

MITSUBISHI CNC MELDAS 520AM

(MACHINING CENTER,
MILLING,
OTHERS)

PROGRAMMING MANUAL

CONTENTS

1. OUTLINE OF NC PROGRAM	1
1.1 PROGRAM FORMAT	1
1.2 WORD AND ADDRESS	1
1.3 BLOCK	1
1.4 END OF BLOCK	3
1.5 MODAL AND UNMODAL COMMAND	3
2. BASIC CONCEPT FOR PROGRAMMING	4
2.1 CONTROL AXES AND AXIS NAME	4
2.2 MINIMUM PROGRAM UNIT AND MINIMUM MOVEMENT AMOUNT	4
2.3 INCREMENT VALUE COMMAND AND ABSOLUTE VALUE COMMAND	7
2.4 REFERENCE POINT (MACHINE ORIGINAL POINT POSITION) AND ORIGINAL POINT RECOVERY	7
3. OUTLINE OF PROGRAMMING	8
3.1 PROGRAM TO SPECIFY THE MOVEMENT OF TOOLS ALONG THE PROCESS WORD SHAPE	8
3.2 PROGRAM TO COMMAND THE TRAVERSE OF A TOOL AT THE SPECIFIED RATE TO CUT A WORK	9
3.3 PROGRAM TO COMMAND MACHINE OPERATION SUCH AS SPINDLE START AND STOP AND COOLANT ON/OFF	10
3.4 PROGRAM TO COMMAND THE NUMBER OF SPINDLE ROTATIONS	10
3.5 PROGRAM TO SELECT AN APPROPRIATE TOOL FOR WORK MACHINING	10
3.6 CONFIGURATION OF PROGRAM	11
3.7 OTHER PROGRAMMING NOTES	12
3.8 COORDINATE SYSTEM AND COORDINATE ZERO POINT SYMBOLS	12
3.9 OPTIONAL BLOCK SKIP (BLOCK DELETE)	13
4. POSITION COMMANDS	14
4.1 POSITION COMMAND SYSTEMS (G90, G91)	14
4.2 INCH/METRIC CONVERSION (G20, G21)	15
4.3 DECIMAL POINT INPUT	16
5. LIST OF PREPERATORY FUNCTION	19
5.1 LIST OF G COMMAND	19
6. INTERPOLATION FUNCTIONS	21
6.1 POSITIONING (RAPID TRAVERSE: G00)	21
6.2 UNI-DIRECTIONAL POSITIONING (G60)	24
6.3 LINEAR INTERPOLATION (G01)	25

CONTENTS

6.4	PLANE SELECTION (G17, G18, G19)	26
6.4.1	Parameter entry	26
6.4.2	Plane selection system	27
6.5	CIRCULAR INTERPOLATION (G02, G03)	28
6.6	R-SPECIFIED CIRCULAR INTERPOLATION (G02, G03)	31
6.7	HELICAL INTERPOLATION (G17 - G19, G02, G03)	33
6.8	SPIRAL INTERPOLATION (G2.1, G3.1)	36
6.9	IMAGINARY AXIS INTERPOLATION (G07)	40
6.10	THREAD CUTTING	41
6.10.1	Constant lead thread cutting (G33)	41
6.10.2	Continuous thred cutting	43
6.11	INCH THREAD CUTTING (G33)	43
7.	FEED FUNCTIONS	44
7.1	RAPID TRAVERSE RATE	44
7.2	CUTTING FEED RATE	44
7.3	F 1-DIGIT FEED	45
7.4	SYNCHRONOUS FEED (G94, G95)	47
7.5	FEED RATE DESIGNATION AND EFFECTS ON CONTROL AXES	49
7.6	EXACT STOP CHECK (G09)	52
7.7	EXACT STOP CHECK MODE (G61)	53
7.8	AUTOMATIC CORNER OVERRIDE (G62)	54
7.8.1	Description of function	54
7.8.2	Application examples	56
7.8.3	Relations with other functions	57
7.8.4	Checkpoints	58
7.9	TAPPING MODE (G63)	59
7.10	CUTTING MODE (G64)	59
7.11	HIGH-PRECISION CONTROL MODE (G61.1)	60
8.	DWELL	62
8.1	PER-SECOND DWELL (G94/G04)	62
8.2	PER-REVOLUTION DWELL (G95/G04)	63
9.	MISCELLANEOUS FUNCTIONS	64
9.1	MISCELLANEOUS FUNCTIONS (M2-DIGITS BCD)	64
9.2	MISCELLANEOUS FUNCTIONS (M4-DIGITS BCD)	66
9.3	MISCELLANEOUS FUNCTIONS (M8-DIGITS)	66
9.4	SECOND MISCELLANEOUS FUNCTIONS (B/A/C8-DIGITS)	66

10. SPINDLE FUNCTIONS	67
10.1 SPINDLE FUNCTIONS (S2-DIGITS BCD)	67
10.2 SPINDLE FUNCTIONS (S5-DIGITS ANALOG)	67
10.3 SPINDLE FUNCTIONS (S8-DIGITS)	67
10.4 CONSTANT PERIPHERAL SPEED CONTROL (G96, G97)	68
10.4.1 Constant peripheral speed control	68
10.4.2 Spindle clamp speed setting (G92)	68
11. TOOL FUNCTIONS	69
11.1 TOOL FUNCTIONS (T2-DIGITS BCD)	69
11.2 TOOL FUNCTIONS (T8-DIGITS)	69
12. TOOL OFFSET FUNCTIONS	70
12.1 TOOL OFFSET	70
12.1.1 Outline	70
12.1.2 Tool offset amount assignment	71
12.1.3 Tool offset memory	72
12.1.4 Tool offset numbers (H/D)	74
12.1.5 Number of tool offset sets	74
12.2 TOOL LENGTH OFFSET/CANCEL (G43, G44/G49)	75
12.3 TOOL POSITION OFFSET (G45-G48)	78
12.4 CUTTER COMPENSATION (G38-G42)	84
12.4.1 Outline	84
12.4.2 Cutter compensation operations	85
12.4.3 Other commands and operations	93
12.4.4 G41/G42 commands and I,J,K assignment	102
12.4.5 Interrupt during cutter compensation	109
12.4.6 General checkpoints for cutter compensation	107
12.4.7 Changing the offset numbers	110
12.4.8 Cutter compensation start and z-axis cutting	112
12.4.9 Interference check	114
12.5 3-DIMENTIONAL CUTTER COMPENSATION (G40-G42)	120
12.5.1 Function	120
12.5.2 Command method	121
12.5.3 Relationship with other functions	125
12.5.4 Other items relating to 3-dimensional cutter compensation	126
12.6 PROGRAMMED CUTTER COMPENSATION INPUT (G10)	127
12.6.1 Function and objective	127

CONTENTS

13.	PROGRAM SUPPORT FUNCTIONS	132
13.1	CANNED CYCLES	132
13.1.1	Standard canned cycles (G80-G89, G73, G74, G76)	132
13.1.2	Initial point and R point level return (G98, G99)	147
13.1.3	Work coordinate setting in canned cycle mode	148
13.1.4	User's canned cycles (G70-G72, G75, G77-G79)	149
13.2	SPECIAL CANNED CYCLES (G34, G35, G36, G37.1)	150
13.3	SUBPROGRAM CONTROL (M98, M99)	154
13.4	VARIABLE COMMANDS (#n)	160
13.5	PATTERN ROTATION (M98)	162
13.6	COORDINATE ROTATION	167
13.6.1	Programmed coordinate rotation (G68, G69)	169
13.6.2	Parameter coordinate rotation	171
13.7	USER MACROS	172
13.7.1	User macros	172
13.7.2	Macro call instructions (G65, G66, G66.1, G67 etc)	173
13.7.3	Variables	183
13.7.4	Types of variables	185
13.7.5	Arithmetic commands	202
13.7.6	Control commands	207
13.7.7	External output commands	210
13.7.8	Checkpoints	212
13.7.9	Examples using user macros	214
13.8	SCALING (G50,G51)	218
13.8.1	Command format	218
13.8.2	Detailed description	218
13.8.3	Programming examples	220
13.9	G COMMAND MIRROR IMAGE (G50.1, G51.1)	231
13.10	CORNER CHAMFERING, CORNER RI	234
13.10.1	Corner chamfering (, C_)	234
13.10.2	Corner rounding (, R_)	235
13.11	LINEAR ANGLE COMMAND	236
13.12	GEOMETRIC COMMAND	237
13.12.1	Geometric command I	237
13.13	G COMMAND COORDINATE ROTATION INPUT (G10)	239
13.14	CIRCLE CUTTING (G12, G13)	246

14.	COORDINATE SYSTEM SETTING FUNCTIONS	248
14.1	COORDINATE WORDS AND CONTROL AXES	248
14.2	BASIC MACHINE/WORKPIECE/LOCAL COORDINATE SYSTEMS	249
14.3	MACHINE ZERO POINT AND 2ND/3RD/4TH REFERENCE (ZERO) POINTS	250
14.4	BASIC MACHINE COORDINATE SYSTEM	251
14.5	COORDINATE SYSTEM SETTING (G92)	252
14.6	AUTOMATIC COORDINATE SYSTEM SETTING	253
14.7	REFERENCE POINT (ZERO POINT) RETURN (G28, G29)	254
14.8	2ND/3RD/4TH REFERENCE (ZERO) POINT RETURN (G30)	257
14.9	REFERENCE POINT CHECK (G27)	259
14.10	WORK COORDINATE SYSTEM SETTING AND OFFSET (G54 - G59)	260
14.11	LOCAL COORDINATE SYSTEM SETTING (G52)	269
15.	PROTECTION FUNCTIONS	273
15.1	STROKE CHECK BEFORE MOVEMENT (G22, G23)	273
16.	MACHINING MODES	274
16.1	HIGH-SPEED MACHINING MODE (G05)	274
17.	MEASUREMENT SUPPORT FUNCTIONS	279
17.1	AUTOMATIC TOOL LENGTH MEASUREMENT (G37)	279
17.2	SKIP FUNCTION (G31)	282
17.2.1	Function	282
17.2.2	Skip coordinate readout	283
17.2.3	G31 coasting	284
17.2.4	Skip coordinate readout error	285
17.3	MUSTI-STAGE SKIP FUNCTION (G31.n, G04)	287
17.3.1	Function	287
17.3.2	Operation	288

CONTENTS

APPENDIX 1.	LIST OF INPUT KEY CODE FUNCTION	289
APPENDIX 2.	LIST OF COMMAND VALUE RANGES	290
APPENDIX 3.	LIST OF DATA RETENTION STATUSES AND OUTPUT SIGNALS WITH NC SHUTDOWN	291
APPENDIX 4.	SEQUENCE OF PRIORITY AMONG G FUNCTION COMMANDS	295
APPENDIX 5.	ARC CUTTING RADIUS ERRORS	305
APPENDIX 6.	CANNED CYCLE SUBPROGRAMS	306
APPENDIX 7.	DETAILS OF USER MACRO INTERRUPT FUNCTION	314
APPENDIX 8.	LIST OF ALARMS	321
APPENDIX 9.	PAPER TAPE DATA FORMAT	334
APPENDIX 10.	ASCII CODE TABLE	339
APPENDIX 11.	LIST OF VARIABLE NUMBERS	340
APPENDIX 12.	HOW TO USE THE PROCESS PROGRAM DEVELOPED BY 10 μ s COMMAND UNIT	342