

PROGRAMMING MANUAL
FOR
VM, VK, HF, HG TYPE
MACHINING CENTER
SEIKI-SEICOS MII

4 - 1989

HITACHI SEIKI

CONTENTS

INTRODUCTION

1-1	Flow Chart for Machining Work by Machining Center	1 - 1
1-2	Programming Type	1 - 2
2.	Programming Language	2 - 1
2-1	Program No.	2 - 1
2-2	Program	2 - 1
2-3	Main Program	2 - 2
2-4	Subprogram	2 - 2
2-5	Composition of Program	2 - 2
2-6	Address	2 - 3
2-7	Data	2 - 3
2-8	Word	2 - 3
2-9	Block	2 - 3
2-10	Sequence No.	2 - 4
2-11	How to Prepare Sequence No.	2 - 4
2-12	Tape Dimension Specification	2 - 5
2-13	Tape Code	2 - 5
2-14	Tape Format	2 - 6
2-15	Address and Meaning	2 - 6
2-16	Program Zero Point and Coordinate System	2 - 8
2-17	Absolute Command (Absolute Coordinate Value)	2 - 9
2-18	Incremental Command (Incremental Value)	2 - 10
2-19	Right Hand Perpendicularly Crossing Coordinate System	2 - 11
2-20	Z-Axis	2 - 11
2-21	X-Axis	2 - 11
2-22	Y-axis	2 - 12
2-23	Z-Axis Zero Point	2 - 12
2-24	X, Y, Z Standard Coordinate and Actual Work	2 - 13
3.	M, S, T, F, B Functions	3 - 1
3-1	Miscellaneous (M-Functions)	3 - 1
3-2	Command Method of Pallet Change (M-Function)	3 - 2

3-3	Command Method of Spindle Speed rpm (S-Function)	3 - 5
3-4	Tool No. Call (T-Function)	3 - 6
3-5	Programming Example of Tool Change (VK,HF)	3 - 8
3-6	Programming Example of Tool Change (VM, VK, HG).....	3 - 9
3-7	Command Method of Feed Speed (F-Function)	3 - 12
3-8	Table-Indexing Command Method (B-Function)	3 - 13

4.	G-Function (Preparatory Function)	4 - 1
4-1	G $\Delta\Delta\Delta$	4 - 1
4-2	G00 (Positioning)	4 - 2
4-3	G01 (Linear Interpolation)	4 - 3
4-4	G02, G03 (Circular Interpolation)	4 - 4
4-5	G02, G03 (Program Example)	4 - 5
4-6	G02, G03 (Program Example)	4 - 6
4-7	G02, G03 (Program Example)	4 - 7
4-8	Summary on G00, G01, G02, G03	4 - 8
4-9	G04 (Dwell)	4 - 9
4-10	G17, G18, G19 (Plane Indication)	4 - 10
4-11	G27 (Zero-Point Return Check)	4 - 11
4-12	G28 (Auto-Zero Return)	4 - 12
4-13	G29 (Auto-Return from Zero Point)	4 - 13
4-14	G30 (2nd, 3rd,4th Reference Point Return)	4 - 14
4-15	G31 (Skip Function)	4 - 15

5.	G-Function (Diameter Compensation, Length Compensation, Position Compensation)	5 - 1
5-1	Tool Diameter Compensation G41, G42, G40	5 - 1
5-2	Tool Diameter Compensation G41, G42, G40	5 - 2
5-3	Summary of Tool-Diameter Compensation	5 - 3
5-4	G41, G42 (Start-up)	5 - 4
5-5	G41, G42 (Offset Mode)	5 - 6
5-6	G40 (Cancel)	5 - 8
5-7	Example of Tool-Diameter Compensation Program	5 - 10
5-8	Example of Tool-Diameter Compensation Program	5 - 11
5-9	Example of Tool-Diameter Compensation Program	5 - 12

5-10	Example of Tool-Diameter Compensation Program	5 - 13
5-11	How to Consider Tool Length Compensation	5 - 14
5-12	Tool-Length Compensation (G43, G44, G49)	5 - 15
5-13	Tool-Position Offset (G45, G46, G47, G48)	5 - 16
5-14	Tool-Position Offset (G45, G46, G47, G48)	5 - 17
5-15	Example Using Positional Offset for Tool-Diameter Radius .	5 - 18
5-16	Example Using the Tool-Position Offset for Milling Machining	5 - 19
6.	G-Function (Coordinate System)	6 - 1
6-1	Setting of Coordinate System (G92)	6 - 1
6-2	Caution for the Use of (G54 ~ G59) and G92 Work-Coordinate	6 - 2
6-3	Work-Coordinate System (G54 ~ G59)	6 - 3
6-4	Work-Coordinate System G54, G55 and Coordinate System Setting G92	6 - 4
6-5	Example Using Work-Coordinate System	6 - 5
6-6	Selection of Machine-Coordinate System (G53)	6 - 6
6-7	Local Coordinate System (G52)	6 - 7
6-8	G52 Program Example	6 - 8
6-9	G10 Programmable Data Input	6 - 9
7.	G-Function (Canned Cycle)	7 - 1
7-1	Canned Cycle (G73 ~ G89)	7 - 1
7-2	Canned Cycle	7 - 2
7-3	Canned Cycle (Data Type, Return Level)	7 - 3
7-4	Canned Cycle (G73, G74)	7 - 4
7-5	Canned Cycle (G76, G81)	7 - 5
7-6	Canned Cycle (G82, G83)	7 - 6
7-7	Canned Cycle (G84, G85, G86)	7 - 7
7-8	Canned Cycle (G87, G88, G89)	7 - 8
7-9	Example of Canned Cycle Program	7 - 9
7-10	Inducement of the Screen	7 - 10
7-11	Whole Circumference Cutting G302, G303	7 - 22
7-12	Whole Circumference Cutting (High-Speed Feeding Interval Present) G302, G303	7 - 23
7-13	Surrounding Whole Circle Cutting G304, G305	7 - 24
7-14	Example of Program for Whole Circle Cutting G302, G304 ...	7 - 25
7-15	Square Side Outer Circumference Cutting G322, G323	7 - 26

7-16	Herical Cutting G02, G03 (Option)	7 - 27
7-17	Programmable Mirror Image G511, G501 (Option)	7 - 28
7-18	Direct Tapping	7 - 29
7-19	Boring Pattern Cycle G70, G71, G72, G77 (Option)	7 - 33
7-20	Bolt-Hole Cycle G70 (Option)	7 - 35
7-21	Arc Cycle G71 (Option)	7 - 36
7-22	Line at Angle Cycle G72 (Option)	7 - 37
7-23	Grid Cycle G77 (Option)	7 - 38
7-24	Surface Cutting Cycle G324, G325 and G326 (Option)	7 - 39
7-25	Pocket Cutting Cycle G327 ~ G333 (Option)	7 - 49
7-26	Setting of Surface Cutting, Pocket Cutting and Finish Cutting Conditions (Option)	7 - 65
8.	Other Functions	8 - 1
8-1	Optional Block Skip	8 - 1
9.	Practical Example of Program	9 - 1
9-1	Machining Diagram Plate FC30	9 - 1
9-2	Selection of Machining Position	9 - 2
9-3	Setting of Selected Cutting Condition of Tool-Cutter	9 - 2
9-4	Mounting Method	9 - 3
9-5	Relation with Work-Coordinate System	9 - 3
9-6	φ95 Face Cutter	9 - 4
9-7	φ30 Boring	9 - 6
9-8	φ18 Center	9 - 7
9-9	φ8.5 Drill	9 - 8
9-10	M10 Tapping	9 - 9
9-11	φ30 Boring Finishing	9 - 10
9-12	Program of 2 Spindles	9 - 11
10.	Attached Lists	10 - 1
10-1	List for G-Functions (Preparatory Functions)	10 - 1
10-2	List for M-Functions (Miscellaneous Functions)	10 - 4
10-3	Related Items to the Tool-Set	10 - 8
10-4	How to Obtain the Cutting Conditions	10 - 9
10-5	List for Standard Cutting Conditions	10 - 10
10-6	List for Tape Code	10 - 11