

TABLE OF CONTENTS

1 - NUMERICAL CONTROL	1	Quadrants	16
DEFINITION OF NUMERICAL CONTROL	1	Right Hand Coordinate System	17
NC and CNC Technology	1	MACHINE GEOMETRY	17
CONVENTIONAL AND CNC MACHINING	2	Axis Orientation - Milling	17
NUMERICAL CONTROL ADVANTAGES	2	Axis Orientation - Turning	18
Setup Time Reduction	3	Additional Axes	18
Lead Time Reduction	3	5 - CONTROL SYSTEM	19
Accuracy and Repeatability	3	GENERAL DESCRIPTION	20
Contouring of Complex Shapes	3	Operation Panel	20
Simplified Tooling and Work Holding	3	Screen Display and Keyboard	21
Cutting Time and Productivity Increase	4	Handle	22
TYPES OF CNC MACHINE TOOLS	4	SYSTEM FEATURES	22
Mills and Machining Centers	4	Parameter Settings	22
Lathes and Turning Centers	5	System Defaults	23
PERSONNEL FOR CNC	5	Memory Capacity	24
CNC Programmer	5	MANUAL PROGRAM INTERRUPTION	25
CNC Machine Operator	6	Single Block Operation	25
SAFETY RELATED TO CNC WORK	6	Feedhold	25
2 - CNC MILLING	7	Emergency Stop	25
CNC MACHINES - MILLING	7	MANUAL DATA INPUT - MDI	26
Types of Milling Machines	7	PROGRAM DATA OVERRIDE	26
Machine Axes	8	Rapid Motion Override	26
Vertical Machining Centers	8	Spindle Speed Override	27
Horizontal Machining Centers	10	Feedrate Override	27
Horizontal Boring Mill	10	Dry Run Operation	27
Typical Specifications	10	Z-Axis Neglect	28
3 - CNC TURNING	11	Manual Absolute Setting	28
CNC MACHINES - TURNING	11	Sequence Return	28
Types of CNC Lathes	11	Auxiliary Functions Lock	28
Number of Axes	11	Machine Lock	28
AXES DESIGNATION	11	Practical Applications	29
Two-axis Lathe	12	SYSTEM OPTIONS	29
Three-axis Lathe	12	Graphic Display	29
Four-axis Lathe	13	In-Process Gauging	30
Six-axis Lathe	13	Stored Stroke Limits	30
FEATURES AND SPECIFICATIONS	13	Drawing Dimensions Input	30
Typical Machine Specifications	13	Machining Cycles	30
Control Features	14	Cutting Tool Animation	30
4 - COORDINATE GEOMETRY	15	Connection to External Devices	30
REAL NUMBER SYSTEM	15	6 - PROGRAM PLANNING	31
RECTANGULAR COORDINATE SYSTEM	15	STEPS IN PROGRAM PLANNING	31
Axes and Planes	16	INITIAL INFORMATION	31
Point of Origin	16	CNC MACHINE FEATURES	31
		Machine Type and Size	31
		Control System	32
		PART COMPLEXITY	32
		MANUAL PROGRAMMING	32

Disadvantages	32	APPLICATIONS FOR MILLING	51
Advantages	32	APPLICATIONS FOR TURNING	53
CAD/CAM AND CNC	33	G-CODES IN A PROGRAM BLOCK	54
Integration	33	Modality of G-codes	54
Future of Manual Programming	33	Conflicting Commands in a Block	54
TYPICAL PROGRAMMING PROCEDURE	33	Word Order in a Block	55
PART DRAWING	34	GROUPING OF COMMANDS	55
Title Block	34	Group Numbers	55
Dimensioning	34	G-CODE TYPES	56
Tolerances	36	G-Codes and Decimal Point	56
Surface Finish	36		
Drawing Revisions	36		
Special Instructions	37		
METHODS SHEET	37	9 - MISCELLANEOUS FUNCTIONS	57
MATERIAL SPECIFICATIONS	37	DESCRIPTION AND PURPOSE	57
Material Uniformity	37	Machine Related Functions	57
Machinability Rating	37	Program Related Functions	57
MACHINING SEQUENCE	38	TYPICAL APPLICATIONS	58
TOOLING SELECTION	38	Applications for Milling	58
PART SETUP	39	Applications for Turning	58
Setup Sheet	39	Special MDI Functions	58
TECHNOLOGICAL DECISIONS	39	Application Groups	58
Cutter Path	39	M-FUNCTIONS IN A BLOCK	59
Machine Power Rating	40	Startup of M-Functions	60
Coolants and Lubricants	40	Duration of M-Functions	60
WORK SKETCH AND CALCULATIONS	41	PROGRAM FUNCTIONS	60
Identification Methods	41	Program Stop	60
QUALITY IN CNC PROGRAMMING	42	Optional Program Stop	61
		Program End	62
		Subprogram End	62
		MACHINE FUNCTIONS	62
		Coolant Functions	62
		Spindle Functions	63
		Gear Range Selection	64
		Machine Accessories	64
7 - PART PROGRAM STRUCTURE	43		
BASIC PROGRAMMING TERMS	43	10 - SEQUENCE BLOCK	65
Character	43	BLOCK STRUCTURE	65
Word	43	Building the Block Structure	65
Block	44	Block Structure for Milling	65
Program	44	PROGRAM IDENTIFICATION	66
PROGRAMMING FORMATS	44	Program Number	66
WORD ADDRESS FORMAT	44	Program Name	66
FORMAT NOTATION	45	SEQUENCE NUMBERS	67
Short Forms	45	Sequence Number Command	67
Milling System Format	46	Sequence Block Format	67
Turning System Format	46	Numbering Increment	68
Multiple Word Addresses	47	Long Programs and Block Numbers	68
SYMBOLS IN CNC PROGRAMMING	47	END OF BLOCK CHARACTER	68
Plus and Minus Sign	48	STARTUP BLOCK OR SAFE BLOCK	69
TYPICAL PROGRAM STRUCTURE	48	PROGRAM COMMENTS	70
Program Structure Benefits	49	CONFLICTING WORDS IN A BLOCK	70
PROGRAM HEADER	50	MODAL PROGRAMMING VALUES	71
Tool Comments	50	EXECUTION PRIORITY	72
8 - PREPARATORY COMMANDS	51		
DESCRIPTION AND PURPOSE	51		

11 - INPUT OF DIMENSIONS 73

IMPERIAL AND METRIC UNITS 73

Comparable Unit Values 74

ABSOLUTE AND INCREMENTAL MODES 74

Preparatory Commands G90 and G91 75

Absolute Data Input - G90 76

Incremental Data Input - G91 76

Combinations in a Single Block 76

DIAMETER PROGRAMMING 77

MINIMUM MOTION INCREMENT 77

FORMAT OF DIMENSIONAL INPUT 77

Full Address Format 78

Zero Suppression 78

Decimal Point Programming 79

Input Comparison 80

CALCULATOR TYPE INPUT 80

12 - SPINDLE CONTROL 81

SPINDLE FUNCTION 81

Spindle Speed Input 81

DIRECTION OF SPINDLE ROTATION 81

Direction for Milling 82

Direction for Turning 82

Direction Specification 83

Spindle Startup 83

SPINDLE STOP 84

SPINDLE ORIENTATION 84

SPINDLE SPEED - R/MIN 85

SPINDLE SPEED - SURFACE 85

Material Machinability 85

Spindle Speed - Imperial Units 86

Spindle Speed - Metric Units 86

CONSTANT SURFACE SPEED 86

Maximum Spindle Speed Setting 88

Part Diameter Calculation in CSS 89

Cutting Speed Calculation 90

13 - FEEDRATE CONTROL 91

FEEDRATE CONTROL 91

FEEDRATE FUNCTION 91

Feedrate per Minute 91

Feedrate per Revolution 92

FEEDRATE SELECTION 92

ACCELERATION AND DECELERATION 92

Exact Stop Command 93

Exact Stop Mode Command 93

Automatic Corner Override 93

Tapping Mode 93

Cutting Mode 94

CONSTANT FEEDRATE 94

Circular Motion Feedrates 94

MAXIMUM FEEDRATE 95

Maximum Feedrate Considerations 95

FEEDHOLD AND OVERRIDE 95

Feedhold Switch 95

Feedrate Override Switch 95

Feedrate Override Functions 96

E ADDRESS IN THREADING 96

14 - TOOL FUNCTION 97

T-FUNCTION FOR MACHINING CENTERS 97

Tool Storage Magazine 97

Fixed Tool Selection 98

Random Memory Tool Selection 98

Registering Tool Numbers 98

Programming Format 99

Empty Tool or Dummy Tool 99

TOOL CHANGE FUNCTION - M06 99

Conditions for Tool Change 99

AUTOMATIC TOOL CHANGER - ATC 100

Typical ATC System 100

Maximum Tool Diameter 101

Maximum Tool Length 101

Maximum Tool Weight 101

ATC Cycle 102

MDI Operation 102

PROGRAMMING THE ATC 102

Single Tool Work 102

Programming Several Tools 103

Keeping Track of Tools 103

Any Tool in Spindle - Not the First 103

First Tool in the Spindle 104

No Tool in the Spindle 105

First Tool in the Spindle with Manual Change 105

No Tool in the Spindle with Manual Change 106

First Tool in the Spindle and an Oversize Tool 106

No Tool in the Spindle and an Oversize Tool 106

T-FUNCTION FOR LATHES 107

Lathe Tool Station 107

Tool Indexing 107

TOOL OFFSET REGISTERS 108

Geometry Offset 108

Wear Offset 109

Wear Offset Adjustment 110

The R and T Settings 110

15 - REFERENCE POINTS 111

REFERENCE POINT GROUPS 111

Reference Point Groups Relationship 112

MACHINE REFERENCE POINT 112

Return to Machine Zero 113

PART REFERENCE POINT 113

Program Zero Selection 113

Program Zero - Machining Centers 114

Program Zero - Lathes 116

TOOL REFERENCE POINT 116

16 - REGISTER COMMANDS 117

POSITION REGISTER COMMAND	117
Position Register Definition	117
Programming Format	117
Tool Position Setting	118
MACHINING CENTERS APPLICATION	118
Tool Set at Machine Zero	118
Tool Set Away from Machine Zero	118
Position Register in Z-axis	119
Programming Example	119
LATHE APPLICATION	119
Tool Setup	120
Three-Tool Setup Groups	120
Center Line Tools Setup	120
External Tools Setup	121
Internal Tools Setup	121
Corner Tip Detail	121
Programming Example	121

17 - POSITION COMPENSATION 123

DESCRIPTION	123
Programming Commands	123
Programming Format	123
Incremental Mode	124
Motion Length Calculation	124
Position Compensation Along the Z-axis	126
Using G47 and G48	126
Face Milling - One Possible Application	126

18 - WORK OFFSETS 127

WORK AREAS AVAILABLE	127
Additional Work Offsets	128
WORK OFFSET DEFAULT AND STARTUP	128
Work Offset Change	129
Z-axis Application	130
HORIZONTAL MACHINE APPLICATION	131
EXTERNAL WORK OFFSETS	132
LATHE APPLICATIONS	132
Types of Offsets	132
Geometry Offset	132
Wear Offset	133
Tool and Offset Numbers	133
TOOL SETUP	133
Center Line Tools	133
Turning Tools	134
Boring Tools	134
Command Point and Tool Work Offset	134

19 - TOOL LENGTH OFFSET 135

GENERAL PRINCIPLES	135
Actual Tool Length	135
Gauge Line	135
Table Top Face	136

TOOL LENGTH OFFSET COMMANDS	136
Distance-To-Go in Z-axis	136

TOOL LENGTH SETUP	137
On-Machine Tool Length Setting	137
Off-Machine Tool Length Setting	138
Tool Length Offset Amount Register	138

Z-AXIS RELATIONSHIPS	138
Preset Tool Length	139
Tool Length by Touch Off	139
Using a Master Tool Length	140
G43-G44 Difference	140

PROGRAMMING FORMATS	141
Tool Length Offset not Available	141
Tool Length Offset and G92	142
Tool Length Offset and G54-G59	143
Tool Length Offset and Multiple Tools	143

CHANGING TOOL LENGTH OFFSET	144
------------------------------------	------------

HORIZONTAL MACHINE APPLICATION	145
---------------------------------------	------------

TOOL LENGTH OFFSET CANCEL	145
----------------------------------	------------

20 - RAPID POSITIONING 147

RAPID TRAVERSE MOTION	147
G00 Command	147
RAPID MOTION TOOLPATH	148
Single Axis Motion	148
Multiaxis Motion	148
Straight Angular Motion in Rapid Mode	150
Reverse Rapid Motion	150

TYPE OF MOTION & TIME COMPARISON	150
---	------------

REDUCTION OF RAPID MOTION RATE	151
---------------------------------------	------------

RAPID MOTION FORMULAS	151
------------------------------	------------

APPROACH TO THE PART	152
-----------------------------	------------

21 - MACHINE ZERO RETURN 153

MACHINE REFERENCE POSITION	153
Machining Centers	153
Lathes	154
Setting the Machine Axes	154
Program Commands	155
Command Group	155
RETURN TO PRIMARY MACHINE ZERO	155
Intermediate Point	155
Absolute and Incremental Mode	156
Return from Z-depth Position	157
Axes Return Required for the ATC	159
Zero Return for CNC Lathes	159

RETURN POSITION CHECK COMMAND	160
--------------------------------------	------------

RETURN FROM MACHINE ZERO POINT	161
---------------------------------------	------------

RETURN TO SECONDARY MACHINE ZERO	162
---	------------

22 - LINEAR INTERPOLATION 163

LINEAR COMMAND	163
-----------------------	------------

Start and End of the Linear Motion	163
Single Axis Linear Interpolation	163
Two Axes Linear Interpolation	164
Three Axis Linear Interpolation	164
PROGRAMMING FORMAT	164
LINEAR FEEDRATE	165
Feedrate Range	165
Individual Axis Feedrate	165
PROGRAMMING EXAMPLE	166
 23 - BLOCK SKIP FUNCTION	 167
TYPICAL APPLICATIONS	167
BLOCK SKIP SYMBOL	167
CONTROL UNIT SETTING	167
BLOCK SKIP AND MODAL COMMANDS	168
PROGRAMMING EXAMPLES	169
Variable Stock Removal	169
Machining Pattern Change	170
Trial Cut for Measuring	172
Program Proving	173
Barfeeder Application	174
Numbered Block Skip	174
 24 - DWELL COMMAND	 175
PROGRAMMING APPLICATIONS	175
Applications for Cutting	175
Applications for Accessories	175
DWELL COMMAND	175
Dwell Command Structure	176
DWELL TIME SELECTION	176
SETTING MODE AND DWELL	177
Time Setting	177
Number of Revolutions Setting	177
MINIMUM DWELL	177
Practical Considerations	177
NUMBER OF REVOLUTIONS	178
System Setting	178
Time Equivalent	178
LONG DWELL TIME	179
Machine Warm-Up	180
X Axis is the Dwelling Axis	180
Safety and Dwell	180
FIXED CYCLES AND DWELL	180
 25 - FIXED CYCLES	 181
POINT-TO-POINT MACHINING	181
Single Tool Motions vs. Fixed Cycles	182
Basic Concept	182
FIXED CYCLE SELECTION	182
PROGRAMMING FORMAT	183

GENERAL RULES	184
ABSOLUTE AND INCREMENTAL INPUT	185
INITIAL LEVEL SELECTION	185
R-LEVEL SELECTION	186
Z-DEPTH CALCULATIONS	186
DESCRIPTION OF FIXED CYCLES	187
G81 - Drilling Cycle	187
G82 - Spot Drilling Cycle (Drilling with Dwell)	188
G83 - Deep Hole Drilling Cycle - Standard	188
G73 - Deep Hole Drilling Cycle - High Speed	188
G84 - Tapping Cycle - Standard	190
G74 - Tapping Cycle - Reverse	190
G85 - Boring Cycle	191
G86 - Boring Cycle	191
G87 - Backboring Cycle	192
G88 - Boring Cycle	192
G89 - Boring Cycle	193
G76 - Precision Boring Cycle	193
FIXED CYCLE CANCELLATION	194
FIXED CYCLE REPETITION	194
The L or K Address	194
L0 or K0 in a Cycle	195
RIGID TAPPING	195
Comparison - Standard vs. Rigid Tapping	195
Rigid Tapping - Fixed Cycles	195
Rigid Pecking Cycle	196
Cancellation	196
 26 - MACHINING HOLES	 197
SINGLE HOLE EVALUATION	197
Tooling Selection and Applications	197
Program Data	200
DRILLING OPERATIONS	200
Types of Drilling Operations	200
Types of Drills	200
Programming Considerations	201
Nominal Drill Diameter	201
Effective Drill Diameter	201
Drill Point Length	201
Center Drilling	202
Spot Drilling	203
Blind Hole Drilling	203
Through Hole Drilling	204
Flat Bottom Holes	204
Indexable Insert Drilling	205
PECK DRILLING	206
Typical Peck Drilling Application	206
Calculating the Number of Pecks	207
Selecting the Number of Pecks	207
Controlling Breakthrough Depth	207
REAMING	208
Reamer Design	208
Spindle Speeds for Reaming	209
Feedrates for Reaming	209
Stock Allowance	209
Other Reaming Considerations	209
SINGLE POINT BORING	209

Single Point Boring Tool	209
Spindle Orientation	210
Block Tools	211
BORING WITH A TOOL SHIFT	211
Precision Boring Cycle G76	211
Backboring Cycle G87	211
Programming Example	212
Precautions in Programming and Setup	212
ENLARGING HOLES	212
Countersinking	213
Maximum Spot Drill Depth for CSINK	214
Counterboring	214
Spotfacing	214
MULTI Z-DEPTH DRILLING	215
WEB DRILLING	216
TAPPING	216
Tap Geometry	217
Tapping Speed and Feedrate	218
Pipe Taps	219
Tapping Check List	219
HOLE OPERATIONS ON A LATHE	220
Tool Approach Motion	221
Tool Return Motion	221
Drilling and Reaming on Lathes	222
Peck Drilling Cycle - G74	222
Tapping on Lathes	223
Other Operations	224
27 - PATTERN OF HOLES	225
TYPICAL HOLE PATTERNS	225
RANDOM HOLE PATTERN	225
STRAIGHT ROW HOLE PATTERN	226
ANGULAR ROW HOLE PATTERN	226
Pattern Defined by Coordinates	226
Pattern Defined by Angle	227
CORNER PATTERN	228
GRID PATTERN	228
Angular Grid Pattern	229
ARC HOLE PATTERN	230
BOLT HOLE CIRCLE PATTERN	231
Bolt Circle Formula	232
Pattern Orientation	232
POLAR COORDINATE SYSTEM	233
Plane Selection	234
Order of Machining	234
28 - FACE MILLING	235
FACE MILL SELECTION	235
Basic Selection Criteria	235
Face Mill Diameter	235
Insert Geometry	236
CUTTING CONSIDERATIONS	236
Angle of Entry	237

Milling Mode	237
Number of Cutting Inserts	238
PROGRAMMING TECHNIQUES	238
Single Face Mill Cut	239
Multiple Face Mill Cuts	240
USING POSITION COMPENSATION	241
29 - CIRCULAR INTERPOLATION	243
ELEMENTS OF A CIRCLE	243
Radius and Diameter	243
Circle Area and Circumference	244
QUADRANTS	244
Quadrant Points	244
PROGRAMMING FORMAT	245
Arc Cutting Direction	245
Circular Interpolation Block	245
Arc Start and End Points	246
Arc Center and Radius	246
Arc Center Vectors	246
Arc in Planes	247
RADIUS PROGRAMMING	248
Blend Radius	248
Partial Radius	248
FULL CIRCLE PROGRAMMING	248
Boss Milling	250
Internal Circle Cutting - Linear Start	251
Internal Circle Cutting - Circular Start	251
Circle Cutting Cycle	252
ARC PROGRAMMING	253
FEEDRATE FOR CIRCULAR MOTION	253
Feedrate for Outside Arcs	254
Feedrate for Inside Arcs	254
30 - CUTTER RADIUS OFFSET	255
MANUAL CALCULATIONS	255
Cutter Path Center Points	256
Cutter Radius	257
Center Points Calculation	257
COMPENSATED CUTTER PATH	258
Types of Cutter Radius Offset	258
Definition and Applications	258
PROGRAMMING TECHNIQUES	259
Direction of Cutting Motion	259
Left or Right - not CW or CCW	259
Offset Commands	260
Radius of the Cutter	260
History of Offset Types	261
Programming Format	261
Address H or D ?	261
Geometry and Wear Offsets	262
APPLYING CUTTER RADIUS OFFSET	262
Startup Methods	262
Offset Cancellation	264
Cutter Direction Change	265

HOW CUTTER RADIUS OFFSET WORKS	265
Look-Ahead Offset Type	265
Rules for Look-Ahead Cutter Radius Offset	266
Radius of the Cutter	267
Radius Offset Interference	268
Single vs. Multi-axis Startup	269
OVERVIEW OF GENERAL RULES	270
PRACTICAL EXAMPLE - MILLING	271
Part Tolerances	271
Measured Part Size	271
Programmed Offsets	272
D-offset Amount - General Setting	272
Offset Adjustment	273
Offsets for External Cutting	273
Offsets for Internal Cutting	273
One Offset or Multiple Offsets?	274
Preventing a Scrap	274
Program Data - Nominal or Middle?	274
TOOL NOSE RADIUS OFFSET	275
Tool Nose Tip	275
Radius Offset Commands	275
Tool Tip Orientation	275
Effect of Tool Nose Radius Offset	276
Sample Program	276
Minimum Clearance Required	277
Retraction from a Bored Hole	277
Change of Motion Direction	278
31 - PLANE SELECTION	279
WHAT IS A PLANE?	279
MACHINING IN PLANES	279
Mathematical Planes	279
Machine Tool Planes	280
Program Commands for Planes Definition	280
Default Control Status	281
STRAIGHT MOTION IN PLANES	281
CIRCULAR INTERPOLATION IN PLANES	281
G17-G18-G19 as Modal Commands	282
Absence of Axis Data in a Block	282
Cutter Radius Offset in Planes	283
PRACTICAL EXAMPLE	283
FIXED CYCLES IN PLANES	284
32 - CONTOUR MILLING	285
END MILLS	285
High Speed Steel End Mills	286
Solid Carbide End Mills	286
Indexable Insert End Mills	286
Relief Angles	286
End Mill Size	286
Number of Flutes	287
SPEEDS AND FEEDS	287
Coolants and Lubricants	288
Tool Chatter	288
STOCK REMOVAL	289
Plunge Infeed	289

In and Out Ramping	289
Direction of Cut	289
Width and Depth of Cut	290
CORNER RADIUS CALCULATION	290
33 - SLOTS AND POCKETS	291
OPEN AND CLOSED BOUNDARY	291
Open Boundary	291
Closed Boundary	291
PROGRAMMING SLOTS	291
Open Slot Example	291
Closed Slot Example	293
POCKET MILLING	294
General Principles	294
Pocket Types	295
RECTANGULAR POCKETS	295
Stock Amount	296
Stepover Amount	296
Length of Cut	297
Semifinishing Motions	297
Finishing Toolpath	297
Rectangular Pocket Program Example	298
CIRCULAR POCKETS	299
Minimum Cutter Diameter	299
Method of Entry	299
Linear Approach	299
Linear and Circular Approach	300
Roughing a Circular Pocket	301
CIRCULAR POCKET CYCLES	302
34 - TURNING AND BORING	303
TOOL FUNCTION - TURNING	303
T-Address	303
LATHE OFFSETS	304
Offset Entry	304
Independent Tool Offset	304
Tool Offset with Motion	305
Offset Change	305
MULTIPLE OFFSETS	305
General Approach	306
Diameter Tolerances	306
Shoulder Tolerances	307
Diameter and Shoulder Tolerances	307
OFFSET SETTING	308
FUNCTIONS FOR GEAR RANGES	308
AUTOMATIC CORNER BREAK	309
Chamfering at 45 Degrees	309
Blend Radius at 90 Degrees	311
Programming Conditions	311
Programming Example	311
ROUGH AND FINISHED SHAPE	312
Rough Operations	312
Finish Operations	313
Stock and Stock Allowance	313

PROGRAMMING A RECESS	314
SPINDLE SPEED IN CSS MODE	315
LATHE PROGRAM FORMAT	316
Program Format - Templates	316
General Program Format	316
Approach to the Part	316

35 - LATHE CYCLES 317

STOCK REMOVAL ON LATHES	317
Simple Cycles	317
Complex Cycles	317
PRINCIPLES OF LATHE CYCLES	317
G90 - STRAIGHT CUTTING CYCLE	318
Cycle Format	318
Straight Turning Example	319
Taper Cutting Example	319
Straight and Taper Cutting Example	321
G94 - FACE CUTTING CYCLE	322
Cycle Format	322
MULTIPLE REPETITIVE CYCLES	323
General Description	323
Cycle Format Types	323
Cutting Cycles and Part Contour	323
Chipbreaking Cycles	323
CONTOUR CUTTING CYCLES	324
Boundary Definition	324
Start Point and the Points P and Q	324
TYPE I AND TYPE II CYCLES	325
Programming Type I and Type II Cycles	325
Cycle Formatting	325
G71 - STOCK REMOVAL IN TURNING	325
G71 Cycle Format - 6T/10T/11T/15T	326
G71 Cycle Format - 0T/16T/18T/20T/21T	326
G71 for External Roughing	326
G71 for Internal Roughing	327
Direction of Cutting in G71	327
G72 - STOCK REMOVAL IN FACING	327
G72 Cycle Format - 6T/10T/11T/15T	327
G72 Cycle Format - 0T/16T/18T/20T/21T	328
G73 - PATTERN REPEATING CYCLE	328
G73 Cycle Format - 6T/10T/11T/15T	328
G73 Cycle Format - 0T/16T/18T/20T/21T	329
G73 Example of Pattern Repeating	329
G70 - CONTOUR FINISHING CYCLE	330
G70 Cycle Format - All Controls	330
BASIC RULES FOR G70-G73 CYCLES	331
G74 - PECK DRILLING CYCLE	331
G74 Cycle Format - 6T/10T/11T/15T	331
G74 Cycle Format - 0T/16T/18T/20T/21T	331
G75 - GROOVE CUTTING CYCLE	332
G75 Cycle Format - 6T/10T/11T/15T	332
G75 Cycle Format - 0T/16T/18T/20T/21T	332
BASIC RULES FOR G74 AND G75 CYCLES	332

36 - GROOVING ON LATHES 333

GROOVING OPERATIONS	333
Main Grooving Applications	333
Grooving Criteria	333
GROOVE SHAPE	333
Nominal Insert Size	334
Insert Modification	334
GROOVE LOCATION	334
GROOVE DIMENSIONS	334
Groove Position	335
Groove Depth	335
SIMPLE GROOVE PROGRAMMING	335
PRECISION GROOVING TECHNIQUES	336
Groove Width Selection	337
Machining Method	337
Finishing Allowances	338
Groove Tolerances	339
Groove Surface Finish	340
MULTIPLE GROOVES	340
FACE GROOVES	340
Radial Clearance	341
Face Grooving Program Example	341
CORNER GROOVES / NECK GROOVES	342
GROOVING CYCLES	342
G75 Cycle Applications	342
Single Groove with G75	343
Multiple Grooves with G75	343
SPECIAL GROOVES	343
GROOVES AND SUBPROGRAMS	344

37 - PART-OFF 345

PART-OFF PROCEDURE	345
Parting Tool Description	345
Tool Approach Motion	346
Stock Allowance	347
Tool Return Motion	347
Part-off with a Chamfer	347
Preventing Damage to the Part	348

38 - SINGLE POINT THREADING 349

THREADING ON CNC LATHES	349
Form of a Thread	349
Threading Operations	350
TERMINOLOGY OF THREADING	350
THREADING PROCESS	351
Steps in Threading	351
Spindle Synchronization	351
Individual Threading Motions	351
Thread Start Position	353
Thread Cutting Diameter and Depth	353
Number of Threading Passes	353
Thread Depth Calculation	354

Thread Depth Constants	354
Calculation of Threading Passes	355
Thread Cutting Motion	356
Retract from Thread	357
Return to Start Position	357
THREADING FEED AND SPINDLE SPEED	357
Threading Feedrate Selection	358
Spindle Speed Selection	359
Maximum Threading Feedrate	359
Lead Error	360
Rounding Error	360
E-address	361
TOOL REFERENCE POINT	361
BLOCK-BY-BLOCK THREADING	361
BASIC THREADING CYCLE - G92	362
MULTIPLE REPETITIVE CYCLE - G76	363
G76 Cycle Format - One Block Format	364
G76 Cycle Format - Two Block Format	364
One-Block vs Two-Block Format	365
Programming Examples	365
First Thread Diameter Calculation	366
THREAD INFEEED METHODS	366
Radial Infeed (Straight Infeed)	367
Compound Infeed (Flank Infeed)	367
Thread Insert Angle	367
Thread Cutting Type - Address P	368
COMPOUND INFEEED CALCULATIONS	368
Initial Considerations	368
Z-axis Start Position Calculation	369
THREAD RETRACT MOTION	370
Thread Pullout Functions	370
Single Axis Pullout	371
Two-Axis Pullout	371
HAND OF THREAD	371
Configurations for Hand of Thread	372
THREADING TO A SHOULDER	373
Insert Modification	373
Program Testing	375
OTHER THREAD FORMS	375
Thread Depth - ACME example	375
TAPERED THREAD	376
Depth and Clearances	376
Taper Calculation	376
Block by Block Taper Thread	377
Tapered Thread Using a Simple Cycle	378
Tapered Thread and G76 Cycle	378
MULTISTART THREAD	379
Threading Feedrate Calculation	380
Shift Amount	380
Application Example	381
Quality of Multistart Threads	381
THREAD RECUTTING	382
Before Part Removal	382
After Part Removal	382

39 - SUBPROGRAMS	383
MAIN PROGRAM AND SUBPROGRAMS	383
Subprogram Benefits	383
Identification of Subprograms	384
SUBPROGRAM FUNCTIONS	384
Subprogram Call Function	384
Subprogram End Function	384
Block Number to Return to	385
Number of Subprogram Repetitions	385
L0/K0 in a Subprogram Call	386
SUBPROGRAM NUMBERING	387
Organized Approach	388
Protected Subprograms	389
SUBPROGRAM DEVELOPMENT	389
Repeating Pattern Recognition	389
Tool Motion and Subprograms	390
Modal Values and Subprograms	391
MULTI LEVEL NESTING	392
One Level Nesting	392
Two Level Nesting	392
Three Level Nesting	393
Four Level Nesting	393
Nesting Applications	393
CONTOURING WITH A SUBPROGRAM	394
TOOL CHANGE SUBPROGRAM	395
100 000 000 HOLE GRID	395
40 - DATUM SHIFT	397
DATUM SHIFT WITH G92 OR G50	397
Program Zero Shift	397
LOCAL COORDINATE SYSTEM	399
G52 Command	400
MACHINE COORDINATE SYSTEM	400
DATA SETTING	402
Data Setting Command	402
Coordinate Mode	402
WORK OFFSETS	402
Standard Work Offset Input	402
Additional Work Offset Input	403
External Work Offset Input	403
TOOL LENGTH OFFSETS	403
Valid Input Range	404
CUTTER RADIUS OFFSETS	404
LATHE OFFSETS	404
MDI DATA SETTING	405
PROGRAMMABLE PARAMETER ENTRY	405
Modal G10 Command	405
Parameters Notation	406
Program Portability	406
Bit Type Parameter	407
Effect of Block Numbers	408

41 - MIRROR IMAGE 409**BASIC RULES OF MIRROR IMAGE 409**

Toolpath Direction	409
Original Toolpath	410
Mirror Axis	410
Sign of Coordinates	410
Milling Direction	411
Arc Motion Direction	411
Program Start and End	411

MIRROR IMAGE BY SETTING 411

Control Setting	411
Programming - Manual Mirror Setting	412

PROGRAMMABLE MIRROR IMAGE 412

Mirror Image Functions	412
Simple Mirror Image Example	413
Complete Mirror Image Example	414

MIRROR IMAGE ON CNC LATHES 414**42 - COORDINATE ROTATION 415****ROTATION COMMANDS 415**

Center of Rotation	415
Radius of Rotation	415
Coordinate Rotation Cancel	417
Common Applications	417

PRACTICAL APPLICATION 417**43 - SCALING FUNCTION 421****DESCRIPTION 421**

Scaling Function Usage	421
------------------------	-----

PROGRAMMING FORMAT 421

Scaling Center	422
Scaling Factor	422
Rounding Errors in Scaling	423

PROGRAM EXAMPLES 423**44 - CNC LATHE ACCESSORIES 425****CHUCK CONTROL 425**

Chuck Functions	425
Chuck Pressure	426
Chuck Jaws	426

TAILSTOCK AND QUILL 426

Tailstock Body	426
Quill	427
Center	427
Quill Functions	427
Programmable Tailstock	427
Safety Concerns	427

BI-DIRECTIONAL TURRET INDEXING 428

Programming Example	428
---------------------	-----

BARFEEDER ATTACHMENT 429

Bar Stopper	429
-------------	-----

ADDITIONAL OPTIONS 430

Part Catcher or Part Unloader	430
Continuous Operation	430
Parts Counter	431
Pull-Out Finger	431

PROGRAMMING EXAMPLE 431**45 - HELICAL MILLING 433****HELICAL MILLING OPERATION 433**

Helical Interpolation	433
Programming Format	433
Arc Vectors for Helical Interpolation	434
Applications and Usage	434

THREAD MILLING 434

Applying Thread Milling	434
Conditions for Thread Milling	434
Thread Milling Tool	435
Premachining Requirements	435
Clearance Radius	435
Productivity of Thread Milling	435

THE HELIX 435**THREAD MILLING EXAMPLE 437**

Straight Thread	437
Initial Calculations	437
Starting Position	438
Motion Direction and Direction	438
Lead-In Motions	440
Thread Rise Calculation	441
Milling the Thread	441
Lead-Out Motions	442
Complete Program	442
External Thread Milling	442
Tapered Thread Milling	442
Further Considerations	443

THREAD MILLING SIMULATION METHOD 443**HELICAL RAMPING 444****46 - HORIZONTAL MACHINING 445****INDEXING AND ROTARY AXES 445****INDEXING TABLE (B-AXIS) 445**

Units of Increment	445
Direction of Indexing	446
Table Clamp and Unclamp Functions	446
Indexing in Absolute and Incremental Mode	446

B-AXIS AND OFFSETS 447

Work Offset and B-axis	447
Tool Length Offset and B-axis	448

RETURN TO MACHINE ZERO 450**INDEXING AND A SUBPROGRAM 450****COMPLETE PROGRAM EXAMPLE 452****AUTOMATIC PALLET CHANGER - APC 453**

Working Environment	454
Types of Pallets	454
Programming Commands	455
Pallet Changing Program Structure	455

HORIZONTAL BORING MILL	455
47 - LIVE TOOLING ON LATHES	457
TURN-MILL OR MILL-TURN	457
Programming Issues	457
General Terms	457
MACHINE DESIGN	458
Features	458
Benefits	458
C-AXIS PROGRAMMING	458
XZ + C Axes	458
FIXED CYCLES	459
M-functions and C-axis	460
Drilling on Face	460
Drilling on Diameter	461
General Considerations	461
Y-AXIS PROGRAMMING	462
Plane Selection	463
Additional Axes	463
POLAR COORDINATE INTERPOLATION	463
Polar Coordinate Mode	463
Approximation Method	464
CYLINDRICAL INTERPOLATION	465
Z + C-axis	465
Practical Example	465
48 - WRITING A CNC PROGRAM	467
PROGRAM WRITING	467
Legibility of Handwriting	468
Programming Forms	468
Confusing Characters	468
PROGRAM OUTPUT FORMATTING	469
LONG PROGRAMS	471
Program Length Reduction	471
Memory Mode and Tape Mode	472
49 - PROGRAM DOCUMENTS	473
DATA FILES	473
PROGRAM DOCUMENTATION	474
External Documentation	474
Internal Documentation	474
Program Description	475
SETUP AND TOOLING SHEETS	475
Setup Sheet	476
Tooling Sheet	476
Coordinate Sheet	477
DOCUMENTATION FILE FOLDER	477
Identification Methods	477
Operator's Suggestions	478
Filing and Storage	478

50 - PROGRAM VERIFICATION	479
DETECTION OF ERRORS	479
Preventive Measures	479
Corrective Measures	479
GRAPHIC VERIFICATION	480
AVOIDING ERRORS	480
Syntax Errors	481
Logical Errors	481
COMMON PROGRAMMING ERRORS	482
Program Input Errors	482
Calculation Errors	482
Hardware Errors	482
Miscellaneous Errors	482
51 - CNC MACHINING	483
MACHINING A NEW PART	483
Setup Integrity	484
Program Integrity	484
RUNNING THE FIRST PART	485
PROGRAM CHANGES	486
Program Upgrading	486
Program Updating	487
Documentation Change	487
ALTERNATE MACHINE SELECTION	487
MACHINE WARM UP PROGRAM	488
CNC MACHINING AND SAFETY	488
SHUTTING DOWN A CNC MACHINE	489
Emergency Stop Switch	489
Parking Machine Slides	490
Setting the Control System	490
Turning the Power Off	490
EQUIPMENT MAINTENANCE	490
52 - INTERFACING TO DEVICES	491
RS-232C INTERFACE	491
PUNCHED TAPE	492
Tape Reader and Puncher	492
Leader and Trailer	494
Tape Identification	494
Non-printable Characters	494
Storage and Handling	494
DISTRIBUTED NUMERICAL CONTROL	494
TERMINOLOGY OF COMMUNICATIONS	495
Baud Rate	495
Parity	495
Data Bits	495
Start and Stop Bits	495
DATA SETTING	495
CONNECTING CABLES	496
Null Modem	496
Cabling for Fanuc and PC	496

53 - MATH IN CNC PROGRAMMING 497**BASIC ELEMENTS 497**

Arithmetic and Algebra	497
Order of Calculations	497

GEOMETRY 498

Circle	498
PI Constant	499
Circumference of a Circle	499
Length of Arc	499
Quadrants	499

POLYGONS 500**TAPERS 501**

Taper Definition	501
Taper Per Foot	502
Taper Ratio	502
Taper Calculations - Imperial Units	502
Taper Calculations - Metric Units	502

CALCULATIONS OF TRIANGLES 503

Types of Angles and Triangles	503
Right Triangles	504
Similar Triangles	504
Sine - Cosine - Tangent	505
Inverse Trigonometric Functions	506
Degrees and Decimal Degrees	506
Pythagorean Theorem	506
Solving Right Triangles	507

ADVANCED CALCULATIONS 508**CONCLUSION 508****54 - CNC AND CAD/CAM 509****PROGRAMMING MANUALLY ? 509**

CAM Software	509
Desktop Computer Programming	509

TOOLPATH GEOMETRY DEVELOPMENT 510**TOOL PATH GENERATION 510****COMPLETE ENVIRONMENT 510**

Multi Machine Support	511
Associative Operations	511
Job Setup	511
Tooling List and Job Comments	511
Connection Between Computers	511
Program Text Editor	512
Printing Capabilities	512
Pen Plotting	512
CAD Software Access	512
Support for Solids	512
Software Specifications	513
Hardware Specifications	513
Typical Hardware / Software Requirements	514
Utilities and Special Features	514

POST PROCESSORS 514

Customizing Post Processor	514
----------------------------	-----

IMPORTANT FEATURES 515

Input from User	515
Machining Cycles	515
User Interface	515
CAD Interface	515

SUPPORT AND MANAGEMENT 515

Training	516
System Management	516
Technical Support	516

THE END AND THE BEGINNING 516**APPENDICES****A - REFERENCE TABLES 517****Decimal Equivalents 517****Imperial Threads - UNC/UNF 520****Straight Pipe Taps NPS 520****Taper Pipe Taps NPT 521****Metric Coarse Threads 521****Metric Fine Threads 521****B - SPEEDS AND FEEDS 523****SURFACE SPEED AND SPINDLE SPEED 523****CHIPLOAD 524****BASIC FORMULAS 524****RELATED FORMULAS 524**

Tapping Feedrate	524
Z-depth for Spot Drilling	524
Z-depth for Drilling (Through Hole)	525
Dwell Time Calculation	525
Cutting Time	525

C - NCPlot™ FEATURES 527**Full Featured Text Editor 527****Program Formatting Tools 527****Program Translation Tools 527****Advanced G-code Backplotter 527****Flexible Viewport Controls 528****View Controls 528****Calc Tools 528****DXF Importing / Exporting 528****Other Tools 528****Index 529**

Note that downloadable files mentioned in the text and provided to accompany this book, previously were provided on a CD-ROM. For a direct download link for these files, please see the product page for the *CNC Programming Handbook* at the Industrial Press websites <https://books.industrialpress.com> or <https://ebooks.industrialpress.com>. Note that the trial version of NCPlot™ included with these files may not work on current operating systems, newer versions may be found online at <https://www.ncplot.com>.