

PROGRAMMING MANUAL

for

MAZATROL FUSION 640MT/MT 5X

MAZATROL FUSION 640T/TE

MAZATROL FUSION 640T NEXUS/TN

Programming EIA/ISO

MANUAL No. : H734PB0031E

Serial No. :

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 INTRODUCTION	1-1
2 UNITS OF PROGRAM DATA INPUT	2-1
2-1 Overview	2-1
2-2 Detailed description	2-1
3 DATA FORMATS	3-1
3-1 Tape Codes	3-1
3-2 Program Formats	3-5
3-3 Tape Data Storage Format	3-6
3-4 Optional Block Skip	3-6
3-5 Program Number, Sequence Number and Block Number : O, N	3-7
3-6 Parity-H/V	3-8
3-7 List of G-Codes	3-10
4 BUFFER REGISTERS	4-1
4-1 Input Buffer	4-1
4-2 Preread Buffer	4-2
5 POSITION PROGRAMMING	5-1
5-1 Absolute Data Command and Incremental Data Command: G90, G91	5-1
5-2 Inch/Metric Selection Commands: G20/G21	5-2
5-3 Decimal Point Input	5-3
5-4 Selection/Cancellation of X-axis Radial Command: G122.1/G123.1	5-7

6	INTERPOLATION FUNCTIONS	6-1
6-1	Positioning (Rapid Feed) Command: G00	6-1
6-2	Linear Interpolation Command: G01	6-4
6-3	Circular Interpolation Commands: G02, G03	6-6
6-4	Radius Designated Circular Interpolation Commands: G02, G03	6-9
6-5	Plane Selection Commands: G16, G17, G18, G19	6-11
6-5-1	Outline	6-11
6-5-2	Plane selection methods	6-12
6-5-3	Milling mode ON/OFF: G12.1/G13.1 (T32 compatible mode)	6-13
6-5-4	Polar coordinate command ON/OFF: G122/G123	6-16
6-6	Polar Coordinate Interpolation ON/OFF: G12.1/G13.1 (Standard Mode)	6-17
6-7	Cylindrical Interpolation Command: G07.1 (G107) (Standard Mode)	6-21
6-8	Threading	6-24
6-8-1	Constant lead threading: G32	6-24
6-8-2	Inch threading: G32	6-27
6-8-3	Continuous threading	6-28
6-8-4	Variable lead threading: G34	6-29
6-8-5	Threading with C-axis interpolation: G01.1	6-29
6-8-6	Automatic correction of threading start position (for overriding in a threading cycle)	6-32
6-9	Helical Interpolation: G17, G18, G19 and G02, G03	6-34
7	FEED FUNCTIONS	7-1
7-1	Rapid Feed Rates	7-1
7-2	Cutting Feed Rates	7-1

7-3	Asynchronous/Synchronous Feed Commands: G98/G99	7-1
7-4	Selecting a Feed Rate and Effects on Each Control Axis	7-3
7-5	Threading Leads	7-6
7-6	Automatic Acceleration/Deceleration	7-7
7-7	Speed Clamp	7-7
7-8	Exact-Stop Check Command: G09	7-8
7-9	Exact-Stop Check Mode Command: G61	7-11
7-10	Automatic Corner Override Command: G62	7-11
7-11	Cutting Mode Command: G64	7-16
7-12	Geometry Compensation/Accuracy Coefficient: G61.1/,K	7-16
7-12-1	Geometry compensation function: G61.1	7-16
7-12-2	Accuracy coefficient (,K)	7-17
8	DWELL FUNCTIONS	8-1
8-1	Dwell Command in Time: (G98) G04	8-1
8-2	Dwell Command in Number of Revolutions: (G99) G04	8-2
9	MISCELLANEOUS FUNCTIONS	9-1
9-1	Miscellaneous Functions (M3-Digit)	9-1
9-2	No. 2 Miscellaneous Functions (A8/B8/C8-Digit)	9-2
10	SPINDLE FUNCTIONS	10-1
10-1	Spindle Function (S5-Digit Analog)	10-1
10-2	Constant Peripheral Speed Control ON/OFF: G96/G97	10-1
10-3	Spindle Clamp Speed Setting: G50	10-2

11	TOOL FUNCTIONS.....	11-1
11-1	Tool Function (4-Digit T-Code)	11-1
11-2	Tool Function (6-Digit T-Code)	11-2
11-3	Next Tool Automatic Selection	11-2
11-4	Tool Life Management and Spare Tool Change.....	11-3
11-4-1	Tool life management (T32 compatible mode)	11-3
11-4-2	Tool life management (Standard mode)	11-5
12	TOOL OFFSET FUNCTIONS.....	12-1
12-1	Tool Offset.....	12-1
12-2	Tool Position Offset.....	12-3
12-3	Tool Nose Radius Compensation: G40, G41, G42, G46	12-5
12-3-1	Outline	12-5
12-3-2	Tool nose point and compensation directions.....	12-7
12-3-3	Tool nose radius compensation operations	12-10
12-3-4	Other operations during tool nose radius compensation.....	12-17
12-3-5	Commands G41/G42 and I, J, K designation	12-24
12-3-6	Interruptions during tool nose radius compensation	12-29
12-3-7	General precautions on tool nose radius compensation	12-31
12-3-8	Interference check.....	12-32
12-4	Programmed Tool Offset Input: G10 L10.....	12-37
12-5	Programmed Parameter Input: G10, G11.....	12-38
13	PROGRAM SUPPORT FUNCTIONS.....	13-1
13-1	Fixed Cycles for Turning.....	13-1

13-1-1	Longitudinal turning cycle: G90	13-2
13-1-2	Threading cycle: G92	13-4
13-1-3	Edge turning cycle: G94	13-6
13-2	Multiple Repetitive Fixed Cycles.....	13-8
13-2-1	Longitudinal roughing cycle : G71	13-9
13-2-2	Edge roughing cycle: G72.....	13-15
13-2-3	Cast or forged workpiece roughing cycle: G73	13-17
13-2-4	Finishing cycle: G70.....	13-21
13-2-5	Edge cut-off cycle: G74.....	13-22
13-2-6	Longitudinal cut-off cycle : G75	13-26
13-2-7	Multiple repetitive threading cycle: G76.....	13-29
13-2-8	Checkpoints for multiple repetitive fixed cycles: G70 to G76	13-35
13-3	Hole Machining Fixed Cycles: G80 to G89.....	13-38
13-3-1	Outline	13-38
13-3-2	Face/Longitudinal deep hole drilling cycle: G83/G87	13-41
13-3-3	Face/Longitudinal tapping cycle: G84/G88.....	13-42
13-3-4	Face/Longitudinal boring cycle: G85/G89.....	13-43
13-3-5	Face/Longitudinal synchronous tapping cycle: G84.2/G88.2	13-43
13-3-6	Hole machining fixed cycle cancel: G80	13-45
13-3-7	Checkpoints for using hole machining fixed cycles.....	13-45
13-3-8	Sample programs with fixed cycles for hole machining.....	13-47
13-4	Subprogram Control: M98, M99	13-48
13-5	End Processing: M02, M30, M198, M199.....	13-55
13-6	Opposite Turret Mirror Image: G68, G69.....	13-56

13-7	Chamfering and Corner Rounding at Right Angle Corner	13-59
13-8	Chamfering and Corner Rounding at Arbitrary Angle Corner Function	13-62
13-8-1	Chamfering at arbitrary angle corner: , C_.....	13-62
13-8-2	Rounding at arbitrary angle corner: , R_.....	13-63
13-9	Linear Angle Command.....	13-64
14	MACRO CALL FUNCTION.....	14-1
14-1	User Macroprogram.....	14-1
14-2	Macro Call Instructions: G65, G66 (Cancellation: G67).....	14-2
14-3	Variable	14-9
14-4	Types of Variables.....	14-11
14-5	Arithmetic Operation Commands	14-24
14-6	Control Commands.....	14-29
14-7	External Output Commands	14-32
14-8	Precautions	14-34
14-9	Macro Interruption	14-36
14-10	MAZATROL Program Call by Macro Call Command: G65.....	14-41
15	COORDINATE SYSTEM SETTING FUNCTIONS.....	15-1
15-1	Coordinate System Setting Function: G50	15-1
15-2	MAZATROL Coordinate System Cancellation: G52 (or G52.5).....	15-5
15-3	Selection of MAZATROL Coordinate System: G53 (or G53.5).....	15-7
15-4	Selection of Workpiece Coordinate System: G54 to G59	15-9
15-5	Workpiece Coordinate System Shift.....	15-10

15-6	Change of Workpiece Coordinate System by Program Command.....	15-10
15-7	Selection of Machine Coordinate System: G53 (Standard Mode)	15-11
15-8	Selection of Local Coordinate System: G52 (Standard Mode)	15-12
15-9	Automatic Return to Reference Point (Zero Point): G28, G29.....	15-13
15-10	Return to Second Reference Point (Zero Point): G30	15-15
15-11	Floating Reference Point Return: G30.1.....	15-17
15-12	Return to Reference Point Check Command: G27	15-18
15-13	Program Coordinate System Rotation ON/OFF : G68.5/G69.5	15-19
16	MEASUREMENT SUPPORT FUNCTIONS.....	16-1
16-1	Skip Function: G31	16-1
16-1-1	Function description	16-1
16-1-2	Amount of coasting	16-3
16-1-3	Skip coordinate reading error	16-4
16-2	Preparation for Measurement: G36	16-5
16-3	Measurement Computing : G37	16-13
16-4	Measurement Program Example.....	16-15
16-5	Automatic Tool Offset: G36/G37 (Standard Mode).....	16-17
17	PROTECTIVE FUNCTION	17-1
17-1	Stored Stroke Limit ON/OFF: G22/G23	17-1
18	POLYGONAL MACHINING AND HOBGING	18-1
18-1	Polygonal Machining ON/OFF: G51.2/G50.2	18-1
18-2	Selection/Cancellation of Hob Milling Mode: G114.3/G113	18-3

19	TORNADO TAPPING (G130).....	19-1
20	TWO-LINE CONTROL FUNCTION	20-1
20-1	Two Process One Program: G109	20-1
20-2	Specifying/Cancelling Cross Machining Control Axis: G110/G111	20-2
20-3	M, S, T, B Output Function to Counterpart: G112	20-7
20-4	Waiting for Turrets	20-8
20-5	Common Macro Variable between Turrets	20-8
21	FUNCTIONS PROPER TO SQR SERIES	21-1
21-1	Programming for the SQR Series Machine	21-1
21-2	Waiting Command: M950 to M997, P1 to P99999999	21-2
21-3	Selection of Turret for Spindle Speed Function: M560/M561	21-4
21-4	Balanced Cutting	21-5
22	DIFFERENCES OF PROGRAMMING FORMAT BETWEEN T32 COMPATIBLE MODE AND STANDARD MODE	22-1
23	EIA/ISO PROGRAM DISPLAY	23-1
23-1	Procedures for Constructing an EIA/ISO Program	23-1
23-2	Editing Function of EIA/ISO PROGRAM Display	23-2
23-2-1	General	23-2
23-2-2	Operation procedure	23-2
23-3	Macro-Instruction Input	23-8
23-4	Division of Display (Split Screen)	23-9
24	ADVANCED INPUT SENSITIVITY (FOR 2-AXIS MODELS ONLY)...	24-1

APPENDIX	1
1. List of Function Codes.....	1
2. List of Command Values and Setting Ranges	2
3. Arc Cutting Radius Error.....	3
4. Additional Explanation About Incomplete Thread Portion in Threading.....	4
5. Additional Explanation About Y-axis.....	6