

Mazak

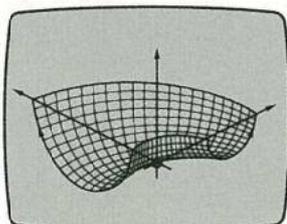
PROGRAMMING MANUAL

for

MAZATROL MATRIX NEXUS
(for Turning Machines)
EIA/ISO Program

MANUAL No. : H740PB0073E

**PROGRAMMING
MANUAL**



CONTENTS

	Page
1 UNITS OF PROGRAM DATA INPUT	1-1
1-1 Units of Program Data Input.....	1-1
1-2 Units of Data Setting.....	1-1
1-3 Ten-Fold Program Data.....	1-1
2 DATA FORMATS.....	2-1
2-1 Tape Codes.....	2-1
2-2 Program Formats	2-5
2-3 Tape Data Storage Format.....	2-6
2-4 Optional Block Skip	2-6
2-5 Program Number, Sequence Number and Block Number: O, N	2-7
2-6 Parity-H/V	2-8
2-7 List of G-Codes	2-10
3 BUFFER REGISTERS.....	3-1
3-1 Input Buffer.....	3-1
3-2 Preread Buffer	3-2
4 POSITION PROGRAMMING.....	4-1
4-1 Dimensional Data Input Method	4-1
4-1-1 Absolute/Incremental data input	4-1
4-2 Inch/Metric Selection: G20/G21.....	4-2
4-3 Decimal Point Input	4-3

4-4	Polar Coordinate Input ON/OFF: G122/G123.....	4-6
4-5	X-axis Radial Command ON/OFF: G122.1/G123.1	4-7
5	INTERPOLATION FUNCTIONS.....	5-1
5-1	Positioning (Rapid Feed) Command: G00.....	5-1
5-2	One-Way Positioning: G60	5-4
5-3	Linear Interpolation Command: G01.....	5-5
5-4	Circular Interpolation Commands: G02, G03.....	5-7
5-5	Radius Designated Circular Interpolation Commands: G02, G03	5-10
5-6	Plane Selection Commands: G17, G18, G19.....	5-12
5-6-1	Outline	5-12
5-6-2	Plane selection methods.....	5-12
5-7	Polar Coordinate Interpolation ON/OFF: G12.1/G13.1.....	5-13
5-8	Virtual-Axis Interpolation: G07	5-17
5-9	Cylindrical Interpolation Command: G07.1	5-18
5-10	Threading	5-21
5-10-1	Constant lead threading: G32.....	5-21
5-10-2	Inch threading: G32	5-24
5-10-3	Continuous threading.....	5-25
5-10-4	Variable lead threading: G34	5-26
5-10-5	Threading with C-axis interpolation: G01.1.....	5-27
5-10-6	Automatic correction of threading start position (for overriding in a threading cycle)	5-29
5-11	Helical Interpolation: G17, G18, G19 and G02, G03	5-31

6	FEED FUNCTIONS	6-1
6-1	Rapid Traverse Rates.....	6-1
6-2	Cutting Feed Rates.....	6-1
6-3	Synchronous/Asynchronous Feed: G99/G98	6-1
6-4	Selecting a Feed Rate and Effects on Each Control Axis.....	6-3
6-5	Threading Leads.....	6-6
6-6	Automatic Acceleration/Deceleration.....	6-7
6-7	Speed Clamp.....	6-7
6-8	Exact-Stop Check Command: G09.....	6-8
6-9	Exact-Stop Check Mode Command: G61.....	6-11
6-10	Automatic Corner Override Command: G62.....	6-11
6-11	Tapping Mode Command: G63	6-16
6-12	Cutting Mode Command: G64.....	6-16
7	DWELL FUNCTIONS	7-1
7-1	Dwell Command in Time: (G98) G04.....	7-1
7-2	Dwell Command in Number of Revolutions: (G99) G04	7-2
8	MISCELLANEOUS FUNCTIONS	8-1
8-1	Miscellaneous Functions (M3-Digit).....	8-1
8-2	No. 2 Miscellaneous Functions (A8/B8/C8-Digit).....	8-2
9	SPINDLE FUNCTIONS	9-1
9-1	Spindle Function (S5-Digit Analog).....	9-1
9-2	Constant Surface Speed Control ON/OFF: G96/G97	9-1

9-3	Spindle Clamp Speed Setting: G50	9-3
10	TOOL FUNCTIONS	10-1
10-1	Tool Function (4-Digit T-Code)	10-1
10-2	Tool Function (6-Digit T-Code)	10-2
10-3	Tool Function (8-Digit T-Code)	10-2
11	TOOL OFFSET FUNCTIONS	11-1
11-1	Tool Offset	11-1
11-2	Tool Position Offset	11-3
11-3	Nose R/Tool Radius Compensation: G40, G41, G42	11-5
11-3-1	Outline	11-5
11-3-2	Tool nose point and compensation directions	11-7
11-3-3	Operations of nose R/tool radius compensation	11-8
11-3-4	Other operations during nose R/tool radius compensation	11-15
11-3-5	Commands G41/G42 and I, J, K designation	11-22
11-3-6	Interruptions during nose R/tool radius compensation	11-27
11-3-7	General precautions on nose R/tool radius compensation	11-29
11-3-8	Interference check	11-30
11-4	Programmed Data Setting: G10	11-35
11-5	Tool Offsetting Based on MAZATROL Tool Data	11-44
11-5-1	Selection parameters	11-44
11-5-2	Tool radius compensation	11-45
11-5-3	Tool data update (during automatic operation)	11-45

12	PROGRAM SUPPORT FUNCTIONS.....	12-1
12-1	Fixed Cycles for Turning.....	12-1
12-1-1	Longitudinal turning cycle: G90	12-2
12-1-2	Threading cycle: G92.....	12-4
12-1-3	Transverse turning cycle: G94.....	12-6
12-2	Compound Fixed Cycles	12-8
12-2-1	Longitudinal roughing cycle: G71	12-9
12-2-2	Transverse roughing cycle: G72.....	12-14
12-2-3	Contour-parallel roughing cycle: G73	12-16
12-2-4	Finishing cycle: G70	12-20
12-2-5	Longitudinal cut-off cycle: G74	12-21
12-2-6	Transverse cut-off cycle: G75.....	12-24
12-2-7	Compound threading cycle: G76	12-27
12-2-8	General notes on the compound fixed cycles G70 to G76	12-34
12-3	Hole Machining Fixed Cycles: G80 to G89.....	12-37
12-3-1	Outline	12-37
12-3-2	Face/Outside deep hole drilling cycle: G83/G87	12-40
12-3-3	Face/Outside tapping cycle: G84/G88.....	12-41
12-3-4	Face/Outside boring cycle: G85/G89.....	12-41
12-3-5	Face/Outside synchronous tapping cycle: G84.2/G88.2	12-42
12-3-6	Hole machining fixed cycle cancel: G80	12-44
12-3-7	Checkpoints for using hole machining fixed cycles	12-44
12-3-8	Sample programs with fixed cycles for hole machining	12-45

12-4	Hole Machining Pattern Cycles: G234.1/G235/G236/G237.1.....	12-46
12-4-1	Overview	12-46
12-4-2	Holes on a circle: G234.1	12-46
12-4-3	Holes on a line: G235	12-48
12-4-4	Holes on an arc: G236	12-50
12-4-5	Holes on a grid: G237.1	12-51
12-5	Subprogram Control: M98, M99	12-53
12-6	End Processing: M02, M30, M998, M999.....	12-60
12-7	Chamfering and Corner Rounding at Right Angle Corner	12-61
12-8	Chamfering and Corner Rounding at Arbitrary Angle Corner	12-64
12-8-1	Chamfering at arbitrary angle corner: , C_	12-64
12-8-2	Rounding at arbitrary angle corner: , R_.....	12-65
12-9	Linear Angle Commands	12-66
12-10	Macro Call Function: G65, G66, G66.1, G67.....	12-67
12-10-1	User macros	12-67
12-10-2	Macro call instructions	12-68
12-10-3	Variables.....	12-77
12-10-4	Types of variables.....	12-79
12-10-5	Arithmetic operation commands	12-104
12-10-6	Control commands.....	12-108
12-10-7	External output commands (Output via RS-232C).....	12-112
12-10-8	External output command (Output onto the hard disk)	12-114
12-10-9	Precautions.....	12-116
12-10-10	Specific examples of programming using user macros	12-118

13	COORDINATE SYSTEM SETTING FUNCTIONS.....	13-1
13-1	Coordinate System Setting Function: G50	13-1
13-2	MAZATROL Coordinate System Cancellation: G52.5	13-5
13-3	Selection of MAZATROL Coordinate System: G53.5	13-7
13-4	Selection of Workpiece Coordinate System: G54 to G59	13-9
13-5	Additional Workpiece Coordinate System Setting and Selection: G54.1 (Option)	13-10
13-6	Workpiece Coordinate System Shift	13-12
13-7	Change of Workpiece Coordinate System by Program Command.....	13-12
13-8	Selection of Machine Coordinate System: G53	13-13
13-9	Selection of Local Coordinate System: G52	13-14
13-10	Automatic Return to Reference Point (Zero Point): G28, G29.....	13-15
13-11	Return to Second Reference Point (Zero Point): G30	13-17
13-12	Return to Reference Point Check Command: G27	13-19
13-13	Programmed Coordinate Conversion ON/OFF: G68.5/G69.5	13-20
14	MEASUREMENT SUPPORT FUNCTIONS.....	14-1
14-1	Skip Function: G31	14-1
14-1-1	Function description.....	14-1
14-1-2	Amount of coasting in the execution of a G31 block.....	14-3
14-1-3	Skip coordinate reading error	14-4
15	PROTECTIVE FUNCTIONS	15-1
15-1	Pre-move Stroke Check ON/OFF: G22/G23	15-1

16	TWO-SYSTEM CONTROL FUNCTION	16-1
16-1	Specifying/Cancelling Cross Machining Control Axis: G110/G111	16-1
16-2	M, S, T, B Output Function to Counterpart: G112	16-5
17	POLYGONAL MACHINING	17-1
17-1	Polygonal Machining ON/OFF: G51.2/G50.2	17-1
18	MEASUREMENT MACROS	18-1
19	TORNADO TAPPING (G130).....	19-1
20	EIA/ISO PROGRAM DISPLAY	20-1
20-1	Procedures for Constructing an EIA/ISO Program	20-1
20-2	Editing Function of EIA/ISO PROGRAM Display.....	20-2
20-2-1	General	20-2
20-2-2	Operation procedure.....	20-2
20-3	Macro-Instruction Input.....	20-8
20-4	Division of Display (Split Screen).....	20-9
20-5	Editing Programs Stored in External Memory Areas	20-12