

HM-043

**PROGRAMMING MANUAL
FOR
VM, VK TYPE
MACHINING CENTER
SEIKI-SEICOS Λ 10M
Edition 1 9-1996**

HITACHI SEIKI

27163

TABLE OF CONTENTS

INTRODUCTION	1-1
1-1 FLOW CHART FOR MACHINING WORK BY MACHINING CENTER	1-2
1-2 Programming type	1-3
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	2-5
2-7 Data	2-6
2-8 Word	2-6
2-9 Block	2-6
2-10 Sequence No.	2-7
2-11 How to prepare sequence No.	2-7
2-12 Tape dimension specification	2-8
2-13 Tape code	2-8
2-14 Tape format	2-9
2-15 Address and meaning	2-10
2-16 Program zero point and coordinate system	2-11
2-17 Absolute command (absolute coordinate value)	2-12
2-18 Incremental command (incremental value)	2-13
2-19 Right hand perpendicularly crossing coordinate system.	2-14
2-20 Z-axis	2-14
2-21 X-axis	2-14
2-22 Y-axis	2-15
2-23 Z-axis zero point	2-15
2-24 X, Y, Z standard coordinate and actual work	2-16
3. M, S, T, F, B functions	3-1
3-1 Miscellaneous (M-function)	3-1
3-2 Command method of pallet change (For VK with APC)	3-2

3-3 Command method of spindle speed (S-function)	3-4
3-4 Tool No. call (T-function)	3-5
3-5 Programming example of tool change (Case of VM, VK)	3-7
3-6 Command method of feed speed (F-function)	3-8
4. G-function (preparatory function)	4-1
4-1 G △△△	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 Exact stop (G09)	4-10
4-11 G17, G18, G19 (plane indication)	4-11
4-12 G27 (zero-point return check)	4-12
4-13 G28 (auto-zero return)	4-13
4-14 G29 (auto-return from zero point)	4-14
4-15 G30 (2nd, 3rd, 4th reference point return)	4-15
4-16 G31 (skip function)	4-16
5. G-function (length compensation,diameter compensation, position compensation)	5-1
5-1 Philosophy of tool length compensation (G43, G44, G49)	5-1
5-2 Tool length compensation (G43, G44, G49)	5-2
5-3 Tool diameter compensation G41, G42, G40	5-5
5-4 Tool diameter compensation G41, G42, G40	5-6
5-5 Summary of tool diameter compensation	5-7
5-6 G41, G42 (start-up)	5-8
5-7 G41, G42 (offset mode)	5-10
5-8 G40 (cancel)	5-12
5-9 Example of tool diameter compensation program	5-15
5-10 Example of tool diameter compensation program	5-16
5-11 Example of tool diameter compensation program	5-17

5-12 Example of tool diameter compensation program	5-18
5-13 Example of tool diameter compensation program	5-19
5-14 Example of tool diameter compensation program	5-20
5-15 Tool diameter compensation vector keep (G38), Offset vector change and Tool radius compensation corner circular arc (G39)	5-21
5-16 Tool compensation by tool number	5-24
5-17 Tool position offset (G45, G46, G47, G48)	5-27
5-18 Example using the positional offset for tool radius	5-29
5-19 Example using the tool-position offset for milling machining	5-30
 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 Addition of work coordinate system pair number (G540 ~ G599)	6-6
6-7 Selection of machine-coordinate system (G53)	6-10
6-8 Local coordinate system (G52)	6-11
6-9 G52 program example	6-12
6-10 Data setting (G10)	6-13
6-11 One directional positioning (G60)	6-16
 7. G-function (canned cycle)	7-1
7-1 Canned cycle (G73 ~ G89)	7-1
7-2 Ganned 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, G80, G81)	7-7
7-6 Canned cycle (G82, G83)	7-8
7-7 Canned cycle (G84, G85, G86)	7-11
7-8 Canned cycle (G87, G88, G89)	7-12
7-9 Example of canned cycle program	7-13
7-10 Herical cutting G02, G03	7-14
7-11 Programmable mirror image G511, G501	7-15

7-11 Programmable mirror image G511, G501	7-15
7-12 Setting mirror image	7-17
7-13 Direct tapping	7-19
7-14 Boring pattern cycle (G70, G71, G72, G77)	7-21
7-15 Bolt-hole cycle (G70)	7-22
7-16 Arc cycle (G71)	7-23
7-17 Line at angle cycle (G72)	7-24
7-18 Grid cycle (G77)	7-25
7-19 True circle cutting (G302 ~ G305)	7-26
7-20 Square side frame outer cutting (G322, G323)	7-33
7-21 Coordinate rotation (G68, G69)	7-36
7-22 Surface cutting cycle (G324, G325, G326)	7-40
7-23 Pocket cutting (G327~G333)	7-52
8. Other function	8-1
8-1 Optional block skip	8-1
8-2 Arbitrary angle chambering and corner R (, C, R)	8-2
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 ϕ 95 Face cutter	9-4
9-7 ϕ 30 Boring	9-5
9-8 ϕ 18 Center	9-6
9-9 ϕ 8.5 Drill	9-7
9-10 M10 Tapping	9-8
9-11 ϕ Boring finishing	9-9
9-12 Program of 2 spindles	9-10
10. Attached list	10-1
10-1 List of G function (preparatory function)	10-1
10-2 List for M function (miscellaneous functions) (VM, VK)	10-6
10-3 Related items to the tool-set	10-12

10-4 How to obtain the cutting condition	10-13
10-5 List for standard cutting conditions	10-14
10-6 List for tape code	10-15
10-7 MC MACHINE DATA	10-18
10-8 TOOLING LIST	10-19
10-9 If Alarm is Issued	10-20