

MITSUBISHI CNC

MELDAS 64(D/M)

PROGRAMMING MANUAL

2736F

CONTENT

Page

PROGRAMMING MANUAL

1.	CONTROL AXES	1
1-1	Coordinate Word and Control Axis	1
1-2	Coordinate Systems and Coordinate Zero Point Symbols	2
2.	INPUT COMMAND UNITS	3
2-1	Input Command Units	3
2-2	Input Setting Units	3
3.	DATA FORMATS	4
3-1	Tape Codes	4
3-2	Program Formats	7
3-3	Tape Memory Format	9
3-4	Optional Block Skip ("/")	9
3-4-1	Optional block skip	9
3-4-2	Precautions for using optional block skip	9
3-5	Program/Sequence/Block Numbers (O, N)	10
3-6	Parity H/V	11
3-7	G Code System	12
4.	BUFFER REGISTER	16
4-1	Input Buffer	16
4-2	Pre-Read Buffers	17
5.	POSITION COMMANDS	18
5-1	Position Command Methods; G90, G91	18
5-2	Inch/Metric Conversion (G20, G21)	20
5-3	Decimal Point Input	21
6.	INTERPOLATION FUNCTIONS	26
6-1	Positioning (Rapid Traverse: G00)	26
6-2	Linear Interpolation (G01)	30
6-3	Plane Selection G17, G18, G19	31
6-3-1	Parameter entry	31
6-3-2	Plane selection System	32
6-4	Circular Interpolation (G02, G03)	33
6-5	R-Specified Circular Interpolation (G02, G03)	36
6-6	Helical Interpolation; G17 to G19, G02, G03	38
6-7	Thread Cutting	41
6-7-1	Constant lead thread cutting (G33)	41
6-7-2	Inch thread cutting (G33)	43
7.	FEED FUNCTIONS	45
7-1	Rapid Traverse Rate	45
7-2	Cutting Feed Rate	45
7-3	Synchronous Feed (G94, G95)	46
7-4	Feed Rate Designation and Effects on Control Axes	48

7-5	Exact Stop Check; G09	52
7-6	Exact Stop Check Mode; G61	54
7-7	Automatic Corner Override; G62	54
7-7-1	Description of function	55
7-7-2	Application examples	57
7-7-3	Relations with other functions	58
7-7-4	Precautions	59
7-8	Tapping Mode; G63	60
7-9	Cutting Mode; G64	60
8.	DWELL	61
8-1	Per-second Dwell; G04	61
9.	MISCELLANEOUS FUNCTIONS	63
9-1	Miscellaneous Functions (M8-digits BCD)	63
9-2	Secondary Miscellaneous Functions (A8/C8-digits)	65
10.	SPINDLE FUNCTIONS	66
10-1	Spindle Functions (S2-digits BCD)	66
10-2	Spindle Functions (S6-digits Analog)	66
10-3	Spindle Functions (S8-digits)	67
10-4	Constant Peripheral Speed Control (G96, G97)	67
10-4-1	Constant peripheral speed control	67
10-4-2	Spindle clamp speed setting (G92)	68
11.	TOOL FUNCTIONS	69
11-1	Tool Functions (T8-digits BCD)	69
12.	TOOL OFFSET FUNCTIONS	70
12-1	Tool Offset	70
12-1-1	Outline	70
12-1-2	Tool offset amount designation method	71
12-1-3	Tool offset memory	72
12-1-4	Tool offset No. (H/D)	74
12-2	Tool Length Offset/Cancel; G43, G44/G49	75
12-3	Tool Diameter compensation	78
12-3-1	Outline	78
12-3-2	Tool diameter compensation operation	79
12-3-3	Other operations during tool diameter compensation	89
12-3-4	G41/G42 commands and I, J, K designation	97
12-3-5	Interrupts during tool diameter compensation	104
12-3-6	General checkpoints for tool diameter compensation	107
12-3-7	Changing of offset No. during compensation mode	108
12-3-8	Start of tool diameter compensation and Z axis cut in operation	110
12-3-9	Interference check	111

12-4	Programmed Tool Offset Input (G10)	118
12-4-1	Function and objective	118
13.	PROGRAM SUPPORT FUNCTIONS	123
13-1	Canned Cycles	123
13-1-1	Standard canned cycles ; G80~G89, G73, G74, G76	123
13-1-2	Initial point and R point level return: G98, G99	139
13-1-3	Setting of workpiece coordinates in canned cycle mode	141
13-2	Special canned cycle; G34, G35, G36, G37.1	142
13-3	Subprogram Control (M98, M99)	146
13-4	Variable Commands	151
13-5	User Macro Specifications	154
13-5-1	User macro commands	154
13-5-2	Macro call instruction	155
13-5-3	Variables	164
13-5-4	Types of variables	166
13-5-5	Arithmetic commands	187
13-5-6	Control commands	193
13-5-7	External output commands	197
13-5-8	Checkpoints	199
13-5-9	Actual examples of using user macros	202
13-6	G Command Mirror Image; G50.1, G51.1	206
13-7	Corner Chamfering, Corner Rounding	209
13-7-1	Corner chamfering (,C_)	209
13-7-2	Corner rounding (,R_)	210
13-8	Linear Angle Command	212
13-9	Circle Cutting; G12, GF13	213
13-10	Program Parameter Input (G10, G11)	215
13-11	Macro Interrupt	216
13-11-1	Outline	216
13-11-2	Operation	216
13-12	Tool Position Return	226
13-12-1	General	226
13-12-2	Command format	226
13-12-3	Operation	227
14.	COORDINATES SYSTEM SETTING FUNCTIONS	229
14-1	Coordinate Words and Control Axes	229
14-2	Basic Machine, Work and Local Coordinate Systems	230
14-3	Machine Zero Point and 2nd, 3rd, 4th Reference Points (Zero Point)	231
14-4	Basic Machine Coordinate System	232

14-5	Coordinate System Setting (G92)	233
14-6	Automatic Coordinate System Setting	234
14-7	Reference (Zero) Point Return (G28, G29)	235
14-8	2nd, 3rd, and 4th Reference (Zero) Point Return (G30)	239
14-9	Reference Point Check (G27)	241
14-10	Work Coordinate System Setting and Offset	242
14-11	Local Coordinate System Setting (G52)	248
15.	MEASUREMENT SUPPORT FUNCTIONS	252
15-1	Automatic Tool Length Measurement (G37)	252
15-2	Skip Function (G31)	256
15-2-1	Function	256
15-2-2	Readout of skip coordinates	258
15-2-3	G31 coasting	258
15-2-4	Skip coordinate readout error (mm)	260
15-3	Multi-stage Skip Function; G31.n, G04	262
15-3-1	Function	262
15-3-2	Operation	264
15-4	Multi-skip Function 2	265
15-4-1	Function	265
15-4-2	Operation	266
APPENDIX 1.	PROGRAM PARAMETER INPUT N NO. CORRESPONDENCE TABLE	269
APPENDIX 2.	PROGRAM ERROR	274
APPENDIX 3.	ORDER OF G FUNCTION COMMAND PRIORITY	288