



MACHINING CENTERS

**STANDARD PROGRAMMING MANUAL
FOR
MONARCH VMC'S
STYLE A and STYLE B
with GE/Fanuc Controls**

Proprietary information of the Monarch Machine Tool Company furnished for customer use only. No other uses are authorized without the written permission of the Monarch Machine Tool Company.

The Monarch Machine Tool Company
641 NYS RT 13 South
Cortland, New York 13045-0749
(607) 753-6001 Fax: (607) 753-7127
Service Fax: (607) 753-6344

TABLE OF CONTENTS

CHAPTER		PAGE
	INTRODUCTION	
1	HOW TO PLAN A PROGRAM	
1.1	Program Planning Sheet	1-2
1.2	Tool and Fixture Considerations	1-2
1.3	Sequence of Operations	1-2
1.4	Establishing the X and Y Program Zero - Section A	1-4
	Establishing the C and Y Program Zero - Section B (VMC-RT and VMC-RTB Machines only)	1-7
1.5	Establishing the Z Program Zero	1-10
1.5.1	Machines with Manual Tool Setting (Style A Machines without TLI)	1-10
1.5.2	Machines with Automatic Tool Setting (All Style B's and Style A's with TLI option)	1-12
1.6	Selection of Speeds and Feeds	1-14
1.7	Operator's Instructions	1-18
1.8	Establishing the W Program Zero (Style A Machines Only)	1-19
1.9	Tooling Diagrams	
1.9.1	Standard Tool Changer	1-24
	Maximum Tool Size, 40 or 60 Position Automatic Tool Changer (Figure 1-10) Style A Machines Only	1-25
	Tool Swing Clearance (Figure 1-11) Style A Machines Only	1-26
	Dimensional Data for Pinned Adapters (Figure 1-12) Style A Machines Only	1-27
	Adapter to Spindle Relationship (Figure 1-13) Style A Machines Only	1-28
1.9.2	ANSI 45V Flange Tool Changer (Style A Machines Only)	1-29
	Maximum Tool Size, 20 or 40 Position ANSI Tool Changer (Figure 1-14) Style A Machines Only	1-30
	Tool Swing Clearance, ANSI (Figure 1-15) Style A Machines Only	1-31
	Boring Bar Orientation, ANSI (Figure 1-16) Style A Machines Only	1-32
	Retention Stud (Figure 1-17)	1-33
1.9.3	ANSI Tool Changer (Style B Machines Only)	1-34
	Maximum Tool Size, V-Flange #40, #45, #50 Taper (Figure 1-18) Style B Machines Only	1-35
	Tool Change Clearance (Figure 1-19) Style B Machines Only	1-36
	Boring Bar V-Flange #45 Taper (Figure 1-20) Style B Machines Only	1-37
	Boring Bar Orientation, ANSI Tool Changer (Figure 1-21) Style B Machines Only	1-38
2	GENERAL INPUT	
2.1	Format Detail	2-1
2.1.1	Decimal Point Programming	2-1
2.1.2	Inch to Metric/Metric to Inch Conversion	2-2
2.2	Basic Program Codes	2-3
2.2.1	Sequence Number (N5)	2-3
2.2.2	Preparatory Commands (G3)	2-3
2.2.3	Axis Commands	2-4
2.2.4	Axis Feedrate Command	2-7
2.2.5	Tool Commands (T2)	2-7

	GENERAL INPUT (continued)	
2.2.6	Tool Radius Compensation (D2)	2-8
2.2.7	Tool Length Compensation (H2)	2-8
2.2.8	Tool Compensation Memory	2-8
2.2.9	Miscellaneous Functions (M3)	2-8
2.2.10	Spindle Speeds (S4)	2-9
2.2.11	Program Number (O4)	2-9
2.2.12	Variables (#3)	2-9
3	WRITING THE PROGRAM	
3.1	Safe Start Modules	3-1
3.1.1	Style A Machines Only	3-1
3.1.2	Style B Machines Only	3-4
3.1.3	Style A and Style B Machines	3-6
3.2	Operation Module	3-8
3.3	Park Module	3-9
3.4	Operator Messages	3-11
4	PREPARATORY FUNCTION WORDS	
	Preparatory Function Words (list)	4-1
4.1	G90 Absolute and G91 Incremental Programming - Section A	4-3
	G90 Absolute and G91 Incremental Programming for VMC-RT and VMC-RTB Machines - Section B	4-5
4.2	Positioning Cycles	4-7
4.2.1	G0 Non Linear Interpolation Positioning Cycle	4-7
4.2.2	G60 Single Direction Positioning	4-8
4.3	G80 Cancel Canned Cycles	4-8
4.4	General Information	4-8
4.4.1	G28 Automatic Reference Point Return	4-8
4.4.2	G43 Tool Length Offset Add	4-8
	G44 Tool Length Offset Subtract	4-8
	G49 Tool Length Offset Cancel	4-8
4.4.3	G54 through G59	4-9
4.4.4	Diagram Symbolology	4-9
4.5	Canned Cycles (G81 thru G89, G73, G74 and G76)	4-10
4.5.1	G81 Drilling Cycle	4-13
4.5.2	G82 Counter Bore Cycle	4-18
4.5.3	G83 Peck Drilling Cycle	4-20
4.5.4	G73 High Speed Peck Drilling Cycle	4-22
4.5.5	Tapping Cycle	4-24
	G184 Air Tapping (Style A Machines Only)	4-24
	G185 Fast Air Tapping (Style A Machines Only)	4-27
	G84 Servo Tapping Cycle	4-29
	G74 Left Hand Servo Tapping	4-30
4.5.6	G85 Boring (Feed In/Out)	4-31
4.5.7	G86 Boring Cycle with Spindle Stop Rapid Retract	4-33
4.5.8	G76 Fine Boring Cycle with Spindle Orient	4-35
4.5.9	G87 Back Boring Cycle	4-37
4.5.10	G89 Boring Cycle with Dwell	4-39

5	MISCELLANEOUS FUNCTION CODES	
	Miscellaneous Function Codes (list)	5-1
	General Rules Applied to Miscellaneous Function Codes	5-2
5.1	M0 Program Stop	5-3
5.2	M1 Optional Stop	5-3
5.3	M2 End of Program	5-3
5.4	M3 Start Spindle Clockwise	5-3
5.5	M4 Start Spindle Counterclockwise	5-3
5.6	M5 Spindle Stop	5-3
5.7	M6 Tool Changer	5-4
5.8	M7 Mist Coolant On (optional)	5-4
5.9	M8 Flood Coolant On	5-4
5.10	M9 Coolant Off	5-4
5.11	M10 High Pressure Tapping (Style A Machines Only)	5-4
5.12	M12 Tool Pocket Reset	5-5
5.13	M13 SEt Tool in Spindle = T Code Command (Style B Machines Only)	5-5
5.14	M15 Programming Erickson Model 600 Indexer #1	5-5
5.15	M18 Coolant Through Spindle (optional)	5-5
5.16	M19 Spindle Orientation	5-5
5.17	M20 Tool Setup (applicable only for machines with Non-ANSI tool changers) Style A Machines Only	5-6
5.18	M24/M25 4th Axis Clamps Off/On (optional)	5-6
5.19	M26/M27 5th Axis Clamps Off/On (optional)	5-6
5.20	M28 Absolute Tool Number Request	5-7
5.21	M29 Machine Lubrication Cycle	5-7
5.22	M30 End of Program with Return to Program Start	5-7
5.23	M31 Increment Parts Count	5-7
5.24	M32 Reset Cycle Time	5-7
5.25	M34 Reset Parts Count, Lot Parts Count	5-7
5.26	M37 Set Lot Parts Count	5-7
5.27	M39 180 Spindle Orient (only for machines with optional spindle probe)	5-7
5.28	M48/M49 Speed and Feedrate Override	5-8
5.29	M50 through M54 (optional)	5-8
5.30	M60 through M64 (optional)	5-8
5.31	M66 Master Tool Set	5-8
5.32	M68 Activate Spindle Probe (optional)	5-8
5.33	M69 Activate TLI (optional)	5-8
5.34	M71/M72 Clamps Off/On (Style A Machines Only)	5-9
5.35	M80 Non Buffering M Code	5-9
5.36	M148/M149 Spindle Air Blast On/Off (optional)	5-9
5.37	M98 Call Subprogram	5-10
5.38	M99 End of Subprogram	5-10
5.39	M395 Zero Set	5-10
5.40	M396/M397 (for Hiperdex Indexer only)	5-10
5.41	M911 - M919, M921 - M929 Multiple Block Delete	5-11
5.42	M950 Vary Tool Offsets (for machines with probe option)	5-11
5.43	Multiple M Codes (optional on GE Fanuc 15 only)	5-12

CHAPTER		PAGE
6	CONTOUR MILLING	
	Contour Milling Codes (list)	6-1
6.1	Cutting Mode G64 and Exact Stop Modes G61/G09	6-2
6.1.1	G64 Cutting Mode	6-2
6.1.2	G61 Exact Stop Mode	6-2
6.1.3	G09 Exact Stop One-Shot	6-3
6.1.4	Deceleration	6-3
6.2	G1 Linear Interpolation	6-4
6.3	G2 and G3 Circular Interpolation	6-9
6.3.1	Arc Center Offsets (I,J,K)	6-9
6.3.2	Arc Radius Programming (R)	6-9
6.3.3	X,Y,Z Departures	6-9
6.4	Examples of Circular Interpolation	6-10
6.5	Circular Contouring with the Third Axis Linear	6-19
6.6	Switchable Plane Circular Contouring	6-22
6.7	G4 Programmed Dwell	6-24
6.8	Considerations Specific to VMC-RT and VMC-RTB	6-25
7	CUTTER RADIUS COMPENSATION	
	Cutter Radius Compensation Codes (list)	7-1
7.1	Cutter Compensation	7-2
7.2	Programming Codes used with Cutter Compensation	7-3
7.2.1	G41/G42	7-3
7.2.2	D(1-99)	7-4
7.2.3	G1,G2,G3	7-4
7.3	Activation of Cutter Compensation	7-5
7.4	Cancellation of Cutter Compensation	7-6
7.5	Examples	7-9
7.6	G39 Corner Arc Command (used only with G41/G42)	7-21
7.7	Work Surface Programming	7-26
7.8	Further Considerations	7-28
7.8.1	Plane Changing with Cutter Compensation	7-28
7.8.2	M0/M1 (Program Stop/Optional Stop) with Cutter Compensation	7-28
7.9	Look-Ahead Capability	7-29
7.10	Summary	7-32
8	SUBPROGRAMS	
9	VARIABLES, MATH CAPABILITY, PROGRAM FLOW WORD COMMANDS AND REPORT GENERATION	
9.1	Variables	9-2
9.1.1	Local Variables	9-3
9.1.2	Common Variables	9-4
9.1.3	System Variables	9-5
	Chart 9-1: Miscellaneous System Variables	9-7
	Chart 9-2: System Variables for Work Offsets	9-8
	Chart 9-3: System Variables for Additional Work Coordinate System	9-9
	Chart 9-4: Tool Offsets	9-10
9.2	Mathematical Capability	9-11
9.2.1	Mathematical Operations (+ - * /)	9-11
9.2.2	Mathematical Functions	9-12

CHAPTER		PAGE
	VARIABLES, MATH CAPABILITY, PROGRAM FLOW ... (continued)	
9.2.3	Algebraic Expressions	9-13
9.3	Program Flow Control Commands	9-16
9.3.1	Unconditional Branch	9-16
9.3.2	Conditional Branch	9-16
9.3.3	Iteration (repetition) using IF and GOTO Statements	9-17
9.3.4	Iteration (repetition) using WHILE DO Statements	9-19
9.4	External Data Output commands	9-25
9.4.1	Command Words	9-25
9.4.2	Parameter Settings	9-27
10	CUSTOM MACRO	
10.1	Program Levels	10-2
10.2	Macro/Program Format	10-3
10.3	Macro Call Commands	10-3
10.4	Variable Usage	10-4
10.4.1	Variable Assignment I	10-5
10.4.2	Variable Assignment II	10-5
	Figure 10-4: Variable Assignment using the Entire Alphabet	10-6
	Figure 10-5: Variable Assignment using Alpha Characters with Understood Subscripts	10-7
10.5	Macro Call Commands	10-8
10.5.1	G65 Macro Call Command	10-8
10.5.2	G66 Modal Macro Command Call Motion Block	10-12
10.5.3	G66.1 Modal Macro Command Call Non-Motion Block	10-14
10.6	Supplementary Examples of Custom Macros	10-22
10.6.1	Safe Start Macro	10-22
10.6.2	Bolt Circle Macro	10-27
10.6.3	Small Inside Circle	10-28
10.6.4	Large Inside Circle	10-30
11	COORDINATE SYSTEMS	
11.1	Machine Coordinate System G53/G54	11-4
11.2	Work Coordinate System	11-5
11.2.1	G92 Work Coordinates System Change	11-5
11.2.2	Work Coordinate System Selection	11-6
11.2.3	G52 Local Coordinate System	11-10
12	TAPE/PROGRAM PREPARATION	
12.1	Tape Material	12-1
12.2	Character Coding	12-1
12.3	Reader Tape Capacity	12-1
12.4	Tape/Program Format	12-2
12.4.1	Tape/Program Start	12-2
12.4.2	Leader Section and Label Skip	12-2
12.4.3	Program Start	12-2
12.4.4	Program Number and Name	12-2
12.4.5	Tape/Program Section	12-3
12.4.6	Tape/Program End	12-4
	Beginning and End of Tape (Figure 12-1)	12-5
	Table of Character Sets (Figure 12-2)	12-6

CHAPTER

PAGE

13	INDEXERS AND ROTARY TABLES	
13.1	M15 Programming the Erickson Model 600 Indexer	13-1
13.2	Programming the Hiperdex 360 Indexer	13-4
13.3	Rotary Table Axis	13-9
13.3.1	Format	13-9
13.3.2	Motion Commands	13-10
13.3.3	Rotary Axis Program Examples	13-11
14	OPTIONS	
14.1	Coordinate Rotation	14-1
14.1.1	Activation of Coordinate Rotation	14-2
14.1.2	Cancellation of Coordinate Rotation	14-3
14.1.3	Examples	14-4
14.2	Scaling	14-7
14.2.1	Activation of Scaling	14-7
14.2.2	Cancellation of Scaling	14-8
14.2.3	Examples	14-8
14.2.4	Coordinate Rotation with Scaling	14-10
14.3	Broken Tool Detection	14-11
14.3.1	M81 Broken Tool Detection/Non-Tapping Canned Cycle	14-11
14.3.2	M82 Look for a Broken Tool Now	14-11
14.3.3	M83 Look for a Broken Tool after Delay	14-11
14.3.4	M84 Continuous Broken Tool Detection	14-11
14.3.5	M85 Broken Tool Detection/Tapping Canned Cycle	14-12
14.3.6	General Rules	14-12
14.3.7	Results of Detecting a Broken Tool	14-12
14.4	Tool Life Management	14-13
14.4.1	Tool Life Data Display during Execution of Machining Program	14-16
14.4.2	Preset Tool Life Count	14-16
14.5	Expanded Tool Life Management (available only w/ GE/Fanuc 15M Controls)	14-17
14.5.1	Setting Tool Life Management Data for Tool Group/s	14-17
14.5.2	Specifying Life Count Type per Tool Group	14-18
14.5.3	Tool Selection	14-18
14.5.4	Expanded Tool Life Management Display Menu	14-19
14.5.5	Editing Tool Life Management Data	14-20
14.6	Polar Coordinate Interpolation (Available only on GE/Fanuc 15MA/MF)	14-23
14.7	Polar Coordinate Input (G15, G16)	14-25
14.8	Spindle Load Monitor	14-32
14.8.1	Load Monitor Enable/Disable	14-32
14.8.2	Setting/Clearing Load Monitor Data	14-33
14.9	Chopping	14-36

15 CUSTOMER SPECIAL APPLICATIONS (if required)

APPENDIX

Parameter 1400; Bit 4	1
Parameter 2202; Bit 2	2
Parameter 2400; Bit 0	2
Parameter 2410	2
Parameter 6210	2
Parameter 6211	3
Parameter 6200; Bit 2	3
Parameter 6400; Bit 0	3
Parameter 6410	3
Parameter 6411	4