

# MITSUBISHI CNC

## MELDAS 300 Series (320L, 330HL)

### PROGRAMMING MANUAL

---

<b>CONTENTS</b>	
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 UNIT .....	3
2.1 INPUT COMMAND UNITS .....	3
2.2 INPUT SETTING UNITS .....	3
2.3 COMMAND UNIT X10 .....	4
3. DATA FORMATS .....	5
3.1 TAPE CODES .....	5
3.2 PROGRAM FORMATS .....	8
3.3 TAPE MEMORY FORMAT .....	10
3.4 OPTIONAL BLOCK SKIP ("") .....	10
3.5 PROGRAM/SEQUENCE/BLOCK NUMBERS (O,N) .....	11
3.6 PARITY H/V .....	12
3.7 G CODE SYSTEM .....	14
4. BUFFER REGISTER .....	16
4.1 INPUT BUFFER .....	16
4.2 PRE-READ BUFFERS .....	16
5. POSITION COMMANDS .....	17
5.1 INCREMENTAL/ABSOLUTE VALUE COMMANDS .....	17
5.2 RADIAL/DIAMETRICAL COMMANDS .....	18
5.3 INCH/METRIC CONVERSION (G20, G21) .....	19
5.4 DECIMAL POINT INPUT .....	19
6. INTERPOLATION FUNCTIONS .....	23
6.1 POSITIONING (RAPID TRAVERSE: G00) .....	23
6.2 LINEAR INTERPOLATION (G01) .....	25
6.3 CIRCULAR INTERPOLATION (G02, G03) .....	27
6.4 R-SPECIFIED CIRCULAR INTERPOLATION (G02, G03) .....	31
6.5 PLANE SELECTION .....	32
6.6 THREAD CUTTING .....	33
7. FEED FUNCTIONS .....	38
7.1 RAPID TRAVERSE RATE .....	38
7.2 CUTTING FEED RATE .....	38
7.3 SYNCHRONOUS/ASYNCHRONOUS FEED (G94, G95) .....	39
7.4 FEED RATE DESIGNATION AND EFFECTS ON CONTROL AXES .....	40

---

7.5	THREAD CUTTING LEADS .....	44
7.6	AUTOMATIC ACCELERATION/DECELERATION .....	44
7.7	SPEED CLAMP .....	45
7.8	EXACT STOP CHECK (G09) .....	45
8.	DWELL .....	48
8.1	PER-SECOND DWELL (G94/G04) .....	48
8.2	PER-REVOLUTION DWELL (G95/G04) .....	49
9.	MISCELLANEOUS FUNCTIONS .....	50
9.1	MISCELLANEOUS FUNCTIONS (M2-DIGITS BCD) .....	50
9.2	MISCELLANEOUS FUNCTIONS (M8-DIGITS) .....	51
9.3	SECONDARY MISCELLANEOUS FUNCTIONS (A8/B8/C8-DIGITS) .....	52
10.	SPINDLE FUNCTIONS .....	53
10.1	SPINDLE FUNCTIONS (\$2-DIGITS BCD) .....	53
10.2	SPINDLE FUNCTIONS (\$5-DIGITS ANALOG) .....	53
10.3	SPINDLE FUNCTIONS (\$8-DIGITS) .....	53
10.4	CONSTANT PERIPHERAL SPEED CONTROL (G96, G97) .....	54
11.	TOOL FUNCTIONS .....	55
11.1	TOOL FUNCTIONS (T4-DIGITS) .....	55
11.2	TOOL FUNCTIONS (T8-DIGITS) .....	55
12.	TOOL OFFSET FUNCTIONS .....	56
12.1	TOOL OFFSET .....	56
12.2	TOOL LENGTH OFFSET .....	58
12.3	TOOL NOSE WEAR OFFSET .....	60
12.4	TOOL NOSE RADIUS COMPENSATION .....	61
12.4.1	OUTLINE .....	61
12.4.2	TOOL NOSE POINT AND COMPENSATION DIRECTIONS .....	63
12.4.3	TOOL NOSE R COMPENSATION OPERATIONS .....	65
12.4.4	OTHER OPERATIONS DURING NODE R COMPENSATION .....	73
12.4.5	G41/G42 COMMANDS AND I, J, K DESIGNATION .....	81
12.4.6	INTERRUPTS DURING TOOL NOSE RADIUS COMPENSATION .....	86
12.4.7	GENERAL CHECKPOINTS FOR TOOL NOSE RADIUS COMPENSATION .....	88
12.4.8	INTERFERENCE CHECK .....	89
12.5	PROGRAMMED TOOL OFFSET INPUT (G10) .....	93

---

13. PROGRAM SUPPORT FUNCTIONS .....	96
13.1 CANNED CYCLES FOR TURNING .....	96
13.1.1 LOGITUDINAL CUTTING CYCLE (G77) .....	97
13.1.2 THREAD CUTTING CYCLE (G78) .....	99
13.1.3 FACE CUTTING CYCLE (G79) .....	101
13.2 MULTIPLE REPETITIVE CANNED CYCLES .....	103
13.2.1 LONGITUDINAL ROUGH TURNING CYCLE I (G71) .....	104
13.2.2 FACE ROUGH TURNING CYCLE I (G72) .....	107
13.2.3 STOCK REMOVAL IN ROUGH MACHINING CYCLE .....	110
13.2.4 FINISHING CYCLE (G70) .....	113
13.2.5 FACE CUT-OFF CYCLE (G74) .....	114
13.2.6 LONGITUDINAL CUT-OFF CYCLE (G75) .....	115
13.2.7 MULTIPLE REPETITIVE THREAD CUTTING CYCLE (G76) .....	116
13.2.8 CHECKPOINTS FOR MULTIPLE REPETITIVE CANNED CYCLE (G70-G76) .....	119
13.3 HOLE MACHINE CANNED CYCLES (G80-G89) .....	121
13.3.1 G38 FACE DEEP HOLE DRILLING CYCLE 1 .....	125
13.3.2 G84 FACE TAPPING CYCLE .....	126
13.3.3 G84 FACE BORING CYCLE .....	127
13.3.4 G80 HOLE MACHINING CANNED CYCLE CANCEL .....	127
13.3.5 CHECKPOINTS FOR USING HOLE MACHINING CANNED CYCLES .....	127
13.4 DEEP HOLE DRILLING CYCLE 2 (G83.2) .....	129
13.5 SUBPROGRAM CONTROL (M98, M99) .....	132
13.6 VARIABLES COMMANDS .....	137
13.7 USER MACRO SPECIFICATIONS .....	139
13.7.1 USER MACRO COMMANDS (G65, G66, G66.1, G67) .....	139
13.7.2 MACRO CALL INSTRUCTIONS .....	140
13.7.3 VARIABLES .....	147
13.7.4 TYPES OF VARIABLES .....	149
13.7.5 ARITHMETIC COMMANDS .....	163
13.7.6 CONTROL COMMANDS .....	169
13.7.7 EXTERNAL OUTPUT COMMANDS .....	172
13.7.8 CHECKPOINT .....	174
13.8 FACING TOOL REST MIRROR IMAGE (G68, G69) .....	176
13.9 CORNER CHAMFERING, CORNER ROUNDING FUNCTION I .....	180
13.10 CORNER CHAMFERING, ROUNDING FUNCTION II .....	182
13.11 LINEAR ANGLE COMMAND .....	187
13.12 GEOMETRIC COMMAND .....	188
13.13 DATA SETTING (G10, G11) .....	190

---

14. COORDINATE SYSTEM SETTING FUNCTIONS .....	200
14.1 COORDINATE WORDS AND CONTROL AXES .....	200
14.2 BASIC MACHING, WORK AND LOCAL COORDINATE SYSTEMS .....	201
14.3 MACHINE ZERO POINT AND 2ND REFERENCE POINT (ZERO POINT) .....	202
14.4 BASIC MACHINE COORDINATE SYSTEM (G53) .....	203
14.5 AUTOMATIC COORDINATE SYSTEM SETTING (G92) .....	204
14.6 REFERENCE (ZERO) POINT RETURN (G28, G29) .....	205
14.7 2ND REFERENCE (ZERO) POINT RETURN (G30) .....	208
14.8 REFERENCE POINT CHECK (G27) .....	210
14.9 WORK COORDINATE SYSTEM SETTING AND OFFSET (G54—G59) .....	211
14.10 LOCAL COORDINATE SYSTEM SETTING (G52) .....	214
15. PROTECTION FUNCTIONS .....	215
15.1 CHUCK BARRIERS/TAILSTOCK BARRIERS .....	215
16. MEASUREMENT SUPPORT FUNCTIONS .....	218
16.1 AUTOMATIC TOOL LENGTH MEASUREMENT (G37) .....	218
16.2 SKIP FUNCTION (G31) .....	221
APPENDIX 1. ....	226
APPENDIX 2. ....	227
APPENDIX 3. ....	228
APPENDIX 4. ....	229
APPENDIX 5. ....	231
APPENDIX 6. ....	233
APPENDIX 7. ....	261
PAST HISTORY OF CHANGES AND REVISIONS .....	273