PROGRAMMING MANUAL

for

MAZATROL FUSION 640

Programming EIA/ISO

MANUAL No. : H734PB0011E

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

02.1999

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

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 Plane selection methods 6-12

6-5-2 Milling mode ON/OFF: G12.1/G13.1 (T32 compatible mode) 6-13

6-5-3 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

6-9 Helical Interpolation: G17, G18, G19 and G02, G03 6-32

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
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 11-2
11-4-1 Tool life management (T32 compatible mode) 11-2

11-4-2 Tool life management (Standard mode) 11-4

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 Face/Longitudinal deep hole drilling cycle: G83/G87 13-41
13-3-2 Face/Longitudinal tapping cycle: G84/G88 13-42
13-3-3 Face/Longitudinal boring cycle: G85/G89 13-43
13-3-4 Face/Longitudinal synchronous tapping cycle: G84.2/G88.2 13-43
13-3-5 Hole machining fixed cycle cancel: G80 13-45
13-3-6 Checkpoints for using hole machining fixed cycles 13-45
13-3-7 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 18-1

18-1 Polygonal Machining ON/OFF: G51.2/G50.2 18-1

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 20-4

20-4 Waiting for Turrets 20-5

20-5 Common Macro Variable between Turrets 20-5

20-6 Phase Matching between Both Spindles 20-5

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 Constant Peripheral Speed Control: 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 FIVE DECIMAL DIGITS FOR INCH SYSTEM (QT SERIES) 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