

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	9 - MISCELLANEOUS FUNCTIONS	57
Drawing Revisions	36	DESCRIPTION AND PURPOSE	57
Special Instructions	37	Machine Related Functions	57
METHODS SHEET	37	Program Related Functions	57
MATERIAL SPECIFICATIONS	37	TYPICAL APPLICATIONS	58
Material Uniformity	37	Applications for Milling	58
Machinability Rating	37	Applications for Turning	58
MACHINING SEQUENCE	38	Special MDI Functions	58
TOOLING SELECTION	38	Application Groups	58
PART SETUP	39	M-FUNCTIONS IN A BLOCK	59
Setup Sheet	39	Startup of M-Functions	60
TECHNOLOGICAL DECISIONS	39	Duration of M-Functions	60
Cutter Path	39	PROGRAM FUNCTIONS	60
Machine Power Rating	40	Program Stop	60
Coolants and Lubricants	40	Optional Program Stop	61
WORK SKETCH AND CALCULATIONS	41	Program End	62
Identification Methods	41	Subprogram End	62
QUALITY IN CNC PROGRAMMING	42	MACHINE FUNCTIONS	62
7 - PART PROGRAM STRUCTURE	43	Coolant Functions	62
BASIC PROGRAMMING TERMS	43	Spindle Functions	63
Character	43	Gear Range Selection	64
Word	43	Machine Accessories	64
Block	44	10 - SEQUENCE BLOCK	65
Program	44	BLOCK STRUCTURE	65
PROGRAMMING FORMATS	44	Building the Block Structure	65
WORD ADDRESS FORMAT	44	Block Structure for Milling	65
FORMAT NOTATION	45	PROGRAM IDENTIFICATION	66
Short Forms	45	Program Number	66
Milling System Format	46	Program Name	66
Turning System Format	46	SEQUENCE NUMBERS	67
Multiple Word Addresses	47	Sequence Number Command	67
SYMBOLS IN CNC PROGRAMMING	47	Sequence Block Format	67
Plus and Minus Sign	48	Numbering Increment	68
TYPICAL PROGRAM STRUCTURE	48	Long Programs and Block Numbers	68
Program Structure Benefits	49	END OF BLOCK CHARACTER	68
PROGRAM HEADER	50	STARTUP BLOCK OR SAFE BLOCK	69
Tool Comments	50	PROGRAM COMMENTS	70
8 - PREPARATORY COMMANDS	51	CONFLICTING WORDS IN A BLOCK	70
DESCRIPTION AND PURPOSE	51	MODAL PROGRAMMING VALUES	71
		EXECUTION PRIORITY	72

11 - INPUT OF DIMENSIONS	73	MAXIMUM FEEDRATE	95
IMPERIAL AND METRIC UNITS	73	Maximum Feedrate Considerations	95
Comparable Unit Values	74	FEEDHOLD AND OVERRIDE	95
ABSOLUTE AND INCREMENTAL MODES	74	Feedhold Switch	95
Preparatory Commands G90 and G91	75	Feedrate Override Switch	95
Absolute Data Input - G90	76	Feedrate Override Functions	96
Incremental Data Input - G91	76	E ADDRESS IN THREADING	96
Combinations in a Single Block	76	14 - TOOL FUNCTION	97
DIAMETER PROGRAMMING	77	T-FUNCTION FOR MACHINING CENTERS	97
MINIMUM MOTION INCREMENT	77	Tool Storage Magazine	97
FORMAT OF DIMENSIONAL INPUT	77	Fixed Tool Selection	98
Full Address Format	78	Random Memory Tool Selection	98
Zero Suppression	78	Registering Tool Numbers	98
Decimal Point Programming	79	Programming Format	99
Input Comparison	80	Empty Tool or Dummy Tool	99
CALCULATOR TYPE INPUT	80	TOOL CHANGE FUNCTION - M06	99
12 - SPINDLE CONTROL	81	Conditions for Tool Change	99
SPINDLE FUNCTION	81	AUTOMATIC TOOL CHANGER - ATC	100
Spindle Speed Input	81	Typical ATC System	100
DIRECTION OF SPINDLE ROTATION	81	Maximum Tool Diameter	101
Direction for Milling	82	Maximum Tool Length	101
Direction for Turning	82	Maximum Tool Weight	101
Direction Specification	83	ATC Cycle	102
Spindle Startup	83	MDI Operation	102
SPINDLE STOP	84	PROGRAMMING THE ATC	102
SPINDLE ORIENTATION	84	Single Tool Work	102
SPINDLE SPEED - R/MIN	85	Programming Several Tools	103
SPINDLE SPEED - SURFACE	85	Keeping Track of Tools	103
Material Machinability	85	Any Tool in Spindle - Not the First	103
Spindle Speed - Imperial Units	86	First Tool in the Spindle	104
Spindle Speed - Metric Units	86	No Tool in the Spindle	105
CONSTANT SURFACE SPEED	86	First Tool in the Spindle with Manual Change	105
Maximum Spindle Speed Setting	88	No Tool in the Spindle with Manual Change	106
Part Diameter Calculation in CSS	89	First Tool in the Spindle and an Oversize Tool	106
Cutting Speed Calculation	90	No Tool in the Spindle and an Oversize Tool	106
13 - FEEDRATE CONTROL	91	T-FUNCTION FOR LATHES	107
FEEDRATE CONTROL	91	Lathe Tool Station	107
FEEDRATE FUNCTION	91	Tool Indexing	107
Feedrate per Minute	91	TOOL OFFSET REGISTERS	108
Feedrate per Revolution	92	Geometry Offset	108
FEEDRATE SELECTION	92	Wear Offset	109
ACCELERATION AND DECELERATION	92	Wear Offset Adjustment	110
Exact Stop Command	93	The R and T Settings	110
Exact Stop Mode Command	93	15 - REFERENCE POINTS	111
Automatic Corner Override	93	REFERENCE POINT GROUPS	111
Tapping Mode	93	Reference Point Groups Relationship	112
Cutting Mode	94	MACHINE REFERENCE POINT	112
CONSTANT FEEDRATE	94	Return to Machine Zero	113
Circular Motion Feedrates	94	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	TOOL LENGTH OFFSET COMMANDS	136
POSITION REGISTER COMMAND	117	Distance-To-Go in Z-axis	136
Position Register Definition	117	TOOL LENGTH SETUP	137
Programming Format	117	On-Machine Tool Length Setting	137
Tool Position Setting	118	Off-Machine Tool Length Setting	138
MACHINING CENTERS APPLICATION	118	Tool Length Offset Amount Register	138
Tool Set at Machine Zero	118	Z-AXIS RELATIONSHIPS.	138
Tool Set Away from Machine Zero	118	Preset Tool Length	139
Position Register in Z-axis	119	Tool Length by Touch Off	139
Programming Example	119	Using a Master Tool Length	140
LATHE APPLICATION	119	G43-G44 Difference	140
Tool Setup	120	PROGRAMMING FORMATS	141
Three-Tool Setup Groups	120	Tool Length Offset not Available	141
Center Line Tools Setup	120	Tool Length Offset and G92	142
External Tools Setup	121	Tool Length Offset and G54-G59	143
Internal Tools Setup	121	Tool Length Offset and Multiple Tools	143
Corner Tip Detail	121	CHANGING TOOL LENGTH OFFSET	144
Programming Example	121	HORIZONTAL MACHINE APPLICATION	145
17 - POSITION COMPENSATION	123	TOOL LENGTH OFFSET CANCEL	145
DESCRIPTION	123	20 - RAPID POSITIONING	147
Programming Commands	123	RAPID TRAVERSE MOTION	147
Programming Format	123	G00 Command	147
Incremental Mode	124	RAPID MOTION TOOLPATH.	148
Motion Length Calculation	124	Single Axis Motion	148
Position Compensation Along the Z-axis	126	Multiaxis Motion	148
Using G47 and G48	126	Straight Angular Motion in Rapid Mode	150
Face Milling - One Possible Application	126	Reverse Rapid Motion	150
18 - WORK OFFSETS	127	TYPE OF MOTION & TIME COMPARISON	150
WORK AREAS AVAILABLE	127	REDUCTION OF RAPID MOTION RATE	151
Additional Work Offsets	128	RAPID MOTION FORMULAS	151
WORK OFFSET DEFAULT AND STARTUP	128	APPROACH TO THE PART	152
Work Offset Change	129	21 - MACHINE ZERO RETURN	153
Z-axis Application	130	MACHINE REFERENCE POSITION	153
HORIZONTAL MACHINE APPLICATION	131	Machining Centers	153
EXTERNAL WORK OFFSETS	132	Lathes	154
LATHE APPLICATIONS	132	Setting the Machine Axes	154
Types of Offsets	132	Program Commands	155
Geometry Offset	132	Command Group	155
Wear Offset	133	RETURN TO PRIMARY MACHINE ZERO	155
Tool and Offset Numbers	133	Intermediate Point	155
TOOL SETUP	133	Absolute and Incremental Mode	156
Center Line Tools	133	Return from Z-depth Position	157
Turning Tools	134	Axes Return Required for the ATC	159
Boring Tools	134	Zero Return for CNC Lathes	159
Command Point and Tool Work Offset	134	RETURN POSITION CHECK COMMAND	160
19 - TOOL LENGTH OFFSET	135	RETURN FROM MACHINE ZERO POINT	161
GENERAL PRINCIPLES	135	RETURN TO SECONDARY MACHINE ZERO	162
Actual Tool Length	135	22 - LINEAR INTERPOLATION	163
Gauge Line	135	LINEAR COMMAND	163
Table Top Face	136		

Start and End of the Linear Motion	163	GENERAL RULES	184
Single Axis Linear Interpolation	163	ABSOLUTE AND INCREMENTAL INPUT	185
Two Axes Linear Interpolation	164	INITIAL LEVEL SELECTION	185
Three Axis Linear Interpolation	164	R-LEVEL SELECTION	186
PROGRAMMING FORMAT	164	Z-DEPTH CALCULATIONS	186
LINEAR FEEDRATE	165	DESCRIPTION OF FIXED CYCLES	187
Feedrate Range	165	G81 - Drilling Cycle	187
Individual Axis Feedrate	165	G82 - Spot Drilling Cycle (Drilling with Dwell)	188
PROGRAMMING EXAMPLE	166	G83 - Deep Hole Drilling Cycle - Standard	188
23 - BLOCK SKIP FUNCTION	167	G73 - Deep Hole Drilling Cycle - High Speed	188
TYPICAL APPLICATIONS	167	G84 - Tapping Cycle - Standard	190
BLOCK SKIP SYMBOL	167	G74 - Tapping Cycle - Reverse	190
CONTROL UNIT SETTING	167	G85 - Boring Cycle	191
BLOCK SKIP AND MODAL COMMANDS	168	G86 - Boring Cycle	191
PROGRAMMING EXAMPLES	169	G87 - Backboring Cycle	192
Variable Stock Removal	169	G88 - Boring Cycle	192
Machining Pattern Change	170	G89 - Boring Cycle	193
Trial Cut for Measuring	172	G76 - Precision Boring Cycle	193
Program Proving	173	FIXED CYCLE CANCELLATION	194
Barfeeder Application	174	FIXED CYCLE REPETITION	194
Numbered Block Skip	174	The L or K Address	194
24 - DWELL COMMAND	175	L0 or K0 in a Cycle	195
PROGRAMMING APPLICATIONS	175	RIGID TAPPING	195
Applications for Cutting	175	Comparison - Standard vs. Rigid Tapping	195
Applications for Accessories	175	Rigid Tapping - Fixed Cycles	195
DWELL COMMAND	175	Rigid Pecking Cycle	196
Dwell Command Structure	176	Cancellation	196
DWELL TIME SELECTION	176	26 - MACHINING HOLES	197
SETTING MODE AND DWELL	177	SINGLE HOLE EVALUATION	197
Time Setting	177	Tooling Selection and Applications	197
Number of Revolutions Setting	177	Program Data	200
MINIMUM DWELL	177	DRILLING OPERATIONS	200
Practical Considerations	177	Types of Drilling Operations	200
NUMBER OF REVOLUTIONS	178	Types of Drills	200
System Setting	178	Programming Considerations	201
Time Equivalent	178	Nominal Drill Diameter	201
LONG DWELL TIME	179	Effective Drill Diameter	201
Machine Warm-Up	180	Drill Point Length	201
X Axis is the Dwelling Axis	180	Center Drilling	202
Safety and Dwell	180	Spot Drilling	203
FIXED CYCLES AND DWELL	180	Blind Hole Drilling	203
25 - FIXED CYCLES	181	Through Hole Drilling	204
POINT-TO-POINT MACHINING	181	Flat Bottom Holes	204
Single Tool Motions vs. Fixed Cycles	182	Indexable Insert Drilling	205
Basic Concept	182	PECK DRILLING	206
FIXED CYCLE SELECTION	182	Typical Peck Drilling Application	206
PROGRAMMING FORMAT	183	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	Milling Mode	237
Spindle Orientation	210	Number of Cutting Inserts	238
Block Tools	211	PROGRAMMING TECHNIQUES	238
BORING WITH A TOOL SHIFT	211	Single Face Mill Cut	239
Precision Boring Cycle G76	211	Multiple Face Mill Cuts	240
Backboring Cycle G87	211	USING POSITION COMPENSATION	241
Programming Example	212	29 - CIRCULAR INTERPOLATION	243
Precautions in Programming and Setup	212	ELEMENTS OF A CIRCLE	243
ENLARGING HOLES	212	Radius and Diameter	243
Countersinking	213	Circle Area and Circumference	244
Maximum Spot Drill Depth for CSINK	214	QUADRANTS	244
Counterboring	214	Quadrant Points	244
Spotfacing	214	PROGRAMMING FORMAT	245
MULTI Z-DEPTH DRILLING	215	Arc Cutting Direction	245
WEB DRILLING	216	Circular Interpolation Block	245
TAPPING	216	Arc Start and End Points	246
Tap Geometry	217	Arc Center and Radius	246
Tapping Speed and Feedrate	218	Arc Center Vectors	246
Pipe Taps	219	Arc in Planes	247
Tapping Check List	219	RADIUS PROGRAMMING	248
HOLE OPERATIONS ON A LATHE	220	Blend Radius	248
Tool Approach Motion	221	Partial Radius	248
Tool Return Motion	221	FULL CIRCLE PROGRAMMING	248
Drilling and Reaming on Lathes	222	Boss Milling	250
Peck Drilling Cycle - G74	222	Internal Circle Cutting - Linear Start	251
Tapping on Lathes	223	Internal Circle Cutting - Circular Start	251
Other Operations	224	Circle Cutting Cycle	252
27 - PATTERN OF HOLES	225	ARC PROGRAMMING	253
TYPICAL HOLE PATTERNS	225	FEEDRATE FOR CIRCULAR MOTION	253
RANDOM HOLE PATTERN	225	Feedrate for Outside Arcs	254
STRAIGHT ROW HOLE PATTERN	226	Feedrate for Inside Arcs	254
ANGULAR ROW HOLE PATTERN	226	30 - CUTTER RADIUS OFFSET	255
Pattern Defined by Coordinates	226	MANUAL CALCULATIONS	255
Pattern Defined by Angle	227	Cutter Path Center Points	256
CORNER PATTERN	228	Cutter Radius	257
GRID PATTERN	228	Center Points Calculation	257
Angular Grid Pattern	229	COMPENSATED CUTTER PATH	258
ARC HOLE PATTERN	230	Types of Cutter Radius Offset	258
BOLT HOLE CIRCLE PATTERN	231	Definition and Applications	258
Bolt Circle Formula	232	PROGRAMMING TECHNIQUES	259
Pattern Orientation	232	Direction of Cutting Motion	259
POLAR COORDINATE SYSTEM	233	Left or Right - not CW or CCW	259
Plane Selection	234	Offset Commands	260
Order of Machining	234	Radius of the Cutter	260
28 - FACE MILLING	235	History of Offset Types	261
FACE MILL SELECTION	235	Programming Format	261
Basic Selection Criteria	235	Address H or D ?	261
Face Mill Diameter	235	Geometry and Wear Offsets	262
Insert Geometry	236	APPLYING CUTTER RADIUS OFFSET	262
CUTTING CONSIDERATIONS	236	Startup Methods	262
Angle of Entry	237	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. Multiaxis 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	36 - GROOVING ON LATHES	333
SPINDLE SPEED IN CSS MODE	315	GROOVING OPERATIONS	333
LATHE PROGRAM FORMAT	316	Main Grooving Applications	333
Program Format - Templates	316	Grooving Criteria	333
General Program Format	316	GROOVE SHAPE	333
Approach to the Part	316	Nominal Insert Size	334
35 - LATHE CYCLES	317	Insert Modification	334
STOCK REMOVAL ON LATHES	317	GROOVE LOCATION	334
Simple Cycles	317	GROOVE DIMENSIONS	334
Complex Cycles	317	Groove Position	335
PRINCIPLES OF LATHE CYCLES	317	Groove Depth	335
G90 - STRAIGHT CUTTING CYCLE	318	SIMPLE GROOVE PROGRAMMING	335
Cycle Format	318	PRECISION GROOVING TECHNIQUES	336
Straight Turning Example	319	Groove Width Selection	337
Taper Cutting Example	319	Machining Method	337
Straight and Taper Cutting Example	321	Finishing Allowances	338
G94 - FACE CUTTING CYCLE	322	Groove Tolerances	339
Cycle Format	322	Groove Surface Finish	340
MULTIPLE REPETITIVE CYCLES	323	MULTIPLE GROOVES	340
General Description	323	FACE GROOVES	340
Cycle Format Types	323	Radial Clearance	341
Cutting Cycles and Part Contour	323	Face Grooving Program Example	341
Chipbreaking Cycles	323	CORNER GROOVES / NECK GROOVES	342
CONTOUR CUTTING CYCLES	324	GROOVING CYCLES	342
Boundary Definition	324	G75 Cycle Applications	342
Start Point and the Points P and Q	324	Single Groove with G75	343
TYPE I AND TYPE II CYCLES	325	Multiple Grooves with G75	343
Programming Type I and Type II Cycles	325	SPECIAL GROOVES	343
Cycle Formatting	325	GROOVES AND SUBPROGRAMS	344
G71 - STOCK REMOVAL IN TURNING	325	37 - PART-OFF	345
G71 Cycle Format - 6T/10T/11T/15T	326	PART-OFF PROCEDURE	345
G71 Cycle Format - 0T/16T/18T/20T/21T	326	Parting Tool Description	345
G71 for External Roughing	326	Tool Approach Motion	346
G71 for Internal Roughing	327	Stock Allowance	347
Direction of Cutting in G71	327	Tool Return Motion	347
G72 - STOCK REMOVAL IN FACING	327	Part-off with a Chamfer	347
G72 Cycle Format - 6T/10T/11T/15T	327	Preventing Damage to the Part	348
G72 Cycle Format - 0T/16T/18T/20T/21T	328	38 - SINGLE POINT THREADING	349
G73 - PATTERN REPEATING CYCLE	328	THREADING ON CNC LATHES	349
G73 Cycle Format - 6T/10T/11T/15T	328	Form of a Thread	349
G73 Cycle Format - 0T/16T/18T/20T/21T	329	Threading Operations	350
G73 Example of Pattern Repeating	329	TERMINOLOGY OF THREADING	350
G70 - CONTOUR FINISHING CYCLE	330	THREADING PROCESS	351
G70 Cycle Format - All Controls	330	Steps in Threading	351
BASIC RULES FOR G70-G73 CYCLES	331	Spindle Synchronization	351
G74 - PECK DRILLING CYCLE	331	Individual Threading Motions	351
G74 Cycle Format - 6T/10T/11T/15T	331	Thread Start Position	353
G74 Cycle Format - 0T/16T/18T/20T/21T	331	Thread Cutting Diameter and Depth	353
G75 - GROOVE CUTTING CYCLE	332	Number of Threading Passes	353
G75 Cycle Format - 6T/10T/11T/15T	332	Thread Depth Calculation	354
G75 Cycle Format - 0T/16T/18T/20T/21T	332		
BASIC RULES FOR G74 AND G75 CYCLES	332		

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	ADDITIONAL OPTIONS	430
BASIC RULES OF MIRROR IMAGE	409	Part Catcher or Part Unloader	430
Toolpath Direction	409	Continuous Operation	430
Original Toolpath	410	Parts Counter	431
Mirror Axis	410	Pull-Out Finger	431
Sign of Coordinates	410	PROGRAMMING EXAMPLE	431
Milling Direction	411	45 - HELICAL MILLING	433
Arc Motion Direction	411	HELICAL MILLING OPERATION	433
Program Start and End	411	Helical Interpolation	433
MIRROR IMAGE BY SETTING	411	Programming Format	433
Control Setting	411	Arc Vectors for Helical Interpolation	434
Programming - Manual Mirror Setting	412	Applications and Usage	434
PROGRAMMABLE MIRROR IMAGE	412	THREAD MILLING	434
Mirror Image Functions	412	Applying Thread Milling	434
Simple Mirror Image Example	413	Conditions for Thread Milling	434
Complete Mirror Image Example	414	Thread Milling Tool	435
MIRROR IMAGE ON CNC LATHES	414	Premachining Requirements	435
42 - COORDINATE ROTATION	415	Clearance Radius	435
ROTATION COMMANDS	415	Productivity of Thread Milling	435
Center of Rotation	415	THE HELIX	435
Radius of Rotation	415	THREAD MILLING EXAMPLE	437
Coordinate Rotation Cancel	417	Straight Thread	437
Common Applications	417	Initial Calculations	437
PRACTICAL APPLICATION	417	Starting Position	438
43 - SCALING FUNCTION	421	Motion Rotation and Direction	438
DESCRIPTION	421	Lead-In Motions	440
Scaling Function Usage	421	Thread Rise Calculation	441
PROGRAMMING FORMAT	421	Milling the Thread	441
Scaling Center	422	Lead-Out Motions	442
Scaling Factor	422	Complete Program	442
Rounding Errors in Scaling	423	External Thread Milling	442
PROGRAM EXAMPLES	423	Tapered Thread Milling	442
44 - CNC LATHE ACCESSORIES	425	Further Considerations	443
CHUCK CONTROL	425	THREAD MILLING SIMULATION METHOD	443
Chuck Functions	425	HELICAL RAMPING	444
Chucking Pressure	426	46 - HORIZONTAL MACHINING	445
Chuck Jaws	426	INDEXING AND ROTARY AXES	445
TAILSTOCK AND QUILL	426	INDEXING TABLE (B-AXIS)	445
Tailstock Body	426	Units of Increment	445
Quill	427	Direction of Indexing	446
Center	427	Table Clamp and Unclamp Functions	446
Quill Functions	427	Indexing in Absolute and Incremental Mode	446
Programmable Tailstock	427	B-AXIS AND OFFSETS	447
Safety Concerns	427	Work Offset and B-axis	447
BI-DIRECTIONAL TURRET INDEXING	428	Tool Length Offset and B-axis	448
Programming Example	428	RETURN TO MACHINE ZERO	450
BARFEEDER ATTACHMENT	429	INDEXING AND A SUBPROGRAM	450
Bar Stopper	429	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	50 - PROGRAM VERIFICATION	479
47 - LIVE TOOLING ON LATHES	457	DETECTION OF ERRORS	479
TURN-MILL OR MILL-TURN	457	Preventive Measures	479
Programming Issues	457	Corrective Measures	479
General Terms	457	GRAPHIC VERIFICATION	480
MACHINE DESIGN	458	AVOIDING ERRORS	480
Features	458	Syntax Errors	481
Benefits	458	Logical Errors	481
C-AXIS PROGRAMMING	458	COMMON PROGRAMMING ERRORS	482
XZ + C Axes	458	Program Input Errors	482
FIXED CYCLES	459	Calculation Errors	482
M-functions and C-axis	460	Hardware Errors	482
Drilling on Face	460	Miscellaneous Errors	482
Drilling on Diameter	461	51 - CNC MACHINING	483
General Considerations	461	MACHINING A NEW PART	483
Y-AXIS PROGRAMMING	462	Setup Integrity	484
Plane Selection	463	Program Integrity	484
Additional Axes	463	RUNNING THE FIRST PART	485
POLAR COORDINATE INTERPOLATION	463	PROGRAM CHANGES	486
Polar Coordinate Mode	463	Program Upgrading	486
Approximation Method	464	Program Updating	487
CYLINDRICAL INTERPOLATION	465	Documentation Change	487
Z + C-axis	465	ALTERNATE MACHINE SELECTION	487
Practical Example	465	MACHINE WARM UP PROGRAM	488
48 - WRITING A CNC PROGRAM	467	CNC MACHINING AND SAFETY	488
PROGRAM WRITING	467	SHUTTING DOWN A CNC MACHINE	489
Legibility of Handwriting	468	Emergency Stop Switch	489
Programming Forms	468	Parking Machine Slides	490
Confusing Characters	468	Setting the Control System	490
PROGRAM OUTPUT FORMATTING	469	Turning the Power Off	490
LONG PROGRAMS	471	EQUIPMENT MAINTENANCE	490
Program Length Reduction	471	52 - INTERFACING TO DEVICES	491
Memory Mode and Tape Mode	472	RS-232C INTERFACE	491
49 - PROGRAM DOCUMENTS	473	PUNCHED TAPE	492
DATA FILES	473	Tape Reader and Puncher	492
PROGRAM DOCUMENTATION	474	Leader and Trailer	494
External Documentation	474	Tape Identification	494
Internal Documentation	474	Non-printable Characters	494
Program Description	475	Storage and Handling	494
SETUP AND TOOLING SHEETS	475	DISTRIBUTED NUMERICAL CONTROL	494
Setup Sheet	476	TERMINOLOGY OF COMMUNICATIONS	495
Tooling Sheet	476	Baud Rate	495
Coordinate Sheet	477	Parity	495
DOCUMENTATION FILE FOLDER	477	Data Bits	495
Identification Methods	477	Start and Stop Bits	495
Operator's Suggestions	478	DATA SETTING	495
Filing and Storage	478	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**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**