

A Comprehensive
Guide to Practical
CNC Programming

CNC PROGRAMMING HANDBOOK

PETER SMID

Foreword by Steve Krar

TABLE OF CONTENTS

1 - NUMERICAL CONTROL 1

DEFINITION OF NUMERICAL CONTROL	1
NC and CNC Technology.	1
CONVENTIONAL AND CNC MACHINING	2
NUMERICAL CONTROL ADVANTAGES	2
Setup Time Reduction	3
Lead Time Reduction	3
Accuracy and Repeatability	3
Contouring of Complex Shapes	3
Simplified Tooling and Work Holding	3
Cutting Time and Productivity Increase	4
TYPES OF CNC MACHINE TOOLS	4
Mills and Machining Centers	4
Lathes and Turning Centers	5
PERSONNEL FOR CNC	5
CNC Programmer	5
CNC Machine Operator	6
SAFETY RELATED TO CNC WORK	6

2 - CNC MILLING 7

CNC MACHINES - MILLING	7
Types of Milling Machines	7
Machine Axes	8
Vertical Machining Centers	8
Horizontal Machining Centers	9
Horizontal Boring Mill	10
Typical Specifications	10

3 - CNC TURNING 11

CNC MACHINES - TURNING	11
Types of CNC Lathes.	11
Number of Axes	11
AXES DESIGNATION	11
Two-axis Lathe	12
Three-axis Lathe	12
Four-axis Lathe	13
Six-axis Lathe	13
FEATURES AND SPECIFICATIONS	13
Typical Machine Specifications.	13
Control Features	14

4 - COORDINATE GEOMETRY 15

REAL NUMBER SYSTEM	15
RECTANGULAR COORDINATE SYSTEM	15

Axes and Planes	16
Point of Origin	16
Quadrants	16
Right Hand Coordinate System	17
MACHINE GEOMETRY.	17
Axis Orientation - Milling	17
Axis Orientation - Turning	18
Additional Axes.	18

5 - CONTROL SYSTEM 19

GENERAL DESCRIPTION	20
Operation Panel	20
Screen Display and Keyboard	21
Handle	22
SYSTEM FEATURES	22
Parameter Settings	22
System Defaults	23
Memory Capacity	24
MANUAL PROGRAM INTERRUPTION.	25
Single Block Operation	25
Feedhold	25
Emergency Stop	25
MANUAL DATA INPUT - MDI	26
PROGRAM DATA OVERRIDE	26
Rapid Motion Override	26
Spindle Speed Override	27
Feedrate Override	27
Dry Run Operation	27
Z Axis Neglect	28
Manual Absolute Setting	28
Sequence Return	28
Auxiliary Functions Lock	28
Machine Lock	28
Practical Applications	29
SYSTEM OPTIONS.	29
Graphic Display.	29
In-Process Gauging	30
Stored Stroke Limits	30
Drawing Dimensions Input	30
Machining Cycles	30
Cutting Tool Animation	30
Connection to External Devices	30

6 - PROGRAM PLANNING 31

STEPS IN PROGRAM PLANNING	31
INITIAL INFORMATION	31
MACHINE TOOLS FEATURES	31
Machine Type and Size.	31

Control System	31
PART COMPLEXITY	32
MANUAL PROGRAMMING	32
Disadvantages	32
Advantages	32
CAD/CAM AND CNC	32
Integration	33
Future of Manual Programming	33
TYPICAL PROGRAMMING PROCEDURE	33
PART DRAWING	34
Title Block	34
Dimensioning	34
Tolerances	35
Surface Finish	35
Drawing Revisions	36
Special Instructions	36
METHODS SHEET	36
MATERIAL SPECIFICATIONS	36
Material Uniformity	36
Machinability Rating	37
MACHINING SEQUENCE	37
TOOLING SELECTION	38
PART SETUP	38
Setup Sheet	38
TECHNOLOGICAL DECISIONS	38
Cutter Path	38
Machine Power Rating	39
Coolants and Lubricants	39
WORK SKETCH AND CALCULATIONS	40
Identification Methods	40
QUALITY IN CNC PROGRAMMING	40

8 - PREPARATORY COMMANDS 47

DESCRIPTION AND PURPOSE.	47
APPLICATIONS FOR MILLING	47
APPLICATIONS FOR TURNING	49
G CODES IN A PROGRAM BLOCK	50
Modality of G-commands	50
Conflicting Commands in a Block	50
Word Order in a Block	51
GROUPING OF COMMANDS	51
Group Numbers	51
G CODE TYPES.	52
G Codes and Decimal Point	52

9 - MISCELLANEOUS FUNCTIONS 53

DESCRIPTION AND PURPOSE.	53
Machine Related Functions	53
Program Related Functions	53
TYPICAL APPLICATIONS	54
Applications for Milling	54
Applications for Turning	54
Special MDI Functions	54
Application Groups	54
M FUNCTIONS IN A BLOCK	55
Startup of M Functions	56
Duration of M Functions	56
PROGRAM FUNCTIONS	56
Program Stop	56
Optional Program Stop	57
Program End	58
Subprogram End	58
MACHINE FUNCTIONS	58
Coolant Functions	58
Spindle Functions	59
Gear Range Selection	60
Machine Accessories	60

10 - SEQUENCE BLOCK 61

BLOCK STRUCTURE	61
Building the Block Structure	61
Block Structure for Milling	61
PROGRAM IDENTIFICATION	62
Program Number	62
Program Name	62
SEQUENCE NUMBERS	63
Sequence Number Command	63
Sequence Block Format	63
Numbering Increment	64
Long Programs and Block Numbers	64
END OF BLOCK CHARACTER	64
STARTUP BLOCK OR SAFE BLOCK	65

7 - PART PROGRAM STRUCTURE 41

BASIC PROGRAMMING TERMS	41
Character	41
Word	41
Block	41
Program	42
PROGRAMMING FORMATS	42
WORD ADDRESS FORMAT	42
FORMAT NOTATION	43
Short Forms	43
Milling System Format	43
Turning System Format	44
Multiple Word Addresses	45
SYMBOLS IN PROGRAMMING	45
Plus and Minus Sign	45
PROGRAM HEADER	45
TYPICAL PROGRAM STRUCTURE.	46

PROGRAM COMMENTS	66
CONFLICTING WORDS IN A BLOCK	66
MODAL PROGRAMMING VALUES	67
EXECUTION PRIORITY	68

11 - INPUT OF DIMENSIONS 69

ENGLISH AND METRIC UNITS	69
Comparable Unit Values	70
ABSOLUTE AND INCREMENTAL MODES	70
Preparatory Commands G90 and G91	71
Absolute Data Input - G90	72
Incremental Data Input - G91	72
Combinations in a Single Block	72
DIAMETER PROGRAMMING	73
MINIMUM MOTION INCREMENT	73
FORMAT OF DIMENSIONAL INPUT	73
Full Address Format	74
Zero Suppression	74
Decimal Point Programming	75
Input Comparison	76
CALCULATOR TYPE INPUT	76

12 - SPINDLE CONTROL 77

SPINDLE FUNCTION	77
Spindle Speed Input	77
DIRECTION OF SPINDLE ROTATION	77
Direction for Milling	78
Direction for Turning	78
Direction Specification	79
Spindle Startup	79
SPINDLE STOP	80
SPINDLE ORIENTATION	80
SPINDLE SPEED - R/MIN	81
SPINDLE SPEED - SURFACE	81
Material Machinability	81
Spindle Speed - English Units	82
Spindle Speed - Metric Units	82
CONSTANT SURFACE SPEED	82
Maximum Spindle Speed Setting	84
Part Diameter Calculation in CSS	85

13 - FEEDRATE CONTROL 87

FEEDRATE CONTROL	87
FEEDRATE FUNCTION	87
Feedrate per Minute	87
Feedrate per Revolution	88
FEEDRATE SELECTION	88
ACCELERATION AND DECELERATION	88

Exact Stop Command	89
Exact Stop Mode Command	89
Automatic Corner Override	89
Tapping Mode	89
Cutting Mode	90

CONSTANT FEEDRATE 90

Circular Motion Feedrates	90
-------------------------------------	----

MAXIMUM FEEDRATE. 91

Maximum Feedrate Considerations	91
---	----

FEEDHOLD AND OVERRIDE 91

Feedhold Switch	91
Feedrate Override Switch	91
Feedrate Override Functions	92

E ADDRESS IN THREADING 92

14 - TOOL FUNCTION 93

T FUNCTION FOR MACHINING CENTERS 93

Tool Storage Magazine	93
Fixed Tool Selection	94
Random Memory Tool Selection	94
Registering Tool Numbers	94
Programming Format	95
Empty Tool or Dummy Tool	95

TOOL CHANGE FUNCTION - M06. 95

Conditions for Tool Change	95
--------------------------------------	----

AUTOMATIC TOOL CHANGER - ATC 96

Typical ATC System	96
Maximum Tool Diameter	97
Maximum Tool Length	97
Maximum Tool Weight	97
ATC Cycle	98
MDI Operation	98

PROGRAMMING THE ATC. 98

Single Tool Work	98
Programming Several Tools	99
Keeping Track of Tools	99
Any Tool in Spindle - Not the First	99
First Tool in the Spindle	100
No Tool in the Spindle	101
First Tool in the Spindle with Manual Change	101
No Tool in the Spindle with Manual Change	102
First Tool in the Spindle and an Oversize Tool	102
No Tool in the Spindle and an Oversize Tool	102

T FUNCTION FOR LATHES 103

Lathe Tool Station	103
Tool Indexing	103

TOOL OFFSET REGISTERS 104

Geometry Offset	104
Wear Offset	105
Wear Offset Adjustment	106
The R and T Settings	106

15 - REFERENCE POINTS 107

REFERENCE POINT GROUPS	107
----------------------------------	-----

Reference Point Groups Relationship	108
MACHINE REFERENCE POINT	108
Return to Machine Zero	109
PART REFERENCE POINT	109
Program Zero Selection	109
Program Zero - Machining Centers	110
Program Zero - Lathes	112
TOOL REFERENCE POINT	112

16 - REGISTER COMMANDS 113

POSITION REGISTER COMMAND	113
Position Register Definition	113
Programming Format	113
Tool Position Setting	114
MACHINING CENTERS APPLICATION	114
Tool Set at Machine Zero	114
Tool Set Away from Machine Zero	114
Position Register in Z Axis	115
Programming Example	115
LATHE APPLICATION	115
Tool Setup	116
Three-Tool Setup Groups	116
Center Line Tools Setup	116
External Tools Setup	117
Internal Tool Setup	117
Corner Tip Detail	117
Programming Example	117

17 - POSITION COMPENSATION 119

DESCRIPTION	119
Programming Commands	119
Programming Format	119
Incremental Mode	120
Motion Length Calculation	120
Position Compensation Along the Z axis	122
Using G47 and G48	122
Face Milling	122

18 - WORK OFFSETS 123

WORK AREAS AVAILABLE	123
Additional Work Offsets	124
WORK OFFSET DEFAULT AND STARTUP	124
Work Offset Change	125
Z Axis Application	126
HORIZONTAL MACHINE APPLICATION	127
EXTERNAL WORK OFFSETS	128
LATHE APPLICATIONS	128
Types of Offsets	128
Geometry Offset	128
Wear Offset	129
Tool and Offset Numbers	129

TOOL SETUP	129
Center Line Tools	129
Turning Tools	130
Boring Tools	130
Command Point and Tool Work Offset	130

19 - TOOL LENGTH OFFSET 131

GENERAL PRINCIPLES	131
Actual Tool Length	131
Gauge Line	131
Table Top Face	132
TOOL LENGTH OFFSET COMMANDS	132
Distance-To-Go in Z Axis	132
TOOL LENGTH SETUP	133
On-Machine Tool Length Setting	133
Off-Machine Tool Length Setting	134
Tool Length Offset Value Register	134
Z AXIS RELATIONSHIPS	134
Preset Tool Length	135
Tool Length by Touch Off	135
Using a Master Tool Length	136
G43-G44 Difference	136
PROGRAMMING FORMATS	137
Tool Length Offset not Available	137
Tool Length Offset and G92	138
Tool Length Offset and G54-G59	139
Tool Length Offset and Multiple Tools	139
CHANGING TOOL LENGTH OFFSET	140
HORIZONTAL MACHINE APPLICATION	141
TOOL LENGTH OFFSET CANCEL	141

20 - RAPID POSITIONING 143

RAPID TRAVERSE MOTION	143
G00 Command	143
RAPID MOTION TOOL PATH	144
Single Axis Motion	144
Multiaxis Motion	144
Straight Angular Motion	146
Reverse Rapid Motion	146
TYPE OF MOTION & TIME COMPARISON	146
REDUCTION OF RAPID MOTION RATE	147
RAPID MOTION FORMULAS	147
APPROACH TO THE PART	148

21 - MACHINE ZERO RETURN 149

MACHINE REFERENCE POSITION	149
Machining Centers	149
Lathes	150
Setting the Machine Axes	150
Program Commands	151
Command Group	151

RETURN TO PRIMARY MACHINE ZERO	151
Intermediate Point	151
Absolute and Incremental Mode	152
Return from the Z Depth Position	153
Axes Return Required for the ATC	155
Zero Return for CNC Lathes	155
RETURN POSITION CHECK COMMAND	156
RETURN FROM MACHINE ZERO POINT	157
RETURN TO SECONDARY MACHINE ZERO	158

22 - LINEAR INTERPOLATION 159

LINEAR COMMAND	159
Start and End of the Linear Motion	159
Single Axis Linear Interpolation	159
Two Axes Linear Interpolation	160
Three Axis Linear Interpolation	160
PROGRAMMING FORMAT	160
LINEAR FEEDRATE	161
Feedrate Range	161
Individual Axis Feedrate	161
PROGRAMMING EXAMPLE	162

23 - BLOCK SKIP FUNCTION 163

TYPICAL APPLICATIONS	163
BLOCK SKIP SYMBOL	163
CONTROL UNIT SETTING	163
BLOCK SKIP AND MODAL COMMANDS	164
PROGRAMMING EXAMPLES	165
Variable Stock Removal	165
Machining Pattern Change	166
Trial Cut for Measuring	168
Program Proving	169
Barfeeder Application	170
Numbered Block Skip	170

24 - DWELL COMMAND 171

PROGRAMMING APPLICATIONS	171
Applications for Cutting	171
Applications for Accessories	171
DWELL COMMAND	171
Dwell Command Structure	172
DWELL TIME SELECTION	172
SETTING MODE AND DWELL	173
Time Setting	173
Number of Revolutions Setting	173
MINIMUM DWELL	173
NUMBER OF REVOLUTIONS	174
System Setting	174
Time Equivalent	174

LONG DWELL TIME	175
Machine Warm-Up	175
X Axis is the Dwelling Axis	176
Safety and Dwell	176
FIXED CYCLES AND DWELL	176

25 - FIXED CYCLES 177

POINT-TO-POINT MACHINING	177
Single Tool Motions vs. Fixed Cycles	178
FIXED CYCLE SELECTION	178
PROGRAMMING FORMAT	179
GENERAL RULES	180
ABSOLUTE AND INCREMENTAL VALUES	180
INITIAL LEVEL SELECTION	181
R LEVEL SELECTION	181
Z DEPTH CALCULATIONS	182
DESCRIPTION OF FIXED CYCLES	183
G81 - Drilling Cycle	183
G82 - Spot-Drilling Cycle	183
G83 - Deep Hole Drilling Cycle - Standard	184
G73 - Deep Hole Drilling Cycle - High Speed	184
G84 - Tapping Cycle - Standard	186
G74 - Tapping Cycle - Reverse	186
G85 - Boring Cycle	187
G86 - Boring Cycle	187
G87 - Backboring Cycle	187
G88 - Boring Cycle	188
G89 - Boring Cycle	188
G76 - Precision Boring Cycle	189
FIXED CYCLE CANCELLATION	189
FIXED CYCLE REPETITION	189
The L or K Address	190
L0 or K0 in a Cycle	190

26 - MACHINING HOLES 191

SINGLE HOLE EVALUATION	191
Tooling Selection and Applications	191
Program Data	194
DRILLING OPERATIONS	194
Types of Drilling Operations	194
Types of Drills	194
Programming Considerations	195
Nominal Drill Diameter	195
Effective Drill Diameter	195
Drill Point Length	195
Center Drilling	196
Through Hole Drilling	196
Blind Hole Drilling	197
Flat Bottom Drilling	197
Indexable Insert Drilling	198
PECK DRILLING	199
Typical Peck Drilling Application	199
Calculating the Number of Pecks	199

Selecting the Number of Pecks	200
Controlling Breakthrough Depth.	200
REAMING	201
Reamer Design	201
Spindle Speeds for Reaming	201
Feedrates for Reaming	201
Stock Allowance	202
Other Reaming Considerations	202
SINGLE POINT BORING	202
Single Point Boring Tool	202
Spindle Orientation	203
Block Tools	203
BORING WITH A TOOL SHIFT	203
Precision Boring Cycle G76	203
Backboring Cycle G87.	204
Programming Example	204
Precautions in Programming and Setup.	205
ENLARGING HOLES	205
Countersinking	205
Counterboring.	206
Spotfacing	207
MULTILEVEL DRILLING	207
WEB DRILLING	208
TAPPING	209
Tap Geometry	210
Tapping Speed and Feedrate	210
Pipe Taps.	211
Tapping Check List	212
HOLE OPERATIONS ON A LATHE	212
Tool Approach Motion	213
Tool Return Motion	213
Drilling and Reaming on Lathes	214
Peck Drilling Cycle - G74.	214
Tapping on Lathes	215
Other Operations	216

27 - PATTERN OF HOLES 217

TYPICAL HOLE PATTERNS	217
RANDOM HOLE PATTERN	217
STRAIGHT ROW HOLE PATTERN	218
ANGULAR ROW HOLE PATTERN	218
Pattern Defined by Coordinates	218
Pattern Defined by Angle	219
CORNER PATTERN	220
GRID PATTERN	220
Angular Grid Pattern	221
ARC HOLE PATTERN	222
BOLT HOLE CIRCLE PATTERN	223
Bolt Circle Formula	224
Pattern Orientation	224
POLAR COORDINATE SYSTEM	225
Plane Selection	226
Order of Machining.	226

28 - FACE MILLING 227

CUTTER SELECTION	227
Basic Selection Criteria	227
Face Mill Diameter	227
Insert Geometry	228
CUTTING CONSIDERATIONS	228
Angle of Entry	229
Milling Mode	229
Number of Cutting Inserts	230
PROGRAMMING TECHNIQUES	230
Single Face Mill Cut	231
Multiple Face Mill Cuts	232
USING POSITION COMPENSATION	233

29 - CIRCULAR INTERPOLATION 235

ELEMENTS OF A CIRCLE	235
Radius and Diameter	235
Circle Area and Circumference	236
QUADRANTS	236
Quadrant Points	236
PROGRAMMING FORMAT	237
Arc Cutting Direction	237
Circular Interpolation Block.	237
Arc Start and End Points.	238
Arc Center and Radius	238
Arc Center Vectors	238
Arc in Planes	239
RADIUS PROGRAMMING	240
Blend Radius	240
Partial Radius	240
FULL CIRCLE PROGRAMMING	240
Boss Milling	242
Internal Circle Cutting - Linear Start	243
Internal Circle Cutting - Circular Start	243
Circle Cutting Cycle	244
ARC PROGRAMMING.	245
FEEDRATE FOR CIRCULAR MOTION	245
Feedrate for Outside Arcs	246
Feedrate for Inside Arcs	246

30 - CUTTER RADIUS OFFSET 247

MANUAL CALCULATIONS	247
Tool Path Center Points	248
Cutter Radius	249
Center Points Calculation	249
COMPENSATED CUTTER PATH	250
Types of Cutter Radius Offset.	250
Definition and Applications.	250
PROGRAMMING TECHNIQUES	250
Direction of Cutting Motion	251

Left or Right - not CW or CCW	251	High Speed Steel End Mills	276
Offset Commands	251	Solid Carbide End Mills	276
Radius of the Cutter	252	Indexable Insert End Mills	276
History of Offset Types	252	Relief Angles	276
Programming Format	253	End Mill Size	276
Address H or D ?	253	Number of Flutes	277
Geometry and Wear Offsets	254	SPEEDS AND FEEDS	277
APPLYING CUTTER RADIUS OFFSET	254	Coolants and Lubricants	278
Startup Methods	254	Tool Chatter	278
Offset Cancellation	256	STOCK REMOVAL	279
Cutter Direction Change	256	Plunge Infeed	279
HOW THE RADIUS OFFSET WORKS	256	In and Out Ramping	279
Look-Ahead Offset Type	257	Direction of Cut	279
Rules for Look-Ahead Cutter Radius Offset	257	Width and Depth of Cut	280
Radius of the Cutter	258		
Radius Offset Interference	259		
Single vs. Multiaxis Startup	260		
OVERVIEW OF GENERAL RULES	261		
PRACTICAL EXAMPLE - MILLING	262	33 - SLOTS AND POCKETS	281
Part Tolerances	262	OPEN AND CLOSED BOUNDARY	281
Measured Part Size	262	Open Boundary	281
Programmed Offsets	263	Closed Boundary	281
D Offset Amount - General Setting	263	PROGRAMMING SLOTS	281
Offset Adjustment	264	Open Slot Example	281
Offset for External Cutting	264	Closed Slot Example	283
Offset for Internal Cutting	265	POCKET MILLING	284
One Offset or Multiple Offsets?	265	General Principles	284
Preventing a Scrap	265	Pocket Types	285
Program Data - Nominal or Middle?	266	RECTANGULAR POCKETS	285
TOOL NOSE RADIUS OFFSET	266	Stock Amount	286
Tool Nose Tip	266	Stepover Amount	286
Radius Offset Commands	266	Length of Cut	287
Tool Tip Orientation	266	Semifinishing Motions	287
Effect of Tool Nose Radius Offset	267	Finishing Tool Path	287
Sample Program	267	Rectangular Pocket Program Example	288
Minimum Clearance Required	268	CIRCULAR POCKETS	289
Change of Motion Direction	268	Minimum Cutter Diameter	289
		Method of Entry	289
		Linear Approach	289
		Linear and Circular Approach	290
		Roughing a Circular Pocket	291
		CIRCULAR POCKET CYCLES	292
31 - PLANE SELECTION	269		
WHAT IS A PLANE?	269	34 - TURNING AND BORING	293
MACHINING IN PLANES	269	TOOL FUNCTION - TURNING	293
Mathematical Planes	269	T Address	293
Machine Tool Planes	270	LATHE OFFSETS	294
Program Commands for Planes Definition	270	Offset Entry	294
Default Control Status	271	Independent Tool Offset	294
STRAIGHT MOTION IN PLANES	271	Tool Offset with Motion	295
CIRCULAR INTERPOLATION IN PLANES	271	Offset Change	295
G17-G18-G19 as Modal Commands	272	MULTIPLE OFFSETS	295
Absence of Axis Data in a Block	272	General Approach	296
Cutter Radius Offset in Planes	273	Diameter Tolerances	296
PRACTICAL EXAMPLE	273	Shoulder Tolerances	297
FIXED CYCLES IN PLANES	274	Diameter and Shoulder Tolerances	297
		OFFSET SETTING	298
32 - PERIPHERAL MILLING	275		
END MILLS	275		

FUNCTIONS FOR GEAR RANGES	298
AUTOMATIC CORNER BREAK	299
Chamfering at 45 Degrees	299
Blend Radius at 90 Degrees	301
Programming Conditions	301
Programming Example	301
ROUGH AND FINISHED SHAPE	302
Rough Operations	302
Finish Operations	303
Stock and Stock Allowance	303
PROGRAMMING A RECESS	304
SPINDLE SPEED IN CSS MODE	305
LATHE PROGRAM FORMAT	306
Program Format - Templates	306
General Program Format	306
Approach to the Part	306

35 - LATHE CYCLES 307

STOCK REMOVAL ON LATHES	307
Simple Cycles	307
Complex Cycles	307
PRINCIPLES OF LATHE CYCLES	307
G90 - STRAIGHT CUTTING CYCLE	308
Cycle Format	308
Straight Turning Example	309
Taper Cutting Example	309
Straight and Taper Cutting Example	311
G94 - FACE CUTTING CYCLE	312
Cycle Format	312
MULTIPLE REPETITIVE CYCLES	313
General Description	313
Cycle Format Types	313
Cutting Cycles and Part Contour	313
Chipbreaking Cycles	313
CONTOUR CUTTING CYCLES	314
Boundary Definition	314
Start Point and the Points P and Q	314
TYPE I AND TYPE II CYCLES	315
Programming Type I and Type II Cycles	315
Cycle Formatting	315
G71 - STOCK REMOVAL IN TURNING	315
G71 Cycle Format - 10T/11T/15T	316
G71 Cycle Format - 0T/16T/18T/20T/21T	316
G71 for External Roughing	316
G71 for Internal Roughing	317
Direction of Cutting in G71	317
G72 - STOCK REMOVAL IN FACING	317
G72 Cycle Format - 10T/11T/15T	317
G72 Cycle Format - 0T/16T/18T/20T/21T	318
G73 - PATTERN REPEATING CYCLE	318
G73 Cycle Format - 10T/11T/15T	318
G73 Cycle Format - 0T/16T/18T/20T/21T	319
G73 Example of Pattern Repeating	319
G70 - CONTOUR FINISHING CYCLE	320

G70 Cycle Format - All Controls	320
BASIC RULES FOR G70-G73 CYCLES	321
G74 - PECK DRILLING CYCLE	321
G74 Cycle Format - 10T/11T/15T	321
G74 Cycle Format - 0T/16T/18T/20T/21T	321
G75 - GROOVE CUTTING CYCLE	322
G75 Cycle Format - 10T/11T/15T	322
G75 Cycle Format - 0T/16T/18T/20T/21T	322
BASIC RULES FOR G74 AND G75 CYCLES	322

36 - GROOVING ON LATHES 323

GROOVING OPERATIONS	323
Main Grooving Applications	323
Grooving Criteria	323
GROOVE SHAPE	323
Nominal Insert Size	324
Insert Modification	324
GROOVE LOCATION	324
GROOVE DIMENSIONS	324
Groove Position	325
Groove Depth	325
SIMPLE GROOVE PROGRAMMING	325
PRECISION GROOVING TECHNIQUES	326
Groove Width Selection	327
Machining Method	327
Finishing Allowances	328
Groove Tolerances	329
Groove Surface Finish	330
MULTIPLE GROOVES	330
FACE GROOVES	330
Radial Clearance	331
Face Grooving Program Example	331
CORNER GROOVES / NECK GROOVES	332
GROOVING CYCLES	332
G75 Cycle Applications	332
Single Groove with G75	333
Multiple Grooves with G75	333
SPECIAL GROOVES	333
GROOVES AND SUBPROGRAMS	334

37 - PART-OFF 335

PART-OFF PROCEDURE	335
Parting Tool Description	335
Tool Approach Motion	336
Stock Allowance	337
Tool Return Motion	337
Part-off with a Chamfer	337
Preventing Damage to the Part	338

38 - SINGLE POINT THREADING 339

THREADING ON CNC LATHES	339
Form of a Thread	339
Threading Operations	340
TERMINOLOGY OF THREADING	340
THREADING PROCESS	341
Steps in Threading	341
Thread Start Position	342
Thread Cutting Diameter and Depth	342
Thread Cutting Motion	343
Retract from Thread	344
Return to Start Position	344
THREADING FEED AND SPINDLE SPEED	344
Threading Feedrate Selection	345
Spindle Speed Selection	345
Maximum Threading Feedrate	346
Lead Error	347
TOOL REFERENCE POINT	348
BLOCK-BY-BLOCK THREADING	348
BASIC THREADING CYCLE - G92	349
MULTIPLE REPETITIVE CYCLE - G76	350
G76 Cycle Format - 10T/11T/15T	350
G76 Cycle Format - 0T/16T/18T	351
Programming Example	351
First Thread Calculation	352
THREAD INFEEED METHODS	353
Radial Infeed	353
Compound Infeed	353
Thread Insert Angle - Parameter A	354
Thread Cutting Type - Parameter P	354
ONE-BLOCK METHOD CALCULATIONS	355
Initial Considerations	355
Z Axis Start Position Calculation	355
THREAD RETRACT MOTION	357
Thread Pullout Functions	357
Single Axis Pullout	357
Two-Axis Pullout	357
HAND OF THREAD	358
THREADING TO A SHOULDER	358
Insert Modification	358
Program Testing	360
OTHER THREAD FORMS	360
Thread Depth	360
TAPERED THREAD	361
Depth and Clearances	361
Taper Calculation	361
Block by Block Taper Thread	362
Tapered Thread Using a Simple Cycle	363
Tapered Thread and a Multi Repetitive Cycle	363
MULTISTART THREAD	364
Threading Feedrate Calculation	364
Shift Amount	365
Application Example	366
THREAD RECUTTING	366

39 - SUBPROGRAMS**367**

MAIN PROGRAM AND SUBPROGRAMS	367
Subprogram Benefits	367
Identification of Subprograms	368
SUBPROGRAM FUNCTIONS	368
Subprogram Call Function	368
Subprogram End Function	368
Block Number to Return to	369
Number of Subprogram Repetitions	369
L0 in a Subprogram Call	370
SUBPROGRAM NUMBERING	371
Organized Approach	372
Protected Subprograms	373
SUBPROGRAM DEVELOPMENT	373
Repeating Pattern Recognition	373
Tool Motion and Subprograms	374
Modal Values and Subprograms	375
MULTI LEVEL NESTING	376
One Level Nesting	376
Two Level Nesting	376
Three Level Nesting	377
Four Level Nesting	377
Nesting Applications	377
CONTOURING WITH A SUBPROGRAM	378
TOOL CHANGE SUBPROGRAM	379
100 000 000 HOLE GRID	379

40 - DATUM SHIFT**381**

DATUM SHIFT WITH G92 OR G50	381
Program Zero Shift	381
LOCAL COORDINATE SYSTEM	383
G52 Command	384
MACHINE COORDINATE SYSTEM	384
DATA SETTING	386
Data Setting Command	386
Coordinate Mode	386
WORK OFFSETS	386
Standard Work Offset Input	386
Additional Work Offset Input	387
External Work Offset Input	387
TOOL LENGTH OFFSETS	387
Valid Input Range	388
CUTTER RADIUS OFFSETS	388
LATHE OFFSETS	388
MDI DATA SETTING	389
PROGRAMMABLE PARAMETER ENTRY	389
Modal G10 Command	389
Parameters Notation	390
Program Portability	390
Bit Type Parameter	391
Effect of Block Numbers	392

41 - MIRROR IMAGE 393

BASIC RULES OF MIRROR IMAGE	393
Tool Path Direction	393
Original Tool Path	394
Mirror Axis	394
Sign of Coordinates	394
Milling Direction	395
Arc Motion Direction	395
Program Start and End	395
MIRROR IMAGE BY SETTING	395
Control Setting	395
Programming - Manual Mirror Setting	396
PROGRAMMABLE MIRROR IMAGE	396
Mirror Image Functions	396
Simple Mirror Image Example	397
Complete Mirror Image Example	398
MIRROR IMAGE ON CNC LATHES	398

42 - COORDINATE ROTATION 399

ROTATION COMMANDS	399
Center of Rotation	399
Radius of Rotation	399
Coordinate Rotation Cancel	401
Common Applications	401
PRACTICAL APPLICATION	401

43 - SCALING FUNCTION 405

DESCRIPTION	405
Scaling Function Usage	405
PROGRAMMING FORMAT	405
Scaling Center	406
Scaling Factor	406
Rounding Errors in Scaling	407
PROGRAM EXAMPLES	407

44 - CNC LATHE ACCESSORIES 409

CHUCK CONTROL	409
Chuck Functions	409
Chuck Pressure	410
Chuck Jaws	410
TAILSTOCK AND QUILL	410
Tailstock Body	410
Quill	411
Center	411
Quill Functions	411
Programmable Tailstock	411
Safety Concerns	411
BI-DIRECTIONAL TURRET INDEXING	412
Programming Example	412

BARFEEDER ATTACHMENT	413
Bar Stopper	413

ADDITIONAL OPTIONS 414

Part Catcher or Part Unloader	414
Continuous Operation	414
Parts Counter	415
Pull-Out Finger	415

PROGRAMMING EXAMPLE 415

45 - HELICAL MILLING 417

HELICAL MILLING OPERATION	417
Helical Interpolation	417
Programming Format	417
Arc Modifiers for Helical Interpolation	418
Applications and Usage	418
THREAD MILLING	418
Applying Thread Milling	418
Conditions for Thread Milling	418
Thread Milling Tool	419
Premachining Requirements	419
Clearance Radius	419
Productivity of Thread Milling	419

THE HELIX 419

THREAD MILLING EXAMPLE 421

Straight Thread	421
Initial Calculations	421
Starting Position	422
Motion Rotation and Direction	422
Lead-In Motions	423
Thread Rise Calculation	424
Milling the Thread	424
Lead-Out Motions	425
Complete Program	425
External Thread Milling	425
Tapered Thread Milling	425
Further Considerations	426

THREAD MILLING SIMULATION METHOD 426

HELICAL RAMPING 427

46 - HORIZONTAL MACHINING 429

INDEXING AND ROTARY AXES	429
INDEXING TABLE (B AXIS)	429
Units of Increment	429
Direction of Indexing	430
Table Clamp and Unclamp Functions	430
Indexing in Absolute and Incremental Mode	430
B AXIS AND OFFSETS	431
Work Offset and B Axis	431
Tool Length Offset and B Axis	432
RETURN TO MACHINE ZERO	434
INDEXING AND A SUBPROGRAM	434
COMPLETE PROGRAM EXAMPLE	436
AUTOMATIC PALLET CHANGER - APC	437

Working Environment	438
Types of Pallets	438
Programming Commands	439
Pallet Changing Program Structure	439
HORIZONTAL BORING MILL	439

47 - WRITING A CNC PROGRAM 441

PROGRAM WRITING	441
Legibility of Handwriting	442
Programming Forms	442
Confusing Characters	442
PROGRAM OUTPUT FORMATTING	443
LONG PROGRAMS	445
Program Length Reduction	445
Memory Mode and Tape Mode	446

48 - PROGRAM DOCUMENTS 447

DATA FILES	447
PROGRAM DOCUMENTATION	448
External Documentation	448
Internal Documentation	448
Program Description	449
SETUP AND TOOLING SHEETS	449
Setup Sheet	450
Tooling Sheet	450
Coordinate Sheet	451
DOCUMENTATION FILE FOLDER	451
Identification Methods	451
Operator's Suggestions	452
Filing and Storage	452

49 - PROGRAM VERIFICATION 453

DETECTION OF ERRORS	453
Preventive Measures	453
Corrective Measures	453
GRAPHIC VERIFICATION	454
AVOIDING ERRORS	454
Syntax Errors	455
Logical Errors	455
COMMON PROGRAMMING ERRORS	456
Program Input Errors	456
Calculation Errors	456
Hardware Errors	456
Miscellaneous Errors	456

50 - CNC MACHINING 457

MACHINING A NEW PART	457
Setup Integrity	458
Program Integrity	458

RUNNING THE FIRST PART	459
PROGRAM CHANGES	460
Program Upgrading	460
Program Updating	461
Documentation Change	461
ALTERNATE MACHINE SELECTION	461
MACHINE WARM UP PROGRAM	462
CNC MACHINING AND SAFETY	462
SHUTTING DOWN A CNC MACHINE	463
Emergency Stop Switch	463
Parking Machine Slides	464
Setting the Control System	464
Turning the Power Off	464
EQUIPMENT MAINTENANCE	464

51 - INTERFACING TO DEVICES 465

RS-232C INTERFACE	465
PUNCHED TAPE	466
Tape Reader and Puncher	466
Leader and Trailer	468
Tape Identification	468
Non-printable Characters	468
Storage and Handling	468
DISTRIBUTED NUMERICAL CONTROL	468
TERMINOLOGY OF COMMUNICATIONS	469
Baud Rate	469
Parity	469
Data Bits	469
Start and Stop Bits	469
DATA SETTING	469
CONNECTING CABLES	470
Null Modem	470
Cabling for Fanuc and PC	470

52 - MATH IN CNC PROGRAMMING 471

BASIC ELEMENTS	471
Arithmetic and Algebra	471
Order of Calculations	471
GEOMETRY	472
Circle	472
PI Constant	473
Circumference of a Circle	473
Length of Arc	473
Quadrants	473
POLYGONS	474
TAPERS	475
Taper Definition	475
Taper Per Foot	476
Taper Ratio	476
Taper Calculations - English Units	476
Taper Calculations - Metric Units	476
CALCULATIONS OF TRIANGLES	477

Types of Angles and Triangles	477
Right Triangles	478
Similar Triangles	478
Sine - Cosine - Tangent	479
Inverse Trigonometric Functions	480
Degrees and Decimal Degrees	480
Pythagorean Theorem	480
Solving Right Triangles	481
ADVANCED CALCULATIONS	482
CONCLUSION	482

53 - CNC AND CAD/CAM 483

PROGRAMMING MANUALLY ?	483
CAM Software	483
Desktop Computer Programming	483
TOOL PATH GEOMETRY DEVELOPMENT	484
TOOL PATH GENERATION	484
COMPLETE ENVIRONMENT	484
Multi Machine Support	485
Associative Operations	485
Job Setup	485
Tooling List and Job Comments	485
Connection Between Computers	485
Program Text Editor	486
Printing Capabilities	486
Pen Plotting	486
CAD Software Access	486
Support for Solids	486
Software Specifications	487

Hardware Specifications	487
Typical Hardware Requirements	488
Utilities and Special Features	488
POST PROCESSORS	488
Customizing Post Processor	488
IMPORTANT FEATURES	489
Input from User	489
Machining Cycles	489
User Interface	489
CAD Interface	489
SUPPORT AND MANAGEMENT	489
Training	490
System Management	490
Technical Support	490
THE END AND THE BEGINNING	490

A - REFERENCE TABLES 491

Decimal Equivalents	491
English Threads - UNC/UNF	494
Straight Pipe Taps NPS	494
Taper Pipe Taps NPT	495
Metric Coarse Threads	495
Metric Fine Threads	495
Index	497