</td>
<td>25</td>
</tr>
<tr>
<td>2–9  Chapter Summary</td>
<td>25</td>
</tr>
<tr>
<td>Review Exercises</td>
<td></td>
</tr>
</tbody>
</table>

## CHAPTER 3  *Tooling for Hole and Milling Operations*

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3–1  Chapter Objectives</td>
<td>32</td>
</tr>
<tr>
<td>3–2  Introduction</td>
<td>32</td>
</tr>
<tr>
<td>3–3  Tooling for Drilling Operations</td>
<td>32</td>
</tr>
</tbody>
</table>
CHAPTER 4  Exploring Features of CNC Machining Centers 56

4–1 Chapter Objectives 56
4–2 Introduction 56
4–3 Background on CNC Machining Centers 56
4–4 Tooling Systems Used with Automatic Tool Changers 60
4–5 Methods of Securing Tools in Tool Holders 62
4–6 Methods of Securing Tooling Systems to the CNC Spindle 64
4–7 Automatic Tool Changer Systems 65
4–8 Pallet Loading Systems 67
4–9 Features of the Machine Control Unit (MCU) Machining Centers 70
4–10 Chapter Summary 77
        Review Exercises

CHAPTER 5  Review of Basic Blueprint Reading for CNC Programmers 80

5–1 Chapter Objectives 80
5–2 Introduction 80
5–3 Sheet Sizes 80
5–4 Drawing Formats 81
5–5 Interpreting Lines in Drawings 87
5–6 Projection Conventions Used in Drawings 91
5–7 Visualizing 3D Objects from 2D Orthographic Views 93
5–8 Auxiliary Views 93
5–9 Sectional Views 93
5–10 Reading Dimensions 93
5–11 Reading Threads and Thread Notes 115
5–12 Reading Surface Finish Symbols and Notes 125
5–13 Chapter Summary 131
        Review Exercises
        Bibliography
9-3 Essential CNC Shop Activities 182
9-4 Part Drawing Study 182
9-5 Methodizing of Operations for CNC Machining Centers 185
9-6 Deciding on a CNC Machine 185
9-7 Methods of Holding the Part During Machining 186
9-8 Machining Determination 193
9-9 Cutting Conditions 194
9-10 Writing a Programming Manuscript 194
9-11 Inputting Programs to the Machine Control Unit 194
9-12 Setup Procedure 195
9-13 Debugging and Verifying the Program 200
9-14 Part Production 201
9-15 Chapter Summary 201

Review Exercises

CHAPTER 10  Word Address Programming 203

10-1 Chapter Objectives 203
10-2 Introduction 203
10-3 Programming Language Format 203
10-4 Programming Language Terminology 204
10-5 Arrangement of Addresses in a Block 205
10-6 Program and Sequence Numbers (O, N Codes) 206
10-7 Preparatory Functions (G Codes) 207
10-8 Dimension Words (X, Y, Z... Codes) 209
10-9 Feed Rate (F Code) 210
10-10 Spindle Speed (S Code) 210
10-11 Miscellaneous Machine Functions (M Codes) 211
10-12 Automatic Tool Changing (M6 Code) 211
10-13 Tool Length Offset and Cutter Radius Compensation (H, D Codes) 212
10-14 Comments 212
10-15 Chapter Summary 213

Review Exercises

CHAPTER 11  Running the Mill Simulator 214

11-1 Chapter Objectives 214
11-2 Introduction 214
11-3 Predator Simulation Software 214
11-4 System Requirements 215
11-5 Conventions Used in This Chapter 215
11-6 Installation 215
11-7 Running a Mill Simulation Job Stored on CD 219
11-8 Using the Virtual CNC to Inspect the Machined Part 221
### CHAPTER 12 Programming Hole Operations 226

- 12-1 Chapter Objectives 226
- 12-2 Introduction 226
- 12-3 Fixed or Canned Cycles 226
- 12-4 Hole Operation Commands 228
- 12-5 Writing a Hole Operation Program 233
- 12-6 Chapter Summary 240

Review Exercises

### CHAPTER 13 Programming Linear Profiles 247

- 13-1 Chapter Objectives 247
- 13-2 Introduction 247
- 13-3 Linear Interpolation Commands 247
- 13-4 Writing a Linear Profiling Program 247
- 13-5 Determining Cutter Offsets for Inclined Line Profiles 253
- 13-6 Chapter Summary 268

Review Exercises

### CHAPTER 14 Programming Circular Profiles 275

- 14-1 Chapter Objectives 275
- 14-2 Introduction 275
- 14-3 Specifying the Plane for Circular Arc Interpolation 275
- 14-4 Circular Interpolation Commands 275
- 14-5 Circular Interpolation via Direct Radius Specification 281
- 14-6 Profiling at Constant Feed Rate 283
- 14-7 Determining Cutter Offsets for Line-Arc Profiles 284
- 14-8 Chapter Summary 297

Review Exercises

### CHAPTER 15 Programming with Cutter Diameter Compensation 308

- 15-1 Chapter Objectives 308
- 15-2 Introduction 308
- 15-3 Cutter Diameter Compensation 308
- 15-4 Advantages of Using Cutter Diameter Compensation 310
- 15-5 Some Restrictions with Cutter Diameter Compensation 310
- 15-6 Cutter Diameter Compensation Commands 312
- 15-7 Cutter Diameter Compensation with Z-Axis Movement 319
- 15-8 Cutter Diameter Compensation Interruptions 320
- 15-9 Chapter Summary 327

Review Exercises
CHAPTER 16  Programming with Subprograms

16–1 Chapter Objectives 333
16–2 Introduction 333
16–3 Subprogram Concept 333
16–4 Fanuc Commands for Calling a Subprogram and Returning to the Main Program 334
16–5 Additional Subprogram Control Features 337
16–6 General Rules for Subprogramming 338
16–7 Chapter Summary 355
Review Exercises

CHAPTER 17  Introduction to the CNC Lathe

17–1 Chapter Objectives 362
17–2 Introduction 362
17–3 Background on the CNC Lathe 362
17–4 CNC Lathe Axes of Motion 367
17–5 Features of the Machine Control Unit (MCU) Lathes 367
17–6 Basic Lathe Operations 374
17–7 Tooling for CNC Lathe Operations 377
17–8 Tool Speeds, Feeds, and Depth of Cut for Lathe Operations 381
17–9 Feed Directions and Rake Angles for Lathe Operations 383
17–10 Chapter Summary 385
Review Exercises

CHAPTER 18  Running the Lathe Simulator

18–1 Chapter Objectives 388
18–2 Introduction 388
18–3 Running a Lathe Simulation Job Stored on CD 388
18–4 Using the Virtual CNC to Inspect the Machined Part 393

CHAPTER 19  Fundamental Concepts of CNC Lathe Programming

19–1 Chapter Objectives 395
19–2 Introduction 395
19–3 Establishing Locations via Cartesian Coordinates (CNC Lathes) 397
19–4 Types of Tool Positioning Modes (CNC Lathes) 397
## Contents

19-5 Reference Point, Machine Origin, and Program Origin (FANUC Controllers)  398
19-6 Methodizing of Operations for CNC Lathes  400
19-7 Setup Procedures for CNC Lathes  402
19-8 Important Preparatory Functions (G Codes) for Lathes  408
19-9 Important Miscellaneous Functions (M Codes) for Lathes  409
19-10 Setting the Machining Origin  409
19-11 Feed Rate (F Code)  411
19-12 Spindle Speed (S Code)  412
19-13 Spindle Speed with Constant Surface Speed Control  412
19-14 Spindle Speed with Clamp Speed and Constant Surface Speed Controls  413
19-15 Automatic Tool Changing  414
19-16 Tool Edge Programming  415
19-17 Tool Nose Radius Compensation Programming  415
19-18 Setting Up Tool Nose Radius Compensation  416
19-19 Some Restrictions with Tool Nose Radius Compensation  418
19-20 Tool Nose Radius Compensation Commands  418
19-21 Linear Interpolation Commands (CNC Lathes)  421
19-22 Circular Interpolation Commands (CNC Lathes)  423
19-23 Grooving Commands  429
19-24 Return to Reference Point Command  431
19-25 Chapter Summary  432

**Review Exercises**

## CHAPTER 20  Techniques and Fixed Cycles for CNC Lathe Programming

20-1 Chapter Objectives  440
20-2 Introduction  440
20-3 Turning and Boring Cycle: G90  440
20-4 Facing Cycle: G94  446
20-5 Multiple Repetitive Cycles: G70 to G75  448
20-6 Stock Removal in Turning and Boring Cycle: G71  449
20-7 Finish Turning and Boring Cycle: G70  449
20-8 Peck Drilling and Face Grooving Cycle: G74  456
20-9 Peck Cutoff and Grooving Cycle: G75  463
20-10 Thread Cutting on CNC Lathes and Turning Centers  469
20-11 Single-Pass Threading Cycle: G32  469
20-12 Multiple-Pass Threading Cycle: G92  473
20-13 Multiple Repetitive Threading Cycle: G76  475
20-14 Chapter Summary  478

**Review Exercises**
CHAPTER 21  Modern Computer-Aided Part Programming  489

21-1  Chapter Objectives  489
21-2  Introduction  489
21-3  Modern Developments in the Process of CNC Programming  489
21-4  Using an Off-Line Computer to Write and Store Part Programs  490
21-5  An Introduction to Computer-Aided Programming Languages  490
21-6  Basic Elements of the APT Programming Language  491
21-7  An Introduction to CAD/CAM Technology  493
21-8  The Elements Comprising CAD  493
21-9  The Elements Comprising CAM  497
21-10  Creating a Complete Part Program Using Mastercam CNC Software  499
21-11  Chapter Summary  515

Review Exercises

Appendix A  517

Important Safety Precautions  517

Appendix B  519

Summary of G Codes for Milling Operations (FANUC Controllers)  519
Summary of G Codes for Turning Operations (FANUC Controllers)  521
Summary of M Codes for Milling and Turning Operations (FANUC Controllers)  522
Summary of Auxiliary Functions (FANUC Controllers)  523

Appendix C  526

Recommended Speeds and Feeds for Drilling  526
Recommended Speeds and Feeds for Milling  527
Recommended Speeds and Feeds for Turning  528

Appendix D  530

Summary of Important Machining Formulas  530

Appendix E  532

Tables of Important GDT Symbols  532

Appendix F  534

Identification System for OD and ID Tools  534
<table>
<thead>
<tr>
<th>Appendix G</th>
<th>537</th>
</tr>
</thead>
<tbody>
<tr>
<td>Writing and Verifying Mill Part Programs via Predator Simulations Software</td>
<td>537</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Appendix H</th>
<th>552</th>
</tr>
</thead>
<tbody>
<tr>
<td>Writing and Verifying Turn Part Programs via Predator Simulation Software</td>
<td>552</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Glossary</th>
<th>568</th>
</tr>
</thead>
</table>

| Index | 575 |