

TABLE OF CONTENTS

1 - PART PROGRAM DEVELOPMENT	1
Program Development Drawing	1
Drawing Evaluation	2
Material and Stock	3
Part Setup	3
Part Reference Point	3
Part Orientation	3
Selecting Part Zero	4
Tooling Selection	4
Identifying Machining Operations	4
Face Milling	5
Contour Milling	6
Circular Pocket Milling	6
Slot Milling	7
Spot Drilling	8
Drilling	9
Tapping	9
Summary of Tools Used	9
Machining Data	10
Spindle Speed	10
Cutting Feedrate	11
Tooling Data	11
Details of Operations	11
Tool 1 - Face Milling	12
Tool 2 - Outside Contour	13
Tool 2 - Circular Pocket	15
Tool 3 - Slot Milling	16
Tool 4 - Spot Drilling	17
Tool 5 - Drilling	18
Tool 6 - Tapping	19
Complete Program	20
2 - CALCULATING CONTOUR POINTS	23
Tools and Knowledge	23
Mathematical Knowledge	23
Organized Approach	25
Process of Calculating XY Coordinates	25
Step 1 - Establish the Main Contour Points	26
Step 2 - Fill-in the Coordinate Sheet	26
Step 3 - Identify Calculation Zones	27
Step 4 - Helpful Ideas for Calculations	27
Step 5 - Calculations for Zone 1	29
Step 6 - Calculations for Zone 2	31
Updating Coordinate Sheet	32
Writing the CNC Program	32

3 - FORMULAS FOR CONTOURING	33
Contour Point Between Two Lines (Lathe)	33
Contour Point Between Line and Arc	34
Intersecting Contour Point	34
Tangent Contour Point	35
Calculating the Sharp Point	39
Contour Point Between Two Arcs	40
Intersecting Arcs	40
Tangent Arcs	41
4 - USING CUTTER RADIUS OFFSET	43
General Concepts	43
Benefits Of Cutter Radius Offset	44
Controlling Cutter Radius	44
Radius Offset Commands	45
Commands G40-G41-G42	45
Using the D-offset Number	45
Basic Programming Techniques	46
Cutter Radius Activation	46
Cutter Radius Application	47
Cutter Radius Cancellation	47
D-offset Stored Amount	47
Equidistant Centerline - G40 Mode	48
Equidistant Centerline - G41/G42 Mode	48
Drawing Dimensions - G40 Mode	48
Drawing Dimensions - G41/G42 Mode	48
Radius vs Diameter	48
Contour Lead-In and Lead-Out	49
Methods For Lead-In - Linear Motion	49
Methods For Lead-In - Arc Motion	52
Methods For Lead-Out - Linear Motion	53
Methods For Lead-Out - Arc Motion	53
Program Example	53
Internal Contours	54
Linear Slot Machining	54
Circular Slot Machining	55
Finishing Internal Contour	56
Maintaining Dimensional Sizes	58
Basic Rule	58
Handling Dimensional Tolerances	58
Handling Cutter Radius Offset Errors	60
Common Errors	60
Offset Programmed Too Late or Too Early	61
Offset Start or End on an Arc	62
Tool Nose Radius Offset	62
Command Point and Radius Center	62
Tool Tip Orientation	63
Common Tool Nose Radius Errors	64

5 - PART REVERSAL IN MILLING	67
Project Description	67
Material and Setup Conditions	67
Cutting Tools	68
Material Removal	68
Machining Process	69
Clamping 1	69
Clamping 2	70
Program Zero Selection	70
First Clamping	70
Second Clamping	71
Programming Methods	72
Tool Length Settings	72
First Clamping	73
Second Clamping	74
Using the WORK OFFSET Method - G54-G55	74
Common Toolpath	76
Program Listing - WORK OFFSETS G54-G55	77
Program Listing - WORK OFFSETS G54-G55 - with Subprograms	79
Using the LOCAL COORDINATES Method - G52	81
Program Listing - LOCAL COORDINATE SYSTEM G52	83
Using the DATUM SHIFT Method - G10	85
Program Listing - DATUM SHIFT G10	86
Summary	88
6 - USING TAPERED END MILLS	89
Types of Tapered End Mills	89
Tool Material	90
Range of Taper Angles	90
Flat Tip Tapered End Mills	90
Ball Tip Tapered End Mills	91
Effective Diameter Calculation	91
Flat Tip	91
Stock Removal	93
Ball Tip with Specified Radius	94
Flat Tip with Added Blend Radius	94
Tapered Holes	96
7 - SPECIAL PURPOSE G-CODES	97
Single Direction Positioning - G60	97
Special Cutting Modes	98
Exact Stop Check G09 - G61	99
Automatic Corner Override - G62	100
Tapping Mode - G63	100
Normal Cutting Mode - G64	101
Stored Stroke Limits Definitions - G22 - G23	101
Spindle Fluctuation G25 - G26	103

Machine Zero Commands - G27- G28 - G29 - G30	104
Primary Machine Zero Return - G28	104
Return from Machine Zero - G29	106
Machine Zero Return Position Check - G27	106
Secondary Machine Zero Return - G30	108
Position Register - G92/G50	108
G92 Position Register for Milling	109
G50 Position Register for Turning	111
Tool Change Position	113
Conversion of G50 to Geometry Offset	117
Skip Command - G31	118
Other Seldom Used G-codes	119
Tool Length Offset Negative - G44	119
Tool Length Offset Cancel - G49	119
Conclusion	122
8 - TOOL LENGTH OFFSET CHANGE	123
Tool Length Offset	123
Offset Adjustment	124
Practical Application	124
Programming Method 1 - No Offset Adjustment	125
Programming Method 2 - With Offset Adjustment	125
Programming Method 3 - Advanced Macro Method	126
Offset Adjustment - Setup for Two Parts	128
Method 1 - One Work Offset + One Length Offset	128
Method 2 - Two Work Offsets + One Length Offset	129
Method 3 - Two Work Offsets + Two Length Offsets	130
9 - BLOCK SKIP APPLICATIONS	131
General Applications	131
Similar Parts Applications	132
Programming a Trial Cut	134
Trial Cut for Milling	134
Trial Cut for Turning	136
Irregular Stock Removal	137
Variable Stock in Milling	137
Variable Stock in Turning	139
Summary of Rules	139
Block Skip Within A Block	140
Conflicting Words	140
One Program - Two Materials	140
Numbered Block Skip Functions	142
10 - STANDARD AND RIGID TAPPING	143
Standard Tapping Method	143
Basic Principles	143
Why Underfeed?	144
Feed-In Slower - Feed-Out Faster	144

Rigid Tapping Method	146
Basic Principles	146
Benefits	146
Setup	147
Possible Problems	147
Programming Approach	147
11 - POLAR COORDINATES	149
Definition and G-codes	149
Polar Coordinates and Planes	150
G15 - G16 Polar Coordinates	151
Programming Format	151
Toolpath Direction	153
Applications in Planes	154
12 - SUBPROGRAM DEVELOPMENT	155
Definition and Usage	155
Drawing Evaluation	156
Subprogram Planning	156
Depth Control	157
Width of Cut Control	157
Cutting Tool Selection	157
Developing the Subprogram	158
Method 1 - Full Width and Full Depth	158
Method 2 - Full Width and Divided Depth	159
Method 3 - Smaller Width and Full Depth	160
Round Pocket Subprogram	162
Single Depth Pocket with Stepovers	162
Multidepth Pocket with Stepovers	164
Rough and Finish Cuts with a Subprogram	165
One Toolpath for Two Cuts	165
Lead-In and Lead-Out	167
Common Contour Toolpath	167
Main Program	168
13 - TURNING AND BORING IN DEPTH	169
Program Zero Selection	169
Corner Radius and Back Angle Selection	170
Cutter Radius Offset	171
Imaginary Tool Point.	172
Stock Allowance	172
Contour Shape	173
Cutting Tool Used.	173
Stock Allowance in X and Z axes.	173
Grinding Allowance	174

Tool Approach Techniques - Lead-In	176
Approaching the Front Face	176
Approaching a Diameter	176
Approaching a Chamfer	177
Approaching a Radius	177
Approaches to Avoid	179
Tool Retract Techniques - Lead-Out	179
Retract from a Face	179
Retract from a Diameter	180
Retract from a Chamfer	180
Retract from a Radius	181
Retracts to Avoid	181
One Job - Two Operations	182
About Jaws	182
Single Setup - Two Chuckings	183
Two Setups - Two Operations	184
Multi Cut Facing	184
Width of Cut Distribution	184
Breaking Corners	185
Direction Specification	186
Using Tailstock	186
Types of Tailstock	187
Programming a Tailstock with a Bar Stopper	187
Using 45-degree Tool	189
Machining Thin Stock	192
Adjusting Chuck Pressure	192
Using an Inner Plug / Outer Ring	192
Using Special Split Jaws	192
G70/G71/G72 Cycle Methods	193
Programming Formats - G71	193
Programming Formats - G72	194
G70 - Finishing Cycle	196
G71 and G72 Compared	196
Programming Undercuts	198
Hard Turning	198
14 - PROGRAMMING TAPERS	199
What is a Taper?	199
Taper Definitions	199
Taper per Foot	200
Taper Ratios	203
Taper Defined as Percentage	205
Taper Angle Defined in D-M-S	206
Taper Length and Angle	206
Chamfers	206
45° Chamfer	207
Start Chamfer with a Clearance	209
End Chamfer with a Clearance	209
Other Chamfers	210

Tapers with Leads	211
Taper with a Lead Chamfer	212
Taper with a Lead Fillet	213
15 - TECHNIQUES FOR GROOVING	215
Tooling for Grooves	215
Cutting Width	215
Cutting Depth	215
Groove Location	216
Setting the Command Point	216
Plunge and Retract Method	217
G75 Cycle	217
Grooving for Precision	218
Machining Procedure	219
Programming Procedure	220
Deep Grooving	221
Grooves with Tapers - O-Ring Grooves	222
Grooves with Tapers - V-Pulley Grooves	225
Insert Selection	225
Depth Calculation	226
Tool Setup and Program	226
16 - TECHNIQUES FOR THREADING	227
Types of Thread Forms	227
UN - Unified National and Metric	227
Other Thread Forms	228
Thread Depth Calculation	229
Infeed Methods	230
Tool Motions	231
Cutting Conditions	231
Acceleration and Deceleration	231
Cutting Depth	232
Hand of Thread	232
External Threading - Right Hand Thread	232
External Threading - Left Hand Thread	233
Internal Threading - Right Hand Thread	233
Internal Threading - Left Hand Thread	233
Sample Thread Evaluation	234
Initial Data	234
Cutting Conditions	234
Number of Threading Passes	235
Distribution of Depth Cuts	236
G32 Threading Method	236
Radial Infeed Example	237
Flank Infeed Example	237
Tapping with G32	238
G92 Threading Method	240
G76 Threading Method	241
Programming Format	241

17 - RESTRICTIONS IN THREADING	243
Thread Programming Basics	243
Threading Feedrate	244
Standard Example	244
Final Threading Program - Single Start	245
Special Example	245
Final Threading Program - Multi Start	246
Slow Spindle Speed	247
Metric Applications	247
Long Thread Programming	248
Defining a Long Thread	248
Lead Error	249
Number of Decimal Places	250
18 - PRACTICAL THREAD MILLING	251
Thread Milling - General	251
Helical Interpolation	251
Is Helical Interpolation Available ?	252
Benefits of Thread Milling	252
Selecting Tools	253
Initial Factors	253
Types of Thread Milling Cutters	254
Accuracy Issues	254
Thread Mill Data	255
Cutting Direction	255
External and Internal Thread Milling	255
Climb Milling and Conventional Thread Milling	255
Right Hand and Left Hand Thread Milling	255
External Thread Milling Illustrated	256
Internal Thread Milling Illustrated	257
Helix - Helical Curve	258
Programming External Threads	260
Tooling Selection	260
Cutting Conditions	261
Lead-In and Lead-Out	261
Cutter Radius Offset	262
Program Development - External Thread	262
Programming Internal Threads	266
Tooling Selection	266
Cutting Conditions	266
Lead-In and Lead-Out	267
Cutter Radius Offset	267
Program Development - Internal Thread	268
Pipe Thread Milling	271
Thread Milling Software	272

19 - KNURLING ON CNC LATHES	273
Knurling Operations	273
Tooling Selection	273
Knurling Pitch	274
Programming and Machining Techniques	275
Tool Motions	275
Depth and Feedrate	275
Troubleshooting	276
20 - FOUR-AXIS LATHES	277
General Setup	277
Tool Tip Numbers	278
Programming Method	278
Spindle Speed and Feedrate	278
M-Functions	279
Synchronization Functions	279
Program Structure	279
21 - PALLET CHANGERS	283
Types of Automatic Pallets	283
Rotary Pallets	283
Shuttle Pallets	283
Setup and Work Areas	284
Programming Methods	285
M60 Function	285
General Format	285
Programming Example	286
Initial Conditions	286
Part Program	286
22 - WORKING WITH PLANES	289
Mathematical Planes	289
Machine Planes	290
G-codes for Plane Selection	290
Effect of Planes in Programming	291
Planes and Circular Motion	291
Planes and Cutter Radius Offset	293
Working With Planes In Detail	295
G17 with G41 and G02	295
G17 with G42 and G03	296
G18 with G42 and G03	297
G18 with G41 and G02	298
G19 with G41 and G02	299
G19 with G42 and G03	300
Using Right-Angle Attachment	301
Basic Concepts	301
Side Face Drilling	302
Side Face Milling	304

23 - PROGRAMMING CAMS	307
Overview of Cams	307
Cam Drawing Example	308
Cam RISE and FALL - Sections Evaluation	309
The RISE Section	310
Calculating Radius Length	311
Calculating XY Coordinates	311
The FALL Section	312
Calculating Radius Length	313
Calculating XY Coordinates	314
Summary	314
Writing the Program	315
24 - INTRODUCTION TO MACROS	317
Special Introduction	317
Skills Required	317
Macro is an Option	318
Common Features and Applications	318
Macro Structure	319
Macro Definition and Call	320
Variable Declarations and Expressions	322
Macro Functions	323
Branching and Looping	326
Macro Development - Bolt Circle	327
Evaluation of Drawings	328
Bolt Hole Macro Features	329
Assignment of Variables	330
Internal Calculations	330
Other Calculations	330
Final Considerations	331
Macro Call	332
25 - DID YOU KNOW THAT ... ?	333
26 - REFERENCES AND RESOURCES	335
Index	339