

PROGRAMMING MANUAL

for

MAZATROL M PLUS

(EIA/ISO Programming)

Manual No. : H733PB0013E

Before using this machine and equipment, fully understand the contents of this manual to ensure proper operation. Should any questions arise, please ask the nearest Technical/Service Center.

IMPORTANT NOTICE

1. Be sure to observe the safety precautions described in this manual and the contents of the safety plates on the machine and equipment. Failure may cause serious personal injury or material damage. Please replace any missing safety plates as soon as possible.
2. No modifications are to be performed that will affect operation safety. If such modifications are required, please contact the nearest Technical/Service Center.
3. For the purpose of explaining the operation of the machine and equipment, some illustrations may not include safety features such as covers, doors, etc. Before operation, make sure all such items are in place.
4. This manual was considered complete and accurate at the time of publication, however, due to our desire to constantly improve the quality and specification of all our products, it is subject to change or modification. If you have any questions, please contact the nearest Technical/Service Center.
5. Always keep this manual near the machinery for immediate use.
6. If a new manual is required, please order from the nearest Technical/Service Center with the manual No. or the machine name, serial No. and manual name.

Issued by *Manual Publication Section, Yamazaki Mazak Corporation, Japan*

CONTENTS

	Page
1. CONTROL AXES	1-1
1-1 Coordinate Words and Control Axes	1-1
2. UNITS OF PROGRAM DATA INPUT	2-1
2-1 Units of Program Data Input	2-1
2-2 Units of Data Setting	2-1
2-3 Ten-Fold Program Data	2-2
3. DATA FORMATS	3-1
3-1 Tape Codes	3-1
3-2 Program Formats	3-5
3-3 Tape Storage Format	3-7
3-4 Optional Block Skip: /	3-7
3-4-1 Usage of optional block skip	3-7
3-4-2 Operating notes on use of optional block skip	3-7
3-5 Program Number, Sequence Number, and Block Number: O, N	3-8
3-6 Parity-H/V	3-9
3-7 List of G-Codes	3-11
3-8 List of M-Codes	3-12
4. BUFFER REGISTERS	4-1
4-1 Input Buffer	4-1
4-2 Pre-Read Buffer	4-2
5. POSITION PROGRAMMING	5-1
5-1 Position Programming Commands: G90, G91	5-1
5-2 Inch/Metric Selection Commands: G20, G21	5-3
5-3 Decimal Point Input	5-4
6. INTERPOLATION FUNCTIONS	6-1
6-1 Positioning (Rapid Feed) Command: G00	6-1
6-2 One-Way Positioning Command: G60	6-5
6-3 Linear Interpolation Command: G01	6-6
6-4 Plane Selection Commands: G17, G18, G19	6-8
6-4-1 Parameter registration	6-8
6-4-2 Plane selection methods	6-9
6-5 Circular Interpolation Commands: G02, G03	6-10
6-6 Radius-Designated Circular Interpolation Commands: G02, G03	6-13
6-7 Helical Interpolation Commands: G17, G18, G19 and G02, G03	6-14
6-8 Spiral Interpolation Commands: G2.1, G3.1 (Option)	6-18
6-9 Virtual-Axis Interpolation Command: G07	6-22
6-10 Spline Interpolation Command: G06.1 (Option)	6-24

7.	FEED FUNCTIONS	7-1
7-1	Rapid Feed Rates	7-1
7-2	Cutting Feed Rates	7-1
7-3	Synchronous Feed Commands: G94, G95	7-2
7-4	Selecting a Feed Rate, and Effects on Each Control Axis	7-3
7-5	Exact-Stop Check Command: G09	7-7
7-6	Exact-Stop Check Mode Command: G61	7-9
7-7	Automatic Corner Override Command: G62	7-9
7-7-1	Detailed description	7-10
7-7-2	Sample programs	7-11
7-7-3	Correlations to other command functions	7-13
7-7-4	Precautions	7-13
7-8	Tapping Mode Command: G63	7-14
7-9	Cutting Mode Command: G64	7-15
7-10	Inverse Time Feed Command: G93 (Option)	7-15
8.	DWELL FUNCTIONS	8-1
8-1	Dwell Command in Time: (G94) G04	8-1
8-2	Dwell Command in Number of Revolutions: (G95) G04	8-2
9.	MISCELLANEOUS FUNCTIONS	9-1
9-1	Miscellaneous Functions (M3-Digit)	9-1
9-2	No. 2 Miscellaneous Functions (B3-Digit)	9-2
10.	SPINDLE FUNCTION	10-1
11.	TOOL FUNCTIONS	11-1
11-1	Tool Function (T3-Digit)	11-1
11-2	Tool Function (T8-Digit)	11-1
12.	TOOL OFFSET FUNCTIONS	12-1
12-1	Tool Offset	12-1
12-1-1	Overview	12-1
12-1-2	Selecting the amounts of tool offset	12-2
12-1-3	Tool offset memories	12-2
12-1-4	Tool offset numbers (H/D)	12-4
12-1-5	Number of sets of tool offset numbers	12-4
12-2	Tool Length Offset/Cancellation Commands: G43, G44/G49	12-5
12-3	Tool Position Offset Commands: G45 to G48	12-8
12-4	Tool Diameter Offset Function: G40, G41, G42	12-14
12-4-1	Overview	12-14
12-4-2	Tool diameter offsetting	12-14
12-4-3	Tool diameter offsetting operation using other commands	12-23
12-4-4	Corner movement	12-32
12-4-5	Interruptions during tool diameter offsetting	12-32
12-4-6	General precautions on tool diameter offsetting	12-35

12-4-7	Offset number updating during the offset mode	12-36
12-4-8	Excessive cutting due to tool diameter offsetting	12-38
12-4-9	Interference check	12-41
12-5	Three-Dimensional Tool Diameter Offsetting (Option)	12-48
12-5-1	Function description	12-48
12-5-2	Programming methods	12-49
12-5-3	Correlations to other functions	12-54
12-5-4	Miscellaneous notes on three-dimensional tool diameter offsetting	12-54
12-6	Programmed Input of Tool Offset Data: G10	12-55
12-7	Tool Offsetting Based on MAZATROL Tool Data	12-60
12-7-1	Selecting parameters	12-60
12-7-2	Tool length offsetting	12-60
12-7-3	Tool diameter offsetting	12-62
12-7-4	Tool data update (during automatic operation)	12-62
13.	AUXILIARY FUNCTIONS FOR PROGRAMMING	13-1
13-1	Fixed-Cycle Functions	13-1
13-1-1	Function description	13-1
13-1-2	List of fixed-cycle functions	13-1
13-1-3	Fixed-cycle machining data format	13-2
13-1-4	G71.1 (Chamfering cutter CW)	13-5
13-1-5	G72.1 (Chamfering cutter CCW)	13-6
13-1-6	G73 (High-speed deep-hole drilling)	13-7
13-1-7	G74 (Reverse tapping)	13-8
13-1-8	G75 (Boring)	13-9
13-1-9	G76 (Boring)	13-10
13-1-10	G77 (Back spot facing)	13-11
13-1-11	G78 (Boring)	13-12
13-1-12	G79 (Boring)	13-13
13-1-13	G81 (Spot drilling)	13-14
13-1-14	G82 (Drilling)	13-15
13-1-15	G83 (Deep-hole drilling)	13-16
13-1-16	G84 (Tapping)	13-17
13-1-17	G85 (Reaming)	13-18
13-1-18	G86 (Boring)	13-19
13-1-19	G87 (Back boring)	13-20
13-1-20	G88 (Boring)	13-21
13-1-21	G89 (Boring)	13-21
13-1-22	Synchronous tapping (Option)	13-22
13-2	Initial Point and R-Point Level Return Commands: G98, G99	13-26
13-3	Workpiece Coordinate Setting during the Fixed-Cycle Mode	13-27
13-4	Subprogram Control Commands: M98, M99	13-27
13-5	Mutual Subprogram Call between EIA/ISO and MAZATROL (Option)	13-32
13-6	Variables Commands	13-35
13-7	Figure Rotate Command: M98 (Option)	13-37

13-8	Programmed Coordinate Rotation	13-43
13-9	User Macros (Option)	13-45
13-9-1	User macros	13-45
13-9-2	Macro call instructions	13-46
13-9-3	Variables	13-55
13-9-4	Types of variables	13-57
13-9-5	Arithmetic operation commands	13-77
13-9-6	Control commands	13-82
13-9-7	External output commands	13-86
13-9-8	Precautions	13-88
13-9-9	Specific examples of programming using user macros	13-90
13-10	Scaling Commands: G50, G51	13-94
13-10-1	Programming formats	13-94
13-10-2	Detailed description	13-94
13-10-3	Sample programs	13-97
13-11	Mirror Image On/Off G-Code Commands: G50.1, G51.1	13-108
13-12	Linear Angle Commands	13-110
13-13	Geometric Commands	13-111
13-14	Corner Chamfering and Corner Rounding Commands	13-112
13-14-1	Corner chamfering (,C_)	13-112
13-14-2	Corner rounding (,R_)	13-114
14.	COORDINATE SYSTEM SETTING FUNCTIONS	14-1
14-1	Coordinate Words and Control Axes	14-1
14-2	Fundamental Machine Coordinate System, Workpiece Coordinate Systems, and Local Coordinate Systems	14-2
14-3	Machine Zero Point and Second, Third, and Fourth Reference Points	14-3
14-4	Fundamental Machine Coordinate System: G53	14-4
14-5	Coordinate System Setting Command: G92	14-5
14-6	Automatic Coordinate System Setting	14-6
14-7	Reference Point (Zero Point) Return Commands: G28, G29	14-7
14-8	Second-, Third-, or Fourth-Reference Point (Zero Point) Return Command: G30	14-10
14-9	Reference Point Check Command: G27	14-13
14-10	Workpiece Coordinate System Setting/Offsetting Commands: G54 through G59	14-13
14-11	Additional Workpiece Coordinate System Setting/Offsetting Command: G54.1 (Option)	14-20
14-12	Local Coordinate System Setting Command: G52	14-26
14-13	Reading/Writing of MAZATROL Program Basic Coordinates	14-31
14-13-1	Calling a macroprogram (for data writing)	14-31
14-13-2	Data reading	14-32
14-13-3	Rewriting	14-32
14-14	Workpiece Coordinate System Rotation	14-34

15.	PROTECTION FUNCTIONS	15-1
15-1	Before-Movement Stroke Limit Check Commands: G22, G23	15-1
16.	SKIP COMMAND: G31	16-1
16-1	Function Description	16-1
16-2	Skip Coordinate Reading	16-2
16-3	Amount of Coasting	16-3
16-4	Skip Coordinate Reading Error	16-4
16-5	Multi-Step Skip Commands: G31.1, G31.2, G31.3, G04	16-6
16-5-1	Function	16-6
16-5-2	Machine action	16-7
17.	THREADING: G33 (Option)	17-1
17-1	Equal-Lead Threading	17-1
17-2	Continuous Threading	17-4
17-3	Inch Threading	17-4
18.	AUTOMATIC TOOL LENGTH MEASUREMENT: G37 (Option)	18-1
19.	DYNAMIC OFFSETTING: M173, M174 (Option)	19-1
20.	HIGH-SPEED MACHINING MODE FEATURE (OPTION)	20-1
21.	FIVE-SURFACE MACHINING FUNCTION (OPTION) - <i>Project</i> -	21-1
21-1	Coordinate Systems for Five-Surface Machining	21-1
21-2	Surface Selection Command	21-2
21-3	Head Offsetting	21-2
21-3-1	Specification of head offsetting	21-2
21-3-2	Cancellation of head offsetting	21-3
22.	ARBITRARY-SURFACE MACHINING FUNCTION (OPTION) - <i>Project</i> -	22-1
22-1	Three-Dimensional Coordinate Conversion: G68	22-1
22-2	Simultaneous-Operation M-Codes	22-9
22-3	Table Rotational Machining (Option)	22-10
22-4	Workpiece Coordinate Conversion	22-12
22-5	Inclined-Surface Synchronous Tapping	22-14
22-6	Inclined-Surface Boring	22-15
22-7	Sample Program	22-16
23.	PROGRAMS EXAMPLES	23-1
24.	EIA/ISO PROGRAM DISPLAY	24-1
24-1	Procedures for Constructing an EIA/ISO Program	24-1
24-2	Editing Function of EIA/ISO PROGRAM Display	24-2
24-3	Macro-Instruction Input	24-9