GE Fanuc Automation

Computer Numerical Control Products

Series 0 / 00 / 0-Mate

Descriptions Manual

GFZ-61392E/03 August 1992

Warnings, Cautions, and Notes as Used in this Publication

Warning

Warning notices are used in this publication to emphasize that hazardous voltages, currents, temperatures, or other conditions that could cause personal injury exist in this equipment or may be associated with its use.

In situations where inattention could cause either personal injury or damage to equipment, a Warning notice is used.

Caution

Caution notices are used where equipment might be damaged if care is not taken.

Note

Notes merely call attention to information that is especially significant to understanding and operating the equipment.

This document is based on information available at the time of its publication. While efforts have been made to be accurate, the information contained herein does not purport to cover all details or variations in hardware or software, nor to provide for every possible contingency in connection with installation, operation, or maintenance. Features may be described herein which are not present in all hardware and software systems. GE Fanuc Automation assumes no obligation of notice to holders of this document with respect to changes subsequently made.

GE Fanuc Automation makes no representation or warranty, expressed, implied, or statutory with respect to, and assumes no responsibility for the accuracy, completeness, sufficiency, or usefulness of the information contained herein. No warranties of merchantability or fitness for purpose shall apply.

©Copyright 1992 GE Fanuc Automation North America, Inc.

All Rights Reserved.

CONTENTS

GENERAL

I. CNC FUNCTIONS

1. OUTLINE . . . . . 1 - 1

2. CONTROLLED AXES . . . . . 1 - 18
2.1 Controlled Axes . . . . . 1 - 18
2.2 Cf Axis Control . . . . . 1 - 19
2.3 Cs Axis Control . . . . . 1 - 19
2.4 Axes Control by PMC . . . . . 1 - 20
2.5 Change of Axis Name . . . . . 1 - 20

3. PREPARATORY FUNCTION (G FUNCTION) . . . . . 1 - 21

4. INTERPOLATION FUNCTIONS . . . . . 1 - 26
4.1 Positioning (G00) . . . . . 1 - 26
4.2 Linear Interpolation (G01) . . . . . 1 - 26
4.3 Circular Interpolation (G02, G03) . . . . . 1 - 26
4.4 Polar Coordinate Interpolation (G112, G113) . . . . . 1 - 27
4.5 Cyrindrical Interpolation (G107) . . . . . 1 - 28
4.6 Helical Cutting (G02, G03) . . . . . 1 - 29
4.7 Single Direction Positioning (G60) . . . . . 1 - 29

5. THREAD CUTTING . . . . . 1 - 30
5.1 Thread Cutting (G32) . . . . . 1 - 30
5.2 Variable-lead Thread Cutting (G34) . . . . . 1 - 30
5.3 Continuous Thread Cutting . . . . . 1 - 31

6. FEED FUNCTIONS . . . . . 1 - 32
6.1 Rapid Traverse . . . . . 1 - 32
6.2 Cutting Feed Rate . . . . . 1 - 32
6.2.1 Tangential speed constant control . . . . . 1 - 32
6.2.2 Cutting feed rate clamp . . . . . 1 - 32
6.2.3 Feed per minute (G94) . . . . . 1 - 32
6.2.4 Feed per revolution (G99) . . . . . 1 - 33
6.2.5 Synchronous feed (G95) . . . . . 1 - 33
6.2.6 F1-digit feed . . . . . 1 - 33
6.3 Override . . . . . 1 - 34
6.3.1 Feed rate override . . . . . 1 - 34
6.3.2 Rapid traverse override . . . . . 1 - 34
6.3.3 Jog override . . . . . 1 - 34
6.3.4 Override cancel . . . . . 1 - 34
6.3.5 Automatic corner override (G62) . . . . . 1 - 34
6.4 Automatic Acceleration/Deceleration . . . . . 1 - 34
6.5 Linear Acceleration/Deceleration After Cutting Feed Interpolation . . . . . 1 - 35
6.6 Dwell (G04) . . . . . 1 - 35
6.7 Exact Stop (G09) . . . . . 1 - 35
6.8 Exact Stop Mode (G61) . . . . . 1 - 35
6.9 Cutting Mode (G64) . . . . . 1 - 36
6.10 Tapping Mode (G63) . . . . . 1 - 36

7. REFERENCE POINT . . . . . 1 - 37
7.1 Manual Reference Point Return . . . . . 1 - 37
7.2 Automatic Reference Point Return (G28, G29) . . . . . 1 - 37
7.3 Reference Point Return Check (G27) . . . . . 1 - 37

7.4 2nd, 3rd and 4th Reference Point Return (G30) . . . . . 1 - 37

8. COORDINATE SYSTEM . . . . . 1 - 39
8.1 Coordinate System Setting . . . . . 1 - 39
8.2 Coordinate System Shift . . . . . 1 - 39
8.3 Automatic Coordinate System Setting . . . . . 1 - 39
8.4 Setting Work Coordinate System-method by G54-G59 . . . . . 1 - 40
8.5 Work Coordinate System Shift . . . . . 1 - 40
8.6 Direct Measured Value Input for Work Coordinate System Shift . . . . . 1 - 40

9. COORDINATE VALUE AND DIMENSION . . . . . 1 - 41
9.1 Absolute and Incremental Programming (G90, G91) . . . . . 1 - 41
9.2 Polar Coordinate Command (G15, G16) . . . . . 1 - 41
9.3 Inch/Metric Conversion (G20, G21) . . . . . 1 - 41
9.4 Decimal Point Input/Pocket Calculator Type Decimal Point Input . . . . . 1 - 42
9.5 Diameter and Radius Programming . . . . . 1 - 42

10. SPINDLE FUNCTIONS . . . . . 1 - 43
10.1 S Code Output . . . . . 1 - 43
10.2 S Analog Voltage Output . . . . . 1 - 43
10.3 Constant Surface Speed Control (G96, G97) . . . . . 1 - 43
10.4 Spindle Speed Clamp (G50) . . . . . 1 - 44
10.5 S Analog Voltage Control by PMC . . . . . 1 - 44
10.6 Spindle Speed Override . . . . . 1 - 44
10.7 Actual Spindle Speed Output . . . . . 1 - 44
10.8 Spindle Speed Fluctuation Detecting Function (G25, G26) . . . . . 1 - 44
10.9 Multi-spindle Control . . . . . 1 - 45
10.10 Serial Spindle . . . . . 1 - 46
10.11 Cs Contour Control . . . . . 1 - 46
10.12 Spindle Synchronization Control . . . . . 1 - 47

11. TOOL FUNCTIONS . . . . . 1 - 48
11.1 T Code Output . . . . . 1 - 48
11.2 Tool Life Management . . . . . 1 - 48
11.3 Simplified Tool Life Management . . . . . 1 - 48
11.4 Automatic Tool Length Measurement (G37) . . . . . 1 - 50

12. MISCELLANEOUS FUNCTIONS . . . . . 1 - 51
12.1 Miscellaneous Function (M2 digit) . . . . . 1 - 51
12.2 Second Miscellaneous Function (B Function) . . . . . 1 - 51

13. PROGRAM CONFIGURATION . . . . . 1 - 52
13.1 Program Number . . . . . 1 - 52
13.2 Main Program . . . . . 1 - 52
13.3 Sub Program . . . . . 1 - 52
13.4 Sequence Number . . . . . 1 - 52
13.5 Tape Codes . . . . . 1 - 52
13.6 Basic Addresses and Command Value Range . . . . . 1 - 53
13.7 Tape Format . . . . . 1 - 54
13.8 Label Skip . . . . . 1 - 55
13.9 Control-in/Control-out . . . . . 1 - 55
13.10 Optional Block Skip . . . . . 1 - 55
13.11 Program Name . . . . . 1 - 56
13.12 F10/11 Tape Format . . . . . 1 - 56
13.13 Directory Display of the Floppy Cassette . . . . . 1 - 56

14. FUNCTIONS TO SIMPLIFY PROGRAMMING . . . . . 1 - 57
14.1 Canned Cycles (G73, G74, G76, G80 - G89, G98, G99) (M series) . . . . . 1 - 57

14.2 Canned Cycles . . . . . 1 - 60
14.2.1 Cutting cycle A (G90) . . . . . 1 - 60
14.2.2 Thread cutting cycle (G92) . . . . . 1 - 61
14.2.3 Cutting cycle B (G94) . . . . . 1 - 61
14.3 Multiple Repetitive Canned Cycle (G70-G76) . . . . . 1 - 62
14.3.1 Rough cutting cycle of outer diameter (G71) . . . . . 1 - 62
14.3.2 Rough facing cycle (G72) . . . . . 1 - 62
14.3.3 Pattern repeating (G73) . . . . . 1 - 63
14.3.4 Finish cycle (G70) . . . . . 1 - 63
14.3.5 Deep hole drill cycle (G74) . . . . . 1 - 63
14.3.6 Outer diameter grooving cycle (G75) . . . . . 1 - 64
14.3.7 Threading cycle (G76) . . . . . 1 - 65
14.4 Chamfering and Corner R . . . . . 1 - 66
14.5 Optional Angle Corner Rounding . . . . . 1 - 67
14.6 Optional Angle Chamfering . . . . . 1 - 67
14.7 Mirror Image for Double Turrets (G68, G69) . . . . . 1 - 67
14.8 Direct Drawing Dimensions Programming . . . . . 1 - 68
14.9 External Motion Function (G80, G81) . . . . . 1 - 70
14.10 Conversational Programming with Graphic Function . . . . . 1 - 70
14.11 Menu Programming . . . . . 1 - 71
14.12 Scaling (G50,G51) . . . . . 1 - 72
14.13 Coordinate System Rotation (G68,G69) . . . . . 1 - 72
14.14 Canned Cycle for Drilling . . . . . 1 - 73
14.15 Rigid Tap . . . . . 1 - 73
14.16 Grinding Canned Cycle . . . . . 1 - 74
14.16.1 Grinding canned cycle (GCC) . . . . . 1 - 74
14.16.2 Grinding canned cycle (GSC) . . . . . 1 - 75
14.17 Conversational Input Function with Figure for Cylindlical Grinding Machine (Order-made macro) . . . . . 1 - 81
14.18 Angular Axis Control (GCC, GSC) . . . . . 1 - 82
14.19 Normal Direction Control (GSC) . . . . . 1 - 82
14.20 Infeed Control . . . . . 1 - 84
14.21 Multi-step Skip . . . . . 1 - 84

15. COMPENSATION FUNCTIONS . . . . . 1 - 86
15.1 Tool Offset (G43, G44, G49) (T series) . . . . . 1 - 86
15.2 Tool Length Compensation (G43, G44, G49) (M series) . . . . . 1 - 86
15.2.1 Tool length compensation A . . . . . 1 - 86
15.2.2 Tool length compensation B . . . . . 1 - 86
15.3 Cutter Compensation B, C (G39 - G42) (M series) . . . . . 1 - 86
15.4 Tool Offset Number . . . . . 1 - 87
15.5 Tool Offset Value Program Input (G10) . . . . . 1 - 87
15.6 Tool Geometry Offset and Tool Wear Offset . . . . . 1 - 88
15.7 Tool Nose Radius Compensation (G40-G42) (T series) . . . . . 1 - 88
15.8 Stored Pitch Error Compensation . . . . . 1 - 89
15.9 Backlash Compensation . . . . . 1 - 89
15.10 Y Axis Offset (T series) . . . . . 1 - 89
15.11 Continuous Dressing (GSC) . . . . . 1 - 89

16. MEASUREMENT FUNCTION . . . . . 1 - 91
16.1 Skip Function (G31) . . . . . 1 - 91
16.2 High-speed Skip Signal Input . . . . . 1 - 91
16.3 Tool Length Measurement . . . . . 1 - 91
16.4 Automatic Tool Offset . . . . . 1 - 91
16.5 Direct Measured Value Input A of Tool Offset . . . . . 1 - 92
16.6 Offset Value Counter Input . . . . . 1 - 93
16.7 Direct Measured Value Input B of Tool Offset . . . . . 1 - 93

17. CUSTOM MACRO ..... 1 - 95
17.1 Custom Macro A ..... 1 - 95
17.2 Custom Macro B ..... 1 - 97
17.3 Custom Macro Modal Call ..... 1 - 99
17.4 Pattern Data Input ..... 1 - 100
17.5 Order Made Macro ..... 1 - 100
17.6 Macro Executer ..... 1 - 101
17.7 Interruption Type Custom Macro ..... 1 - 101

18. AXIS CONTROL ..... 1 - 102
18.1 Mechanical Handle Feed ..... 1 - 102
18.2 Servo Off ..... 1 - 102
18.3 Mirror Image (M Series) ..... 1 - 102
18.4 X-Axis Mirror Image (T series) ..... 1 - 102
18.5 Additional Axis Neglect ..... 1 - 102

19. AUTOMATIC OPERATION ..... 1 - 103
19.1 Operation Mode ..... 1 - 103
19.2 Selection of Execution Program ..... 1 - 103
19.3 Activation of Automatic Operation ..... 1 - 104
19.4 Execution of Automatic Operation ..... 1 - 104
19.5 Automatic Operation Stop ..... 1 - 104
19.5.1 Program stop (M00, M01) ..... 1 - 104
19.5.2 Program end (M02, M30) ..... 1 - 105
19.5.3 Feed hold ..... 1 - 105
19.5.4 Thread cutting cycle retract ..... 1 - 105
19.5.5 Reset ..... 1 - 106
19.5.6 Sequence number comparison and stop ..... 1 - 106
19.6 Manual Interruption during Automatic Operation ..... 1 - 106
19.6.1 Handle interruption ..... 1 - 106
19.6.2 Position signal output ..... 1 - 107
19.6.3 Manual absolute ON/OFF ..... 1 - 107
19.7 Program Restart ..... 1 - 107

20. MANUAL OPERATION ..... 1 - 108
20.1 Manual Feed ..... 1 - 108
20.2 Incremental Feed ..... 1 - 109
20.3 Manual Handle Feed ..... 1 - 110

21. PROGRAM TEST FUNCTIONS ..... 1 - 111
21.1 Machine Lock ..... 1 - 111
21.2 Auxiliary Function Lock ..... 1 - 111
21.3 Dry Run ..... 1 - 111
21.4 Single Block ..... 1 - 112
21.5 Z-axis Command Cancel ..... 1 - 112

22. CRT/MDI ..... 1 - 113
22.1 CRT/MDI Panel ..... 1 - 113
22.2 Setting and Display ..... 1 - 119
22.2.1 MDI Operation/MDI Operation B ..... 1 - 119
22.2.2 Setting and display of offset value ..... 1 - 120
22.2.3 Setting and display of setting data ..... 1 - 120
22.2.4 Setting and display of parameter ..... 1 - 121
22.2.5 Setting and display of custom macro variables ..... 1 - 121
22.2.6 Program display ..... 1 - 122
22.2.7 Program list display ..... 1 - 122
22.2.8 Current position display ..... 1 - 123
22.2.9 Alarm display ..... 1 - 124

22.2.10 Command value display . . . . . 1 - 125
22.2.11 Run time display/parts count display . . . . . 1 - 125
22.3 Software Operator's Panel . . . . . 1 - 126
22.4 General Purpose Switch for Software Operator's Panel . . . . . 1 - 127
22.5 Program Protect Key . . . . . 1 - 127
22.6 Graphic Display . . . . . 1 - 127
22.7 Dinamic graphic display . . . . . 1 - 128
22.8 Language . . . . . 1 - 128
22.9 Created Tool Post Name Display (TT) . . . . . 1 - 128

23. PART PROGRAM STORAGE & EDITING . . . . . 1 - 129
23.1 Part Program Storage & Editing . . . . . 1 - 129
23.2 Number of Registered Programs . . . . . 1 - 129
23.3 Part Program Storage Length . . . . . 1 - 129
23.4 Play Back . . . . . 1 - 129
23.5 Graphic Conversational Input . . . . . 1 - 129
23.6 Pattern Data Input . . . . . 1 - 129
23.7 Menu Programming . . . . . 1 - 129
23.8 I/O Device External Control . . . . . 1 - 129
23.9 Background Editing . . . . . 1 - 130
23.10 Simple Synchronous Control . . . . . 1 - 130
23.11 Expanded Part Program Editing . . . . . 1 - 130
23.12 Input/Output Simultaneous Operation . . . . . 1 - 131
23.13 Operation When Converting the Machining Program Prepared by Graphic Conversation to an NC Format Program and Outputting Externally Via the Reader/puncher Interface . . . . . 1 - 131

24. HIGH-SPEED MACHINING FUNCTION . . . . . 1 - 132

25. POLIGONAL TURNING . . . . . 1 - 133

26. SELF DIAGNOSTIC FUNCTIONS . . . . . 1 - 134

27. DATA INPUT/OUTPUT . . . . . 1 - 135
27.1 I/O Device Interface . . . . . 1 - 135
27.2 Input Devices . . . . . 1 - 135
27.2.1 FANUC FLOPPY CASSETTE . . . . . 1 - 135
27.2.2 FANUC PPR . . . . . 1 - 135
27.2.3 Portable tape reader . . . . . 1 - 136
27.2.4 FANUC PROGRAM FILE Mate . . . . . 1 - 136
27.2.5 FANUC FA Card . . . . . 1 - 136

28. SAFETY FUNCTIONS . . . . . 1 - 137
28.1 Emergency Stop . . . . . 1 - 137
28.2 Overtravel Functions . . . . . 1 - 137
28.2.1 Overtravel . . . . . 1 - 137
28.2.2 Stored stroke limit . . . . . 1 - 137
28.2.3 External setting function of stored stroke limit . . . . . 1 - 137
28.2.4 Expanded Stored Stroke Check . . . . . 1 - 138
28.3 Interlock . . . . . 1 - 138
28.4 External Deceleration . . . . . 1 - 138

29. STATUS OUTPUT . . . . . 1 - 139
29.1 CNC Ready Signal . . . . . 1 - 139
29.2 Servo Ready Signal . . . . . 1 - 139
29.3 Alarm Signal . . . . . 1 - 139
29.4 Distribution End Signal1 . . . . . 1 - 139
29.5 Automatic Operation Signal . . . . . 1 - 139

29.6 Automatic Operation Start Lamp Signal . . . . . 1 - 139
29.7 Feed Hold Signal . . . . . 1 - 139
29.8 Power Ready Signal . . . . . 1 - 139
29.9 Reset Signal . . . . . 1 - 139
29.10 Battery Alarm Signal . . . . . 1 - 139
29.11 Clock Function . . . . . 1 - 139
29.12 Position Switch Function . . . . . 1 - 139

30. EXTERNAL DATA INPUT . . . . . 1 - 141
30.1 External Tool Compensation . . . . . 1 - 141
30.2 External Message Function . . . . . 1 - 141
30.3 External Key-Input Function . . . . . 1 - 141
30.4 External Data Input . . . . . 1 - 141
30.5 External Machine Origin Shift . . . . . 1 - 142

31. EXTERNAL WORK NUMBER SEARCH . . . . . 1 - 143

32. MACHINE INTERFACE . . . . . 1 - 144

33. PROGRAMMABLE MACHINE CONTROLLER (PMC) . . . . . 1 - 145

34. CONTROL UNIT . . . . . 1 - 146
34.1 Cabinet . . . . . 1 - 146
34.2 Power . . . . . 1 - 146
34.2.1 Input power voltage . . . . . 1 - 146
34.2.2 Input unit . . . . . 1 - 146
34.2.3 Multi-tap transformer . . . . . 1 - 146
34.3 Battery unit . . . . . 1 - 146
34.4 Environmental Requirements . . . . . 1 - 147
34.5 Weight . . . . . 1 - 147

35. CABLES AND CONNECTORS . . . . . 1 - 148

36. SERVO SYSTEM (Digital Servo) . . . . . 1 - 149
36.1 Position Detection System . . . . . 1 - 149
36.2 Position Detector . . . . . 1 - 149
36.3 Servo Motor . . . . . 1 - 149
36.4 Velocity Control Unit . . . . . 1 - 149
36.5 Regenerative Electric Discharge Unit . . . . . 1 - 149
36.6 Servo Power Transformer . . . . . 1 - 149

37. POSITION CODER . . . . . 1 - 150

38. FUNCTION FOR 4-AXIS LATHES (0-TTC) . . . . . 1 - 151
38.1 General . . . . . 1 - 151
38.2 Balance Cut (G68, G69) . . . . . 1 - 152
38.3 Interference Check Function . . . . . 1 - 152

39. MACHINE OPERATOR'S PANEL . . . . . 1 - 153

40. REMOTE BUFFER . . . . . 1 - 154
40.1 Remote Buffer . . . . . 1 - 154
40.2 High-Speed Remote Buffer A . . . . . 1 - 154
40.3 High-Speed Remote Buffer B . . . . . 1 - 155

II. CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION (0-TC, 0-TF, 0-MC, 0-MF, 0-Mate MF)

1. GENERAL . . . . . 2 - 1

2. GRAPHIC CONVERSATION FOR 2-AXES LATHE (0-TF) . . . . . 2 - 2
   2.1 Menu Screen . . . . . 2 - 3
   2.2 Arbitrary Profile Machining Programs . . . . . 2 - 3
   2.3 Set up Guidance Display . . . . . 2 - 3
   2.4 Automatic Selection of Tools, Cutting Conditions and Pre-tools . . . . . 2 - 4
   2.5 Dynamic Graphic Display (for lathes) . . . . . 2 - 5

3. GRAPHIC CONVERSATION FOR MACHINING CENTER (0-Mate MF/0-MF) . . . . . 2 - 6
   3.1 Keytouch . . . . . 2 - 6
   3.2 Menu Screen . . . . . 2 - 7
   3.3 Optimum Tool Order . . . . . 2 - 8
   3.4 Multi-piece Machining . . . . . 2 - 8
   3.5 Arbitrary Profile Machining Programs . . . . . 2 - 8
   3.6 Automatic Selection of Cutting Conditions and Pre-tools . . . . . 2 - 9
   3.7 Dynamic Graphic Display (For machining center) . . . . . 2 - 10

APPENDIX

APPENDIX 1. TABLE OF RANGE OF COMMAND VALUE . . . . . A - 1
APPENDIX 2. FUNCTION AND TAPE FORMAT . . . . . A - 7
APPENDIX 3. LIST OF CODE . . . . . A - 13
APPENDIX 4. EXTERNAL DIMENSIONS . . . . . A - 16

GENERAL

This manual describes the following products:

Name of products	Abbreviation	
FANUC Series 0-TC	0-TC	Series 0
FANUC Series 0-MF	0-TF	
FANUC Series 0-TTC	0-TCC	
FANUC Series 0-GCC	0-GCC	
FANUC Series 0-MC	0-MC	
FANUC Series 0-MF	0-MF	
FANUC Series 0-GSC	0-GSC	
FANUC Series 00-TC	00-TC	Series 00
FANUC Series 00-MC	00-MC	
FANUC Series 00-GCC	00-GCC	
FANUC Series 00-GSC	00-GSC	
FANUC Series 0-Mate TC	0-Mate TC	Series 0-Mate
FANUC Series 0-Mate MC	0-Mate MC	
FANUC Series 0-Mate MF	0-Mate MF	

I. CNC FUNCTIONS

1. OUTLINE

The Series 0/00/0-Mate are high-performance CNCs developed to provide the outstanding speed, precision and efficiency demanded to accomplish the machining of the future and all in a remarkably compact machine configuration. They achieve a substantial boost in machining productivity thanks to their high-speed, high-precision digital servo system, high-speed PMC functions, and abundant CNC functions made possible through the adoption of a high-speed microprocessor.

Seires 0/0 Mate includes following models :

0-Mate MC / 0-MC

For machining center, drilling machine and milling machine

0-Mate MF / 0-MF

Conversational type for machining center, drilling machine and milling machine

0-Mate TC / 0-TC

For all-purpose lathe, automatic lathe and lathe controlled by programmable controller

0-TF

Conversational type for all-purpose lathe, automatic lathe and lathe controlled by programmable controller

0-TTC

For 4-axis lathe such as single spindle and 2 tool posts, twin spindles and 2 tool posts

00-TC, 00-MC

It is a 3-in 1 CNC comprising MMC, CNC and PMC. Functions of 0-TC or 0-MC are adopted for this CNC.

0-GCC/00-GCC

For cylindrical grinding function

0-GSC/00-GCC

For surface grinding function

CNC functions for the above eight models are explained in this chapter. See chapter II for Conversational Programming Functions. Also see the operator's manuals of MMC for details of MMC functions. A high-speed micro processor and many custom LSIs have been employed for the control circuit, successfully decreasing the number of circuit elements to the greatest extent. Therefore, its reliability is much higher, and a small control unit with a power supply unit can be easily built into the magnetic power cabinet on the machine. Construction of the control unit is of panel mount type which allows easy construction of an integrated system of machinery with an electrical unit. A simple, heavy current circuit on the machine also allows construction of more compact, united systems of machine and electrical units, because a PMC (programmable machine cont-rol) can be in corporated into the control unit.

High-speed, high-accuracy machining can be done since the digital servo having a high-speed processor and custom LSIs exclusively employed for it is used in the control circuit of the servo system.

As for the drive unit, it can be made smaller than conventional types by using transformer.

As for the control motor, an absolute position detector of a genuinely electronic type can be fitted on the detector for an AC servo motor. No operation of reference point return is required when power is supplied, by employing an absolute position detector of full electronic type.

Therefore, restarting of operation after cut-off the power supply can be easily done.

The spindle motor for CNC is an AC spindle motor which is small and of high power.

High-speed, high-accuracy machining can be done since the motor is digitally controlled.

○;Standard ☆; Option ★;Option, available for TTC 1st tool post

*;Function included in another option

Table 1(a) Function list (1/5)

Name		Specification	Series 0							Series 0 Mate			Ref. item
			MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
Controlled axis		2 axes	–	–	–	○	○	–	○	–	–	○	2.1
		3 axes	○	○	○	☆	☆	–	☆	○	○	–	2.1
		4 axes	☆	☆	☆	☆	☆	–	☆	☆	☆	–	2.1
		2 axes + 2 axes	–	–	–	–	–	○	–	–	–	–	2.1
		3 axes / 4 axes + 2 axes	–	–	–	–	–	☆	–	–	–	–	2.1
Simultaneous controllable axes		2 axes	○	○	○	○	○	–	○	○	○	○	2.1
		3 axes / 4 axes	☆	☆	☆	☆	☆	–	☆	☆	☆	–	2.1
		2 axes + 2 axes	–	–	–	–	–	○	–	–	–	–	2.1
		3 axes / 4 axes + 2 axes	–	–	–	–	–	☆	–	–	–	–	2.1
Cf-axis control			–	–	–	☆	☆	★	☆	–	–	–	2.2
Cs-axis control			–	–	–	☆	☆	★	☆	–	–	–	2.3
Y-axis control (T)			–	–	–	☆	☆	★	☆	–	–	–	2.1
Axis control by PMC		4 axes(T) / 3 axes(M) / 2 axes(TT)	☆	☆	☆	☆	☆	★	☆	–	–	–	2.4
Axis name alteration			–	–	–	–	–	–	☆	–	–	–	2.5
Angular axis control			–	–	☆	–	–	–	☆	–	–	–	14.18
Normal direction control			–	–	☆	–	–	–	–	–	–	–	14.19
Continuous dressing control			–	–	☆	–	–	–	–	–	–	–	15.11
Infeed control			–	–	☆	–	–	–	–	–	–	–	14.20
Least input increment			○	○	○	○	○	○	○	○	○	○	2.1
Least command increment			○	○	○	○	○	○	○	○	○	○	2.1
Increment system 1/10		0.1 μm	☆	☆	☆	☆	☆	☆	☆	–	–	–	2.1
Max.programmable dimensions		±8 digits	○	○	○	○	○	○	○	○	○	○	2.1
Rapid traverse rate			○	○	○	○	○	○	○	○	○	○	6.1
Rapid traverse override		Fo,25,50,100%	○	○	○	○	○	○	○	○	○	○	6.3.2
Range of feedrate	Feed per minute (/min)		○	○	○	○	○	○	○	○	○	○	6.2.3
	Feed per revolution (/rev)		☆	☆	☆	○	○	○	○	–	–	○	6.2.4
Automatic acceleration /deceleration			○	○	○	○	○	○	○	○	○	○	6.4
Linear acceleration/deceleration after cutting feed interporation			☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	6.5
Rapid traverse rate		0~150%	○	○	○	○	○	○	○	○	○	○	6.3.1
Jog override			☆	☆	☆	–	–	–	–	–	–	–	6.3.3

Table 1(a) Function list (2/5)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
Override cancel		○	○	○	○	○	○	○	○	○	○	6.3.4
Manual continuous feed		○	○	○	○	○	○	○	○	○	○	20.1
Manual synchronous feed		–	–	–	○	○	○	○	–	–	○	20.1
Positioning		○	○	○	○	○	○	○	○	○	○	4.1
Single direction positioning		☆	☆	☆	–	–	–	–	–	–	–	4.7
Linear interpolation		○	○	○	○	○	○	○	○	○	○	4.2
Circular interporation		○	○	○	○	○	○	○	○	○	○	4.3
Polar coordinate command		☆	☆	☆	–	–	–	–	☆	☆	–	9.2
Polar coordinate interporation		–	–	–	☆	☆	★	☆	–	–	–	4.4
Cylindrical interpolation		–	–	–	☆	☆	★	☆	–	–	–	4.5
Helical interpolation		☆	☆	☆	–	–	–	–	☆	☆	–	4.6
Thread cutting, synchronous feed		☆	☆	☆	○	○	○	○	–	–	○	5.1
Thread cutting retract		–	–	–	☆	☆	☆	–	–	–	–	19.5.4
Continuous thread cutting		–	–	–	☆	☆	☆	☆	–	–	–	5.3
Variable lead thread cutting		–	–	–	☆	☆	☆	☆	–	–	–	5.2
F1-digit feed		☆	☆	☆	–	–	–	–	–	–	–	6.2.6
Reference point return		○	○	○	○	○	○	○	○	○	○	7
Reference point return check		○	○	○	○	○	○	○	○	○	○	7
2nd reference point return		○	○	○	○	○	○	○	○	○	○	7
3rd/4th reference point return		☆	☆	☆	☆	☆	☆	☆	–	–	–	7
9"CRT/MDI(small type)	High-definition monochrome	○	○	○	○	○	○	○	○	○	○	22.1
Thin-profile display /MDI (small type)	Monochrome	☆	–	☆	☆	–	☆	☆	☆	–	☆	22.1
9"CRT/MDI(small type)	Color	☆	–	☆	☆	–	☆	☆	–	–	–	22.1
9"CRT/MDI(full key)	High-definition monochrome	☆	☆	☆	☆	☆	☆	☆	–	–	–	22.1

Table 1(a) Function list (3/5)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
9"CRT/MDI(full key)	color	☆	–	☆	☆	–	☆	☆	–	–	–	22.1
14"CRT/MDI(full key)	color	☆	☆	☆	–	–	–	–	–	–	–	22.1
Softkey for MDI	5 + 2pieces (For 9")	☆	○	☆	☆	○	☆	☆	☆	○	☆	22.1
	10 + 2 pieces (For 14")	*	*	*	–	–	–	–	–	–	–	22.1
Manual handle feed	1 unit	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	20.3
	2 units	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	20.3
	3 units	*	*	*	–	–	–	–	*	*	–	20.3
Manual handle feed rate	×1, ×10, ×M	*	*	*	*	*	*	*	*	*	*	20.3
Handle interruption		☆	☆	☆	☆	☆	☆	☆	–	–	–	19.6.1
Jog and handle simultaneous mode		○	○	○	○	○	○	○	○	○	○	20.1
Incremental feed	×1, ×10, ×100, ×1000	○	○	○	○	○	○	○	○	○	○	20.2
Rigid tapping		☆	☆	☆	–	–	–	–	☆	☆	–	14.15
Program restart		☆	☆	☆	☆	☆	☆	☆	☆	☆	–	19.7
Reader/puncher interface	Reader / puncher (Ch.1) / ASR 33 interface	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	27.1
	Reader / puncher (Ch.2) interface	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	27.1
Dwell(per sec)		○	○	○	○	○	○	○	○	○	○	6.6
Interlock		○	○	○	○	○	○	○	○	○	○	28.3
Machine lock		○	○	○	○	○	○	○	○	○	○	21.1
Stored stroke check 1		○	○	○	○	○	○	○	○	○	○	28.2.2
Stroke limit external setting		☆	☆	☆	–	–	–	–	–	–	–	28.2.3
Expanded stored stroke check		☆	☆	☆	☆	○	☆	☆	–	–	–	28.2.4

Table 1(a) Function list (4/5)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
Tool post interference check		–	–	–	–	☆	–	–	–	–	–	38.3
Z-axis command cancel		○	○	○	–	–	–	–	○	○	–	21.5
Additional axis neglect		*	*	*	–	–	–	–	*	*	–	18.5
Mechanical handle feed		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	18.1
Skip function		○	○	○	○	○	○	○	○	○	○	16.1
High speed skip function		☆	☆	☆	☆	☆	★	☆	–	–	–	16.2
External deceleration		☆	☆	☆	☆	☆	★	☆	–	–	–	28.4
Position signal output		–	–	–	☆	☆	☆	☆	–	–	–	19.6.2
Battery alarm output		○	○	○	○	○	○	○	○	○	○	29.10
Exact stop mode		○	○	○	–	–	–	–	○	○	–	6.8
Exact stop		○	○	○	○	○	○	○	○	○	○	6.7
Servo off		○	○	○	○	○	○	○	○	○	○	18.2
Manual absolute on/off		○	○	○	○	○	○	○	○	○	○	19.6.3
Chamfering on/off		–	–	–	○	○	○	○	–	–	○	14.2.2
Backlash compensation		○	○	○	○	○	○	○	○	○	○	15.9
Stored pitch error compensation		☆	☆	☆	☆	☆	☆	☆	☆	☆	–	15.8
Clock function		☆	☆	☆	☆	☆	☆	☆	–	–	–	29.11
Position switch		☆	☆	☆	☆	☆	☆	☆	–	–	–	29.12
MDI operation		○	○	○	○	○	○	○	○	○	○	22.2.1
MDI operation B		☆	☆	☆	☆	☆	☆	☆	–	–	–	22.2.1
Y-axis offset (T)		–	–	–	☆	☆	★	☆	–	–	–	15.10
Analog signal input	4 points	☆	☆	☆	☆	☆	☆	☆	–	–	–	15.12
High-speed cycle machining		☆	–	☆	☆	–	–	☆	–	–	–	24
Simple synchronous control		–	–	–	☆	☆	★	☆	–	–	–	23.10

Table 1(a) Function list (5/5)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
Order made macro	64KB	☆	–	☆	☆	–	★	☆	☆	–	☆	17.5
	128KB/256KB/512KB	☆	–	☆	☆	–	★	☆	–	–	–	17.5
Macro executer	64KB/128KB	☆	☆	☆	☆	☆	★	☆	☆	☆	☆	17.6
	256KB/512KB	☆	–	☆	☆	–	★	☆	–	–	–	17.6
Reset		○	○	○	○	○	○	○	○	○	○	19.5.5
Dry run		○	○	○	○	○	○	○	○	○	○	21.3
Single block		○	○	○	○	○	○	○	○	○	○	21.4
Program protect signal		○	○	○	○	○	○	○	○	○	○	22.5
Self-diagnosis function		○	○	○	○	○	○	○	○	○	○	26
Emergency stop		○	○	○	○	○	○	○	○	○	○	28.1
Status display		○	○	○	○	○	○	○	○	○	○	29
Position coder	4000rpm / 6000rpm	☆	☆	☆	○	○	○	○	☆	☆	○	37
Control unit external dimensions	Control unit A 250×400×185mm (W) (H) (D)	○	○	○	○	○	○	○	○	○	○	Appendix 4
	Control unit B 450×400×185mm (W) (H) (D)	○	○	○	○	○	○	○	–	–	–	Appendix 4
Power supply	AC200V + 10%,-15% 50/60Hz ± 1Hz AC220V + 10%,-15% 60Hz ± 1Hz	○	○	○	○	○	○	○	○	○	○	34.2
Connectable servo motor	FANUC AC SERVO MOTOR(Digital)	○	○	○	○	○	○	○	○	○	○	36
Connectable spindle motor	FANUC AC SPINDLE MOTOR etc.	○	○	○	○	○	○	○	○	○	○	

Table 1(b) Function list (Machine operator's panel)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
Software operator's panel		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	22.3
Software operator's panel general purpose switch		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	22.4
Standard machine operator's panel		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	39

Table 1(c) Function list (Position Detector)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
Incremental pulse coder interface		○	○	○	○	○	○	○	○	○	○	36.2
Absolute pulse coder interface		○	○	○	○	○	○	○	○	○	○	36.2

Table 1(d) Function list (PMC)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
PMC-L	6μs/step Display /editing of ladder 3000 steps 5000 steps	☆	☆	☆	☆	☆	–	☆	☆	☆	☆	33
	DI/DO Max.104/72 points	☆	☆	☆	☆	☆	–	☆	☆	☆	☆	33
PMC-M	2μs/step Display /editing of ladder 3000 steps 5000 steps 8000 steps 12000 steps	☆	☆	☆	☆	☆	☆	☆	–	–	–	33
	DI/DO Max.208/144 points	☆	☆	☆	☆	☆	☆	☆	–	–	–	33

Table 1(e) Function list (Program input) (1/2)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
Coordinate system setting		○	○	○	○	○	○	○	○	○	○	8.1
Automatic coordinate system setting		○	○	○	○	○	○	○	○	○	○	8.3
Work coordinate system		☆	☆	☆	–	–	–	–	☆	☆	–	8.4
Direct input of work coordinate system shift amount measured		○	○	○	–	–	–	–	○	○	–	8.5 8.6
Decimal point input		○	○	○	○	○	○	○	○	○	○	9.4
Pocket calculator type decimal point input		○	○	○	○	○	○	○	○	○	○	9.4
Direct drawing dimension programming		–	–	–	☆	○	☆	☆	–	–	–	14.8
Special G code input		–	–	–	☆	–	☆	–	–	–	–	3
Programming input of offset data		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	15.5
Custom macro A		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	17.1
Custom macro B		☆	☆	☆	☆	☆	☆	☆	–	–	–	17.2
Interruption type custom macro		☆	☆	☆	☆	☆	☆	☆	–	–	–	17.7
Chamfering / Corner R		–	–	–	☆	○	☆	☆	–	–	☆	14.4
Optional angle chamfering / corner R		☆	○	☆	–	–	–	–	–	–	–	14.5 14.6
Inch/ metric conversion		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	9.3
Cutter compensation B		☆	–	☆	–	–	–	–	–	–	–	15.5
Cutter compensation C		☆	○	☆	–	–	–	–	☆	○	–	15.5
Tool nose radius compensation		–	–	–	☆	○	☆	☆	–	–	☆	15.7
Canned cycles for grinding		–	–	–	○	○	○	○	–	–	○	14.2
Multiple repetitive cycles		–	–	–	☆	○	☆	–	–	–	☆	14.3
Grinding canned cycle		–	–	–	☆	–	–	☆	–	–	–	14.16
Canned cycles for drilling		☆	○	☆	☆	☆	★	☆	☆	○	–	14.14
X-axis diameter / radius programming		–	–	–	○	○	○	○	–	–	○	9.5

Table 1(e) Function list (Program input) (2/2)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
Playback		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	23.4
Multi step skip		–	–	–	–	–	–	☆	–	–	–	
Counter input of offset value		–	–	–	○	○	○	○	–	–	○	16.6
Radius designation on arc		○	○	○	○	○	○	○	○	○	○	4.3
EIA/ISO automatic recognition		○	○	○	○	○	○	○	○	○	○	13.5
Mirror image		○	○	○	–	–	–	–	○	○	–	18.3
Mirror image for double turret		–	–	–	☆	☆	☆	–	–	–	–	14.7
X-axis mirror image		–	–	–	☆	☆	☆	–	–	–	–	18.4
Automatic corner override		☆	☆	☆	–	–	–	–	–	–	–	6.3.5
Scaling		☆	☆	☆	–	–	–	–	☆	☆	–	14.12
Coordinate system rotation		☆	☆	☆	–	–	–	–	☆	☆	–	14.13
Menu programming		☆	☆	☆	☆	☆	☆	☆	–	–	–	14.11
Tape format for F10/11		☆	☆	–	☆	☆	☆	☆	–	–	–	13.12
Pattern data input		☆	–	☆	–	–	–	☆	–	–	–	17.4
Conversational programming with graphic function		☆	○	–	☆	☆	–	☆	☆	○	–	23.5
Graphic conversation for machining center		☆	○	–	–	–	–	–	–	○	–	II -2
Graphic conversation for 2-axis lathe		–	–	–	☆	○	–	–	–	–	–	II -1
NC statement output		☆	☆	–	☆	☆	–	–	–	–	–	23.13
Graphic conversation for cylindrical grinding machine	Order-made macro	–	–	–	–	–	–	☆	–	–	–	14.17
Polygon turning		–	–	–	☆	☆	★	☆	–	–	–	25
Balance cut		–	–	–	–	–	☆	–	–	–	–	38.2

Table 1(f) Function list (Function Auxoiliary function)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
Miscellaneous function	M-3 digit	○	○	○	○	○	○	○	○	○	○	12.1
2nd auxiliary function	B-6 digit	☆	☆	☆	–	–	–	–	–	–	–	12.2
	B-8 digit	–	–	–	☆	☆	☆	☆	–	–	–	12.2
Miscellaneous function lock		○	○	○	○	○	○	○	○	○	○	21.2
Tool post waiting function		–	–	–	–	–	○	–	–	–	–	38.1

Table 1(g) Function list (Spindle function)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
S-2 digit		○	–	○	○	–	○	○	○	–	○	10.1
S-4 / S-5	Spindle analog output	☆	○	☆	☆	○	☆	☆	☆	○	☆	10.1
Constant surface speed control		☆	☆	☆	☆	○	☆	☆	–	–	☆	10.3
Spindle speed override	50～120%	*	*	*	*	*	*	*	*	*	*	10.6
Spindle gear selection	4 stage(T)/ 3 stages(M)	*	*	*	*	*	*	*	–	–	–	10.1
Actual spindle speed output		–	–	–	☆	☆	☆	☆	–	–	☆	10.7
Spindle speed fluctuation detection		–	–	–	☆	☆	☆	☆	–	–	☆	10.8
Analog voltage control by PMC		*	*	*	*	*	*	*	*	*	*	10.5
Multi-spindle control		–	–	–	☆	☆	–	☆	–	–	–	10.9
Serial spindle		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	10.10
Cs axis contouring control		☆	☆	☆	☆	☆	☆	☆	–	–	–	10.11
Spindle synchronous control		☆	☆	☆	☆	☆	☆	☆	–	–	–	10.12

Table 1(h) Function list (Tool function)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
Tool function	T1 + 1/T2 + 2	–	–	–	○	○	○	○	–	–	○	11.1
	T2/T4	○	○	○	–	–	–	–	○	○	–	11.1
Tool offset memory	± 6 digits, 32 pieces	○	○	○	–	–	–	–	○	○	–	15.4
	64 pieces/99 pieces /200 pieces	☆	☆	☆	–	–	–	–	☆	☆	–	15.4
	± 6 digits, 9/16 pairs	–	–	–	○	○	○	○	–	–	○	15.4
	32 pairs	–	–	–	☆	☆	☆	☆	–	–	–	15.4
Tool offset memory B		☆	☆	☆	–	–	–	–	–	–	–	
Tool length compensation		○	○	○	–	–	–	–	○	○	–	15.2
Tool offset		☆	☆	☆	○	○	○	○	–	–	○	15.1
Tool geometory/wear offset		–	–	–	☆	○	☆	☆	–	–	☆	15.6
Simple tool life management		–	–	–	☆	☆	☆	☆	–	–	–	11.3
Tool life management		☆	☆	☆	–	–	–	–	–	–	–	11.2
External offset		☆	☆	☆	☆	☆	☆	☆	–	–	–	30.1
Tool length automatic measurement		☆	☆	☆	–	–	–	–	–	–	–	11.4
Automatic tool offset		–	–	–	☆	☆	☆	☆	–	–	–	16.4
Direct input of offset value measured A		–	–	–	○	○	○	○	–	–	○	16.5
Direct input of offset value measured B (Tool setter)		–	–	–	☆	☆	–	☆	–	–	–	16.7

Table1(i) Function list(Editing / Operation)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
Part program storage length	10m	○	–	○	○	–	○	○	○	–	○	23.3
	20m / 40m / 80m	☆	–	☆	☆	–	☆	☆	☆	–	☆	23.3
	120m	☆	○	☆	☆	○	★	☆	☆	○	–	23.3
	320m	☆	☆	☆	☆	☆	★	☆	–	–	–	23.3
Registered programs	63 pieces	○	○	○	○	○	○	○	○	○	○	23.2
	125 / 200 pieces	☆	☆	☆	☆	☆	☆	☆	☆	☆	–	23.2
Sequence number search		○	○	○	○	○	○	○	○	○	○	19.2
Program number search		○	○	○	○	○	○	○	○	○	○	19.2
Optional block skip	1 pieces	○	○	○	○	○	○	○	○	○	○	13.10
	9 pieces	☆	☆	☆	☆	☆	☆	☆	–	–	–	13.10
Sequence number comparison and stop		*	*	*	*	*	*	*	*	*	–	19.5.6
External work number search	15 pieces	○	○	○	○	○	○	○	○	○	○	31
Program protect		○	○	○	○	○	○	○	○	○	○	22.5
Background editing		☆	○	☆	☆	○	☆	☆	☆	○	–	23.9
Extended part program editing		☆	☆	☆	☆	☆	☆	☆	–	–	–	23.11

Table1(j) Function list(Display)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
Multi-language display	English	○	○	○	○	○	○	○	○	○	○	22.8
	Japanese	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	22.8
	German / French / Chinese / Korean / Italian / Spanish	☆	☆	☆	☆	☆	☆	☆	☆	☆	–	22.8
Run hour and parts count display		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	22.2.11
Display of spindle speed and T code at all screens		*	*	*	○	○	○	○	–	–	○	10.6
Actual speed display		○	○	○	○	○	○	○	○	○	○	22.2.8
Directory display of floppy cassette		☆	☆	☆	☆	☆	★	☆	☆	☆	–	13.13
Graphic display		☆	–	☆	☆	○	☆	☆	☆	–	–	22.6
Dynamic graphic display		☆	○	☆	☆	○	–	–	–	○	–	22.6
Optional tool post name display		–	–	–	–	–	○	–	–	–	–	22.9

Table1(k) Function list(External data input / output)

Name	Specification	Series 0							Series 0 Mate			Ref. item
		MC	MF	GSC	TC	TF	TTC	GCC	MC	MF	TC	
External key input		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	30.3
External tool offset		☆	☆	☆	☆	☆	☆	☆	–	–	–	30.1
External message		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	30.2
External data input		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	30.4
External machine origin shift		☆	☆	☆	☆	☆	☆	☆	–	–	–	30.5
External work number search	15 pieces	○	○	○	○	○	○	○	○	○	○	31
External I/O device control		☆	☆	☆	☆	☆	☆	☆	–	–	–	23.8
Remote buffer		☆	☆	☆	☆	☆	–	☆	–	–	–	40.1
High-speed remote buffer A		☆	☆	☆	☆	☆	–	☆	–	–	–	40.2
High-speed remote buffer B		☆	☆	☆	–	–	–	–	–	–	–	40.3
Input/Output simultaneous operation		☆	☆	☆	–	–	–	–	–	–	–	23.12
External machine zero point shift		☆	☆	☆	☆	☆	☆	☆	–	–	–	30.4
FANUC FA Card	20/80/160 m	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	27.2.5
FANUC FLOPPY CASSETTE	2,500m	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	27.2.1
FANUC PROGRAM FILE Mate	50,000m, 100,000m	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	27.2.4
FANUC PPR		☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	27.2.2

Table 2 Configuration of Axes

T series

1st axis	2nd axis	3rd axis	4th axis	S/P	5th axis	6th axis
X	Z			S/P		
X	Z	PMC axis	(PMC axis)	S/P		
X	Z	PMC axis	PMC axis	S/P	(PMC axis)	(PMC axis)
X	Z			Cs		
X	Z		PMC axis	Cs	(PMC axis)	(PMC axis)
X	Z	Cf	(PMC axis)	S/P		
X	Z	Cf	PMC axis	S/P	(PMC axis)	(PMC axis)
X	Z	Cf	Y	S/P	(PMC axis)	(PMC axis)

In case of 4 axes lather

1st axis	2nd axis	3rd axis	4th axis	S1/P1	5th axis	6th axis	S2/P2
X1	Z1			S1/P1	X2	Z2	S2/P2
X1	Z1	PMC axis	(PMC axis)	S1/P1	X2	Z2	S2/P2
X1	Z1	PMC axis	PMC axis	S1/P1	X2	Z2	S2/P2
X1	Z1			Cs	X2	Z2	S2/P2
X1	Z1		PMC axis	Cs	X2	Z2	S2/P2
X1	Z1	Cf		S1/P1	X2	Z2	S2/P2
X1	Z1	Cf	PMC axis	S1/P1	X2	Z2	S2/P2
X1	Z1	Cf	Y	S1/P1	X2	Z2	S2/P2

M series

1st axis	2nd axis	3rd axis	4th axis	5th axis	6th axis
X	Y	Z	(PMC axis)	(PMC axis)	(PMC axis)
X	Y	Z	4TH	(PMC axis)	(PMC axis)

(Note 1) Interpolation between PMC axis and main axes (X, Z, Cf, Y axes) is impossible. (T series)

Interpolation between PMC axis and main axes (X, Y, Z, 4th axes) is impossible.(M series)

(Note 2) Axis configurations other than the above are impossible.The axes in parentheses can be omitted.

(Note 3) When PMC axis of 5th axis and 6 th axis is used, specify PMC-M.

(Note 4) Axis is controlled in Mate series.

In case of high speed cycle machining,axis configuration is as following.

In case of 0-TC

1st axis	2nd axis	3rd axis	4th axis	S/P	5th axis	6th axis	7th axis	8th axis
X1	Z	Cf	(PMC axis)	S/P			Y(High speed axis)	
X1	Z	Cf	PMC axis	S/P	(PMC axis)	(PMC axis)	Y(High speed axis)	
X(High speed axis)		Z	(PMC axis)	S/P				
X (High speed axis)		Z	PMC axis	S/P	(PMC axis)	(PMC axis)		

In case of 0-MC

1st axis	2nd axis	3rd axis	4th axis	S/P	5th axis	6th axis	7th axis	8th axis
X(High speed axis)		Y(High speed axis)			(PMC axis)	(PMC axis)	Z	(4TH)
X(High speed axis)		Y(High speed axis)		S/P	(PMC axis)	(PMC axis)	Z	(4TH)
X(High speed axis)		Y	Z		(PMC axis)	(PMC axis)	(4TH)	
X(High speed axis)		Y	Z	S/P	(PMC axis)	(PMC axis)	(4TH)	

(Note 1) In high speed cycle machining , the following options cannot be specified.

- Part prograsm length more than 120m
- Cs axis control (only T series)

·Manual handle Pulse generator interpolation

(Note 2) In high speed cycle machining mode , acceleration / deceleration control is not be performed.

(Note 3) Machining data is compiled by macro compiler.Specify with macro executer.

2. CONTROLLED AXES

2.1 Controlled Axes

(1) Basic Controlled Axes
3 axes (T series)
4 axes (M series)

(2) Controlled axes expansion
4 axes (Total:6 axes, including PMC axis)

(3) Basic Simultaneously Controlled Axes
(a) Automatic operation (Rapid traverse, cutting feed):2 axes
(b) Manual operation:1 axis

(4) Simultaniously controlled axes expansion
(a) Automatic operation (rapid traverse, cutting feed):
3 axes(simultaneous control of optional 3 axes among X, Y, Z and 4th axes)
4 axes (simultaneous control of 4 axes; X, Y, Z and 4th axes)

(5) Name of Axes
X, Y, Z, 4th (A, B,C, U, V, W) (Note 2)

(6) Increment System

(T series)

Least input increment			0.001 mm	0.0001 inch
Least command increment (Note 1)	X axis	Diameter designation	0.0005 mm	0.00005 inch
		Radius designation	0.001 mm	0.0001 mm
	Z axis		0.001 mm	0.0001 mm
	C axis (Note 3)		0.001 deg	0.001 deg
	Y axis		0.001 mm	0.0001 mm

In increment system 1/10 (T series)

Least input increment			0.0001 mm	0.00001 inch
Least command increment (Note 1)	X axis	Diameter designation	0.00005 mm	0.000005 inch
		Radius designation	0.0001 mm	0.00001 mm
	Z axis		0.0001 mm	0.00001 mm
	C axis (Note 3)		0.0001 deg	0.0001 deg
	Y axis		0.0001 mm	0.00001 mm

(M series)

Least input increment		0.001mm	0.0001inch
Least command increment (Note 1)	X,Y,Z axis	0.001mm	0.0001inch
	4th axis (Note2)	0.001deg	0.001deg
		0.001mm	0.0001inch

In increment system 1/10 (T series)

Least input increment		0.0001mm	0.00001inch
Least command increment (Note 1)	X,Y,Z axis	0.0001mm	0.00001inch
	4th axis (Note 2)	0.0010deg	0.0001deg
		0.0001mm	0.00001inch

(7) Maximum stroke
±99999.999 mm
±9999.9999 inch
±9999.9999 deg
In increment system 1/10
±9999.9999 mm
±999.99999 inch
±99999.999 deg

(Note 1) The least command increment is the least increment of command which CNC gives to the machine.

(Note 2) For X axis specifications, either diameter designation or radius designation is selectable by parameter.

(Note 3) The 4th axis can be used also as a linear axis, but the following functions cannot be performed.

. Circular interpolation
. Helical cutting
. Cutter compensation
. Tool length compensation

(Note 4) C axis is controlled by a feed motor (Cf axis)

2.2 Cf Axis Control

(1) Input increment

Least input increment	0.001 deg
Least command increment	0.001 deg
For increment system 1/10	
Least input increment	0.0001 deg
Least command increment	0.0001 deg

(2) Simultaneous controllable axes number
Simultaneous two axes only for linear interpolation
(Simultaneous one axis in manual feed)
Simultaneous three/four axes only for linear interpolation optionally.

(3) Max. command value
±99999.999 deg
For increment system 1/10
±9999.9999 deg

(4) Feed rate

Rapid traverse	100000 deg/min
Cutting feed	100000 deg/min
For increment system 1/10	
Rapid traverse	24000 deg/min
Cutting feed	12000 deg/min

(5) Command address

Absolute command	C
Incremental command	H

2.3 Cs Axis Control

Spindle indexing can be controlled by a spindle motor

(1) Least command increment

$$\frac{360}{4096} = 0.088\text{deg}$$

(2) Command method
M code, address C or address H

2.4 Axis Control by PMC

Independent axes can be controlled by the command from PMC, not the command from CNC.

(1) Axes controllable by PMC
The 3rd axis, 4th axis, 5th axis and 6 axis (T series)
1 axis among 1st to 4th axes, 5th axis and 6 axis (M series)
However,the axis used as a CNC axis cannot be controlled by PMC.
When PMC axis in 5th axis and 6 th axis more than 1 axis is used,specify PMC-M without fail.

(2) Axis name for CRT position display
An alphabet which is not used on CNC axis can be selected by a parameter.
It can also be selected, using a parameter whether or not the axis name is displayed on the CRT position display unit.

(3) No. of simultaneously controlled axes
The axis controlled by PMC operates independently of the CNC axis and other axes controlled by PMC.

(4) Incrementsystem

Least input increment	0.001mm	0.0001inch
Least command increment	0.001mm	0.0001inch

For increment system 1/10, the following increment systems are also available. However, there is a limit as follows.
. Cutting feedrate 6.5 m/min Max.
. Rapid traverse rate 10 m/min Max.

Least input increment	0.0001mm	0.00001inch
Least command increment	0.0001mm	0.00001inch

2.5 Change of Axis Name

The program axis name for the 3rd and 4th axes can be modified by selecting from among U, V, W, A, B, C and Y, using a parameter.

3. PREPARATORY FUNCTION (G FUNCTION)

(1) T series

G code list (1/2)

G code system(Note 7)			Group	Function
A	B	C		
G00 ☆	G00 ☆	G00 ☆	01	Positioning (rapid traverse)
G01	G01	G01		Linear Interpolation (feed)
G02	G02	G02		Circular interpolation CW
G03	G03	G03		Circular interpolation CCW
G04	G04	G04		Dwell
G10	G10	G10		Data setting
G20	G20	G70	06	Inch data input
G21	G21	G71		Metric data input
G22 ☆	G22 ☆	G22 ☆	09	Stored stroke limit function ON
G23	G23	G23		Stored stroke limit function OFF
G25 ☆	G25 ☆	G25 ☆	08	Spindle speed fluctuation detect OFF
G26	G26	G26		Spindle speed fluctuation detect ON
G27	G27	G27	00	Reference point return check
G28	G28	G28		Return to reference point
G30	G30	G30		2nd, 3rd, 4th reference point return
G31	G31	G31		Skip cutting
G32	G33	G33	01	Thread cutting
G34	G34	G34		Variable-lead thread cutting
G36	G36	G36	00	Automatic tool compensation X
G37	G37	G37		Automatic tool compensation Z
G40 ☆	G40 ☆	G40 ☆	07	Tool nose radius compensation cancel
G41	G41	G41		Tool nose radius compensation left
G42	G42	G42		Tool nose radius compensation right
G50	G92	G92	00	Coordinate system setting, max. spindle speed setting
G65	G65	G65		Macro calling
G66	G66	G66	12	Macro modal call
G67	G67	G67		Macro modal call cancel
G68	G68	G68	04	Mirror image for double turrets ON
G69 ☆	G69 ☆	G69 ☆		Mirror image for double turrets OFF

G code list (2/2)

G code system(Note 7)			Group	Function
A	B	C		
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing
G73	G73	G75		Pattern repeating
G74	G74	G76		Peck drilling on Z axis
G75	G75	G77		Grooving on X axis
G76	G76	G78		Multiple threading cycle
G80 ☆	G80 ☆	G80 ☆	10	Cancelling canned cycle for drilling
G83	G83	G83		Canned cycle for drilling
G84	G84	G84		Canned cycle for drilling
G86	G86	G86		Canned cycle for drilling
G87	G87	G87		Canned cycle for drilling
G88	G88	G88		Canned cycle for drilling
G89	G89	G89		Canned cycle for drilling
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Thread cutting cycle
G94	G79	G24		Endface turning cycle
G96	G96	G96	02	Constant surface speed control
G97 ☆	G97 ☆	G97 ☆		Constant surface speed control cancel
G98	G94	G94	05	Feed per minute
G99 ☆	G95 ☆	G95 ☆		Feed per revolution
—	G90 ☆	G90 ☆	03	Absolute programming
—	G91	G91		Incremental programming
—	G98	G98	11	Return to initial level
—	G99	G99		Return to R point level

(2) M series

G code list (1/2)

G code	Group	Function
G00 ☆	01	Positioning (Rapid traverse)
G01 ☆		Linear interpolation (Cutting feed)
G02		Circular interpolation/Helical cutting CW
G03		Circular interpolation/Helical cutting CCW
G04	00	Dwell, Exact stop
G05		High speed cycle machining
G09		Exact stop
G10		Data setting
G11		Data setting mode cancel
G15	17	Polar coordinates command cancel
G16		Polar coordinates command
G17 ☆	02	XY plane selection
G18		ZX plane selection
G19		YZ plane selection
G20	06	Input in inch
G21		Input in mm
G22	00	Stored stroke check function on
G23		Stored stroke check function off
G27		Reference point return check
G28		Return to reference point
G29		Return from reference point
G30		2nd reference point return
G31		Skip function
G33	01	Thread cutting
G39	00	Corner offset circular interpolation
G40 ☆	07	Cutter compensation cancel
G41		Cutter compensation left
G42		Cutter compensation right
G43	08	Tool length compensation + direction
G44		Tool length compensation - direction
G45	00	Tool offset increase
G46		Tool offset decrease
G47		Tool offset double increase
G48		Tool offset double decrease
G49 ☆	08	Tool length compensation cancel
G50 ☆	11	Scaling cancel
G51		Scaling

G code list (2/2)

G code	Group	Function
G52	00	Local corodinate system setting
G53		Machine coordinate system selection
G54	14	Work coordinate system 1 selection
G55		Work coordinate system 2 selection
G56		Work coordinate system 3 selection
G57		Work coordinate system 4 selection
G58		Work coordinate system 5 selection
G59		Work coordinate system 6 selection
G60	00	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64 ☆		Cutting mode
G65	12	Macro command, Macro call
G66		Custom Macro modal call
G67 ☆		Custom Macro modal call cancel
G68	16	Coordinate rotation
G69		Coordinate rotation cancel
G73	09	Peck drilling cycle
G74		Counter tapping cycle
G76		Fine boring
G80 ☆		Canned cycle cancel
G81		Drilling cycle, spot boring
G82		Drilling cycle, counter boring
G83		Peck drilling cycle
G84		Tapping cycle
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
G90 ☆	03	Absolute command
G91		Incremental command
G92	00	Programming of absolute zero point
G94 ☆	05	Feed per minute
G95		Feed per rotation
G96	13	Constant surface speed control
G97		Constant surface speed control cancel
G98 ☆	10	Return to initial point in canned cycle
G99		Return to R point in canned cycle

(Note 1) G codes marked ☆are initial G codes when turning power on.
For G20 and G21, the G code before turning power off remains. G00 or G01 can be selected by parameter setting.

(Note 2) G codes of group 00 are not modal. They are only effective in the block in which they are specified.

(Note 3) If a G code not listed on the table of G codes is inputted, or optional G code not specified in the system is commanded, an alarm (No. 010) is displayed.

(Note 4) A number of G codes can be specified in the same block. When more than one G code of the same group is specified, the G code specified later is effective.

(Note 5) If any G code of group 01 is specified in a canned cycle mode, the canned cycle is automatically cancelled and the G80 condition is entered. However a G code of group 01 is not affected by any of the canned cycle G codes.

(Note 6) A G code is displayed from each group.

(Note 7) G code system B and C are options. Whether G code system B or C is set by parameter No. 0036 : GSPC.

4. INTERPOLATION FUNCTIONS

This manual uses the following notation.

P_ : Combination of optional axis address (of x, y, z, A, B, C) as X_ Y_ Z . . . .

; : End of block (LF for ISO code, CR for EIA code)

4.1 Positioning (G00)

- Positioning is done with each axis independently.
- Tool path generally does not become a line.
- It is decelerated to a stop at the end point, and inposition check is performed (checks whether the machine has come to the specified position).It is possible not to perform the inposition check by parameter. Width of inposition can be set as a parameter.

4.2 Linear Interpolation (G01)

4.3 Circular Interpolation (G02, G03)

(Note 1)

G01:Clockwise (CW) circular interpolation

G02: Counterclockwise (CCW) circular interpolation

(Note 2) Circular radius R designation:

R>0; 0° to 180° circular designatable

R<0; 180° to 360° circular designatable

I, J, K designation : 0°to 360°circular

(Note 3) If an R smaller than the half of the distance between the start point and the end point was specified, this R is neglected, and a 180° circular arc is drawn.

(Note 4) The feedrate in circular interpolation is equal to the cutting feedrate specified by an F code. An error of the command speed from actual tool speed is within ±2%.

(Note 5) If R is omitted, the tool moves straight toward the end point.

4.4 Polar Coordinate Interpolation (G112, G113)

The function in which contour control is done in converting the command programmed in a cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece) is the polar coordinate interpolation . It is an effective function when a cam shaft is commanded by a G code.

G112; Polar coordinate interpolation mode (Polar coordinate interpolation shall be done.)

G113; Polar coordinate interpolation cancel mode (Polar coordinate interpolation is not done)

The G codes shall be commanded in a single block.

(1) Polar coordinate interpolation mode(G112)

The axes (linear axis and rotary axis) on which polar coordinate interpolation is done are set beforehand by parameters.

Change the mode to polar coordinate interpolation mode by commanding G112, and a plane (hereinaffter referred to as polar coordinate interpolation plane) is selected in which linear axis is made to the first axis of the plane, and virtual axis being a right angle with the linear axis is made to the second axis of the plane . Polar coordinate interpolation is carried out on this plane.

In the polar coordinate interpolation mode, the command of linear interpolation (G01) and circular interpolation (G02, G03) is possible.

And both absolute command (G90) and incrtemental command (G91) are possible.

For the program command it is possible to apply cutter compensation.

For the path after cutter compensation is done, polar coordinate interpolation can be made.

As feedrate, specify the tangential speed (relative speed between the workpiece and the tool) on the polar coordinate interpolation plane (cartesian coordinate system) with F.

(2) Polar coordinate interpolation cancel mode (G113)

The polar coordinate interpolation cancel mode is obtained by G13.1 command.

(3) Example of a program

<u>Poalr coordinate interpolation by X axis (Linear axis) and C axis (Rotary axis).</u>

```
00001;
:
:
N100G90G00X120.0C0Z_;  Positioning to the starting
                       position
N200G112;   Starting polar coordinate interpolation
N201G42G01X40.0F D01;
N202C10.0;
N203G03X20.0C20.0R10.0;
N204G01X-40.0;
N205C-10.0;
N206G03X-20.0C-20.0I10.0K0
N207G01X40.0;
N208C0;
N209G40X120.0;
N210G113;
                Canceling polar coordinate
interpolation
N300Z_;
N400X_C_;
:
:
M30;
```

Contour program (Program in cartesian coordinate system of X-C plane)

4.5 Cylindrical Interpolation (G107)

When the form on the expanded side view of a cylinder (form on the cylinder coordinate system) is commanded by a program command, the NC converts the form into a linear axis movement and a rotary axis movement then performs a contour control. This feature is called the cylindrical interpolation. The cyclindrical interpolation is commanded with G107.

G107(Name of rotary axis) Radius of cylinder;
:Cylindrical interpolation mode

G107(Name of rotary axis) 0; :
Cancellation mode of cylindrical interpolation

(1) Cylindrical interpolation mode

Cylindrical interpolation is made between the rotary axis specified in the block of G107 and the other optional linear axis.

Circle interpolation command is allowed as well as linear interpolation, during cylindrical interpolation mode. Also, absolute command and incremental command can be made. Cutter compensation can be added to the program command. Cylindrical interpolation is made for the path after cutter compensation.

Feed rate gives the tangential speed on the expanded plane of the cylinder with F.

(2) Cancellation mode of cylindrical interpolation

G107 (Name of rotary axis) 0;

Cancellation mode of cylindrical interpolation is made when commanded as above.

(3) An example of a program

```
00001   (CYLINDRICALINTERPOLATION)
N1G00G90Z100.0C0;
N2G01G18Z0C0;
N3G107C57299;
N4G01G42Z120.0D01F250;
N5C40.0;
N6G02Z90.0C60.0R30.0;
N7G01Z70.0;
N8G03Z60.0C70.0R10.0;
N9G01C150.0;
N10G03Z70.0C190.0R75.0;
N11G01Z110.0C230.0;
N12G02Z120.0C270.0R75.0;
N13G01C360.0;
N14G40Z100.0;
N15G107C0;
N16M30;
```

4.6 Helical Cutting (G02, G03)

Helical interpolation is enabled by specifying another axis which moves synchronously with the circular interpolation by circular commands. That is, the tool can be moved helically.
The command method is to simply add a move command axis which is not circular interpolation axes, to the circular interpolation commands.
An F command specifies a feed rate along a circular arc. Therefore, the feed rate of the linear axis is as follows:

F×(Length of Linear line) / Length of circular arc

Determine the feed rate so the linear axis feed rate does not exceed any of the various limit values.

(Note 1) Cutter compensation is applied only for a circular arc.
(Note 2) Tool length compensation cannot be used in a block in which a helicalcutting is commanded.

4.7 Single Direction Positioning (G60)

For accurate positioning without backlash, final positioning from only one direction is available.

5. THREAD CUTTING

5.1 Thread Cutting (G32)

G32X(U)__Z(W)__F__;

	Least command increment	Lead command range
metric input	0.001mm	0.0001~ 5000000mm
inch input	0.0001inch	0.000001~ 9.999999inch

By feeding the tool synchronizing with the spindle rotation, thread cutting of the specified lead is performed. Specify the lead of the long axis direction with the F code.

(Note 1) Feed hold cannot be executed during thread cutting.
(Retract function is invailid in G32 thread cutting.)

(Note 2) Thread cutting is restricted as follows.

Maximum spindle revolutions :
4000 rpm (position coder A)
6000 rpm (position coder B)

Maximum feed rate :
15000 mm/min, 600 inch/min

P = R × F

P ≤ Maximum command value of feed rate per minute (15000mm/min, 600inch/min)

P ≤ Feed rate limited by motors employed

where,

P: Feed rate in threading (mm/min, inch/min)

F: Lead (mm. inch)

R: Spindle revolutions

5.2 Variable-lead Thread Cutting (G34)

Specifying an increment or a decrement value for a lead per screw revolution enables variable-lead thread cutting to be performed.

K specifies an increment or a decrement per screw revolution in the following command.

G34 **P** – F – K – ;

F : Lead in longitudinal axis direction at the start point

K : Increment and decrement of lead per spindle revolution

Addresses other than K are the same as in straight/taper thread cutting with G32.

K can take values of:

Metric input

±0.0001 – ±5000.000 mm/rev

Inch input

±0.00001 – ±9.999999 inches/rev

An alarm is informed, for example, when a K exceeding this value is directed, the maximum value of lead is exceeded as a result of increase or decrease by K or the lead has a negative value.

(Note) The "Thread Cutting Cycle Retract" is not effective for G34.

5.3 Continuous Thread Cutting

This function for continuous thread cutting (block overlap) is such that fractional pulses outputted to a joint between move blocks are overlapped with the next move for pulse processing and outputted. Therefore, discontinuous machining sections caused by the interruption of move during continuously block machining are eliminated, thus making it possible to continuously direct the block for thread cutting instructions.

Since the system is controlled in such a manner that the synchronism with the spindle does not deviate in the joint between blocks wherever possible, it is possible to perform special thread-cutting operation in which the lead and shape change midway.

Even when the same section is repeated for thread cutting while changing the depth of cut, this system allows a correct machining without impairing the threads.

(Note 1) Block overlap is effective even for G01 command, producing a more excellent finishing surface.

(Note 2) When extreme micro blocks continue, no block overlap may function.

6. FEED FUNCTIONS

6.1 Rapid Traverse

Least command increment	Rapid traverse rate range
0.001 mm	30-100000 mm/min
0.0001inch	3.0-4000.0 inch/min

In case of 1/10 increment system

Least command increment	Rapid traverse rate range
0.0001 mm	30-24000mm/min
0.00001inch	3.0-960.0inch /min

Positioning is done in rapid motion by the positioning command (G00).

There is no need to program rapid traverse rate, because the rates are set in the parameter (per axis).

The above feed rates are limits according to the CNC's interpolation capacity.

When the whole system is considered, there are also limits according to the servo system.

6.2 Cutting Feed Rate

6.2.1 Tangential speed constant control

In cutting feed, it is controlled so that speed of the tangential direction is always the same commanded speed.

6.2.2 Cutting feed rate clamp

Cutting feed rate upper limit can be set as parameters of each axis. If the actual cutting feed rate (feed rate with override) is commanded exceeding the upper limit, it is clamped to a speed not exceeding the upper limit of that value.

6.2.3 Feed per minute (G94)

Least command increment	Cutting feed rate range
0.001 mm	1 -100000mm/min
0.0001inch	1 -12000 mm/min

In case of 1/10 increment system

Least command increment	Cutting feed rate range
0.0001 mm	0.01- 4000.00inch /min
0.00001inch	0.01- 480.00inch/min

With the per minute feed mode G94, tool feed rate per minute is directly commanded by numerical value after F.
F can be set not to be commanded by setting the initial value of F in parameter.(F initial value setting).
The above feed rates are limits according to the CNC's interpolation capacity.
When the whole system is considered, there are also limits according to the servo system.

6.2.4 Feed per revolution (G99)

With the per revolution feed mode G99, tool feed rate per revolution of the spindle is directly commanded by numeral after F. A position coder must be mounted on the spindle.
The above feed rates are limits according to the CNC's interpolation capacity.
When the whole system is considered, there are also limits according to the servo system.

6.2.5 Synchronous feed (G95)

Least command increment	Range of cutting feed
0.001 mm	0.00001 -500.0000mm/rev
0.0001inch	0.000001-9.999999inch/rev

Per revolution feed mode is commanded by G95. Use the numeric value following F to directly command in order to set the amount of tool feed per revolution of the spindle.
A position coder must be installed on the spindle.
It is used for tapping, etc.
The above feed rate is a restriction by interpolation capacity of CNC. Note that the system is provided with the attached servo system for restriction.

6.2.6 F1-digit feed

When a 1-digit number from 1 to 9 is commanded after the F, the preset speed corresponding to the 1-digit number commanded is set as feed rate.
Set the F1-digit feed rate change input signal on from the machine side, and rotate the manual pulse generator. Feed rate of the currently selected speed will increase or decrease.
Feedrate set or changed will be memorized even after power is turned off.

6.3 Override

6.3.1 Feed rate override

The per minute feed (G94) can be overridden by:
0 to 150% (per every 10%)

6.3.2 Rapid traverse override

Rapid traverse rate can be overridden by:
Fo, 25%,50%, 100%.
Fo:A constant speed set by parameter

6.3.3 Jog override

Jog feedrate (Jog feed, step feed, dry run feed) can be overridden by 0 to 150%(per every 10%).

6.3.4 Override cancel

Feedrate override can be clamped in 100% by a machine side signal.

6.3.5 Automatic corner override (G62)

When a G62 is commanded during cutter compensation, cutting feedrate is automatically overridden at corner.The cutting quantity per unit time of the corner is thus controlled not to increase.This G62 is valid till G61 (exact stop mode), G64 (cutting mode), or G63 (tapping mode) is commanded.

6.4 Automatic Acceleration / Deceleration

Acceleration and deceleration is performed when starting and ending movement, resulting in smooth start and stop.
Automatic acceleration/deceleration is also performed when feed rate changes, so change in speed is also smoothly done.

Rapid traverse:
Linear acceleration/deceleration (time constant is set by a parameter per axis)

Cutting feed:
Exponential acceleration/deceleration (time constant is set by a parameter common to all axes)

Jog feed:
Linear acceleration/deceleration (time constant is set by a parameter per axis)

6.5 Linear Acceleration / Deceleration after Cutting Feed Interpolation

In the linear acceleration/deceleration, the delay for the command caused by the acceleration/deceleration becomes 1/2 compared with that in exponential acceleration/ decereration, substantially reducing the time required for acceleration and decereration.
Also, the radius direction error in the circular interpolation caused by the acceleration/ deceleration is substantially reduced.

Δr : Maximum value of radius error (mm)
v : Feedrate (mm/sec)
r : Circular radius(mm)
T_1 : Acceleration/deceleration time constant (sec)
T_2 : Time constant of servo motor (sec)

The maximum value of error in this radius direction is obtained approximately by the following equation.

$\Delta r = (T_1^2/2 + T_2^2/2)v^2/r$

For exponential acceleration/deceleration

$\Delta r = (T_1^2/24 + T_2^2/2)v^2/r$

For linear acceleration/deceleration after cutting feed interpolation

Consequently, In case of the linear acceleration /deceleration after interpolation, if an error caused by the servo loop time constant is excluded, the radius directional error will be reduced to 1/12, compared with the exponential acceleration/ deceleration.

6.6 Dwell (G04)

With the G04 command, shifting to the next block can be delayed by the time specified.

6.7 Exact Stop (G09)

Move command in blocks commanded with G09 decelerates at the end point, and inposition check is performed. G09 command is not necessary for deceleration at the end point for positioning and inposition check is also done automatically.This function is used when sharp edges are required for workpiece corners in cutting feed.

6.8 Exact Stop Mode (G61)

When G61 is commanded, deceleration of cutting feed command at the end point and inposition check is performed per block thereafter.This G61 is valid till G64(cutting mode), G62 (automatic corner override), or G63 (tapping mode) is commanded.

6.9 Cutting Mode (G64)

When G64 is commanded,deceleration at the end point of each block thereafter is not performed and cutting goes on to the next block.This command is valid till G61 (exact stop mode),G62 (automatic corner override), or G63 (tapping mode) is commanded.

6.10 Tapping Mode (G63)

When G63 is commanded, feed rate override is ignored (always regarded as 100%), and feed hold also becomes invalid. Cutting feed does not decelerate at the end of block to transfer to the next block. And in-tapping mode signal is issued during tapping operation.This G63 is valid till G61 (exact stop mode), G62 (automatic corner override), or G64 (cutting mode) is commanded.

7. REFERENCE POINT

7.1Manual Reference Point Return

Positioning to the reference point can be done by manual operation.

With jogging mode(J),manual reference point return (ZRN) signals, and signal for selecting manual reference point return axis (±x, ±Y, ±Z, ±4th) on, the tool begins to move at rapid traverse mode.When deceleration limit switch mounted on the machine is turned on, it decelerates, and when it is turned off again, it stops at the first grid point,and reference point return end lamp lights. This point is the reference point.

Grid method:

A certain grid of the position detector is appointed as the reference point.

The reference point position can be electrically shifted by the grid shift function.

7.2 Automatic Reference Point Return (G28, G29)

(1) Return to reference point (G28)

With the G28 command, the commanded axis is positioned to the reference point via the commanded point.After positioning,the reference point return end lamp is lit. If G28 was commanded when reference point return is not performed after power on, reference point return is done in the same sequence as the manual reference point return.

(2) Return from reference point (G29)

With the G29 command,the commanded axis is positioned to the point commanded by G29, via the intermediate point commaned by G28.

7.3 Reference Point Return Check (G27)

Program returned to reference point
↓
- Completion lamp is lit when tool returned to the reference point correctly
- Alarm is generated when tool does not return to the reference point

G27 P_; (M series)
G27 X(U)_Z(W)_; (T series)

This function is used to check whether the reference point return command was performed correctly.

When G27 is commanded, reference point return end signal is output if reference point return is performed to the correct position, and alarm arises if it is not positioned correctly to the reference point.

This function is available after power is turned on and reference point return is performed.

7.4 2nd, 3rd and 4th Reference Point Return (G30)

With the G30 command, the commanded axis is positioned to the 2nd, 3rd or 4th reference point. 2nd, 3rd or 4th reference point return end signal is output when positioning ends. Set the 2nd, 3rd or 4th reference point position as parameters.

This function is available after power is turned on and reference point return is performed. It is the same as reference point return G28 except tool returns to the 2nd, 3rd or 4th reference point. This command is usually used when the ATC position is different from reference point.

8. COORDINATE SYSTEM

8.1 Coordinate System Setting

A work coordinate system whose origin is at the specified distance (X, Y, and Z) from the current tool position is commanded by program. Once created, subsequent absolute commands refer to a coordinate value in this work coordinate system.

8.2 Coordinate System Shift

The coordinate system can be shifted by the specified amount by programming after it has once been set.

A new coordinate system can be produced by specifying a difference between the tool A position and the tool B position, using a program when tool A has been replaced with tool B in the above figure.

8.3 Automatic Coordinate System Setting

A coordinate system being preset by a parameter is set by returning a tool to the reference point by manual operation. As a result, the coordinate system is set as if G92; were specified at the reference point.

8.4 Setting Work Coordinate System-method by G54-G59

ZOFS1: Work zero point offset value of work coordinate system 1
ZOFS2: Work zero point offset value of work coordinate system 2
ZOFS3: Work zero point offset value of work coordinate system 3
ZOFS4: Work zero point offset value of work coordinate system 4
ZOFS5: Work zero point offset value of work coordinate system 5
ZOFS6: Work zero point offset value of work coordinate system 6

Setting work coordinate system

Six work coordinate systems can be set.

These six systems are decided by setting the distances of each axis from the machine zero point to the zero points of the coordinate systems, (i.e. the work zero point offset value, by using the CRT/MDI panel).

8.5 Work coordinate system shift

When the coordinate system actually set by the G50 command or the automatic coordinate system setting deviates from the programmed work coordinate system, the set coordinate system can be shifted.

Set the desired shift amount in the work coordinate system shift memory.

8.6 Direct measured value input for work coordinate system shift

When the work coordinate system set with a G50 command or the automatic coordinate system setting function is different from the coordinate system used in programming, surface A and B are cut and the coordinate system can be shifted by storing the measured distance directly as from CRT/MDI.

9. COORDINATE VALUE AND DIMENSION

9.1 Absolute and Incremental Programming (G90,G91)

Absolute command	Incremental command
G91 P__;	G90 P__;

There are two ways to command travels along the axes; the absolute command, and the incremental command. In the absolute command, coordinate value of the end point is programmed; in the incremental command, move distance of the axis itself is programmed.

9.2 Polar Coordinate Command (G15, G16)

The end point coordinate value can be input in polar coordinates (radius and angle). Use G15, G16 for polar coordinates command.

G15: Polar coordinate system command cancel

G16: Polar coordinate system command

Plane selection of the polar coordinates is done same as plane selection in circular interpolation, using G17, G18, G19.

Command radius in the first axis of the selected plane, and angle in the second axis. For example, when the X-Y plane is selected, command radius with address X, and angle with address Y. The plus direction of the angle is counterclockwise direction; the minus direction is the clockwise direction.

Both radius and angle can be commanded in either absolute or incremental command (G90,G91).

The center of the polar coordinates is the zero point of the local coordinates.

Example) Bolt hole cycle

N1 G17 G90 G16;

Polar coordinates command, X-Y plane

N2 G81 X100. Y30. Z-20. R-5. F200. ;

100mm radius, 30° angle

N3 X100. Y150 ;

100mm radius, 150° angle

N4 X100. Y270;

100mm radius, 270° angle

N5 G15 G80 ;

Polar coordinates cancel

9.3 Inch / Metric Conversion (G20,G21)

Conversion of inch and metric input can be commanded by the G code command.

G20 :Inch input

G21 :Metric input

Whether the output is in inch system or metric system is set by a parameter when the machine is installed.Command G20, G21 at the head of the program.
Inch/metric conversion can also be done by MDI setting.
The setting procedure differs according to G20 and G21.
This function cannot be used with the mechanical handle function.

9.4 Decimal Point Input / Pocket Calculator Type Decimal Point Input

Numerals can be input with decimal points. Decimal points can be used basically in numerals with units of distance, speed, and angle. The position of the decimal point is at the mm, inch, deg position.
Use parameters to select input method; whether to input by pocket calculator type input, or by the former type decimal point input.

9.5 Diameter and Radius Programming

Since the work cross section is usually circular in CNC lathe control programming, its dimensions can be specified in two ways:

When the diameter is specified, it is called diameter programming and when the radius is specified, it is called radius programming. Radius programming or diameter programming can be specified by parameter .

10. SPINDLE FUNCTIONS

10.1 S Code Output

The spindle speed is commanded in a 2 digit numeral with sign after the address S. S code signal and strobe signal (SF) is ouput to the machine by this command.

(Note) If S4 digits are commanded in S2 digits, the last two digits are effective.

10.2 S Analog Voltage Output

The spindle revolutions are specified by a 4-digit or 5-digit numeric in the wake of address S.(The circumferential speed can also be specified in case of the constant surface speed control.)

This system outputs an analog voltage to set the spindle speed to meet the specified speed according to S command. It also outputs S analog voltage valid signal ENBS according to M03 (spindle CW), M04 (spindle CCW) and M05 (spindle stop).

The analog voltage can be polarized by M03 or M04 (parameter setting).

10.3 Constant Surface Speed Control (G96, G97)

When the constant surface speed control is on, by directly commanding surface speed with the S, spindle speed in which spindle speed makes surface speed constant to the change of tool position (when absolute coordinate value of X-axis is regarded as the radius), it is output in analog voltage.

Command surface speed in m/min units for metric input, and feet/min units for inch input.

When the constant surface speed control is off, spindle speed (rpm) crresponding to an analog voltage proportional to the spindle motor speed is output to the machine side.

10.4 Spindle Speed Clamp (G50)

Spindle speed can be clamped by G50 S__; command.

10.5 S Analog Voltage Control by PMC

The S analog voltage is controllable by PMC. For this control, two interfaces are provided as follows.

(1) A 12-bit binary code signal corresponding to programmed S codes is output to PMC.

(2) A 12-bit binary code signal for S analog voltage is sent from PMC to CNC.

The spindle control is possible in the automatic/manual modes by returning the 12-bit binary code signal sent from CNC to the CNC input as it is in the automatic mode, or by producing a 12-bit signal corresponding to the spindle speed by PMC and then, sending the signal to CNC in the manual mode, for example.

10.6 Spindle Speed Override

The spindle speed override of 50% to 120% can be applied by 3-bit data produced by PMC software.

10.7Actual Spindle Speed Output

Actual spindle speed is calculated from the feed back pulse from the position coder mounted on the spindle. It is output in 16 bit binary code.

10.8Spindle Speed Fluctuation Detecting Function (G25, G26)

G26 Pp Qq Rr

Spindle speed fluctuation detection on

G25 ;

Spindle speed fluctuation detection off

p: Time (in msec.) for starting check when the commanded speed is not reached after a certain time.

q: Tolerance (%) at which the actual spindle speed is regarded to reach the commanded value when the speed is commanded or changed.
Check starts after the actual speed reaches the tolerance.

r : Spindle speed fluctuation (%) at which the system recognizes that seizure occurs. An alarm is indicated when the spindle speed changes beyond this value.

Alarm is indicated when the spindle speed deviates from the commanded speed according to the conditions of the machine, and an alarm signal (SPAL) is sent to machine side.
This function is useful to prevent seizure of the guide bushing, etc.

10.9 Multi-spindle Control

1) Outline

This multi-spindle control is used to control two spindles in addition to the normal spindle.
The spindle is commanded by one S4/5 digit command. The signal from PMC determines which spindle should be selected.
The spindle which can be added has two-step gear selection for each. One channel can be added as the position coder interface and the selection depends on the signal from PMC.

2) Control method

The rotation command by S command is output as analog voltage of each spindle by the spindle switching signals (SWS1 - SWS3). At the same time, output of two spindles or more can be made.
The spindle analog voltage selected by a switching signal holds the previous voltage even if the switching signal is turned off. Thus, each spindle can be simultaneously rotated by different S command. Also, each spindle is provided with the signals (*SSTP1 - *SSTP3) for stopping the spindle so that unneeded spindles can be stopped.

3) Cautions

a) The constant surface speed control becomes effective for the selected spindle only.

b) The signal during spindle orientation, spindle override signal, spindle stop signal (*SSTP), and polarity signals (SSIN and SGN) are effective for the selected spindle only.

c) When only the 1st axis is selected, the control signals (SIND and R1i - R12I) are valid.

d) The thread cutting, synchronous feed, and spindle fluctuation detection functions are used to calculate and process the spindle speed rate by the pulse from the position coder selected by the PMC signal (PC2SLCT).

Note) The analog interface printed-circuit board is required.

10.10 Serial Spindle

The speed of the serial interface spindle is controlled.

The spindle speed is specified with a 5-digit numeric value following address S. A speed command is output to the spindle motor according to the specified spindle speed. During constant surface speed control, a speed command is output so that it matches the spindle speed reached after constant surface speed control.

10.11 Cs Contour Control

The serial interface spindle permits positioning and linear interpolation with another servo axis. Thus, linear interpolation between the spindle and a servo axis can be specified.

(1) Control mode

The serial interface spindle has two modes.

- The spindle rotation control mode controls the speed of the spindle. (The spindle is rotated according to a speed command.)
- The spindle contour control mode (also called Cs contour control) controls the position of the spindle. (The spindle is rotated according to a move command.)

These modes are switched by a signal sent from the PMC.

(2) Spindle contour control axis

The axis subject to spindle contour control is placed as one of the CNC control axes. Any of the control axes can be selected as the spindle contour control axis.

With the FS0-TTC, two spindles can be controlled. If spindle contour control is performed for the two spindles, one spindle contour control axis must be placed for each tool post. Two spindle contour control axes cannot be placed for one tool post.

(3) Move command

In manual and automatic operation, a move command for the spindle contour control axis is programmed in the same way as for a servo axis.

Example)

Let the name of the spindle contour control axis be C.

G00 C30.0; . . . . (Positioning)

G01 X100.0 Y100.0 C90.0 F1000.0;

. . . . (Linear interpolation)

(4) Automatic loop gain setting when switching between spindle rotation control and spindle contour control is made

(a) Switching from spindle rotation control to spindle contour control

If there is a difference in servo loop gain between the axis subject to spindle contour

control and the other servo axes, linear interpolation with the spindle contour control axis cannot be performed properly.
As soon as spindle rotation control is switched to spindle contour control, an appropriate spindle contour control servo loop gain for a selected gear is automatically set for a necessary servo axis. The axis for which the servo loop gain is to be changed, and the spindle contour control servo loop gain for this axis must be set in parameters for each gear beforehand.

(b) Switching from spindle contour control to spindle rotation control
As soon as spindle contour control is switched to spindle rotation control, the original servo loop gain is set for the servo axis automatically.

10.12 Spindle Synchronization Control

In machine tools having two spindles (such as a lathe), the speeds of the two spindles sometimes have to match. This requires when a workpiece held on the first spindle is transferred to the second spindle while the spindles are rotating, and when acceleration/deceleration is performed while a workpiece is being held by the first and second spindles.
When a workpiece having a different figure is transferred between the spindles, the rotation phases (rotation angles) of the spindles must also match.
The serial interface spindle synchronization control function is provided to provide synchronization control for two spindles.

11. TOOL FUNCTIONS

11.1 T code output

Selection of tools can be done by commanding tool numbers with a 2-digit or 4-digit numeral after address T.

The offset number is specified with the last one digit of the T code.

11.2 Tool Life Management

Tools are classified into groups, and tool life (hours and times of use) is set for each group.When use of the tool exceeds the preset hours or times of use, another tool in the same group which has not yet exceeded the preset life time is selected.If all the tool in a group exceeds the preset life time, a signal is output to inform the operator that the tools must be changed to new tools.

With setting the cutter radius compensation number and the tool length compensation number of the tools, compensation corresponding to each tool can also be done.With use of this function Factory Automatic (FA) comes to a reach.

This function has the following features:

(1) Tool life can be set in hours or times of use.

(2) New tool select signal output
This signal is output when a new tool is selected in a group.This can also be used for automatic measurement in compensations of the new tools.

(3) Tool change signal
When all the tools of a group has exceeded their life time, this signal is output to inform the operator of tool changing required.

(4) Tool skip signal
By inputting this signal, tools still not exceeding their life time, can also be changed.

(5) Tool life management data is display/ modification
All tool life management data is displayed on the CRT screen, informing the operator of the condition of the tools at a single view.If necessary, the data can be modified via the MDI panel.
Number of groups and number of tools per group is selected by parameter from the following.
The part program storage length becomes shorter by approximately 6.3m.

Number of groups	Number of tools
16	16
32	8
64	4
128	2

11.3 Simplified Tool Life Management

This system counts the number of parts by couting M code (M30 or M02) at the program end.

When the number of parts reaches a preset value (tool life), the system judges the tool to have reached its life. The system counts up the life count, and clears the parts counter. It compensates for the T code which has been programmed according to the life count and a parameter.

```
TOOL LIFE DATA                          0123N5678

  NO.                                       DATA
  3910   TOOL LIFE                           300
  3911   PARTS COUNT (LIFE)                  283
  3912   LIFE COUNT                            5
  3913   PARTS COUNT (TOTAL)                1783

ACTUAL POSITION (RELATIVE)
    U   112.000          W   198.000
NO. 3910  =
                         AUTO
```

(a) No. 3910 TOOL LIFE : No. of tool life parts
When the number of parts of No. 3911 reaches this value, the tool is judged to have reached the first tool life.

(b) No.3911 PARTS COUNT (LIFE) : parts number counter
The counter is added by +1, each time M02 or M30 is executed and cleared to 0, each time the counter reaches the tool life once.

(c) No.3912 LIFE COUNT : Life count frequencies
This counter is added by +1, each time the tool reaches its life, and it shows how many times the tool has reached its life.

(d) No.3913 PARTS COUNT (TOTAL) : Total parts count
This value is the same as the number of parts displayed on the POSITION screen, and the following equation holds normally. (Parts count (total)) = (Tool life) × (Life count) + (Parts count)

Compensation for programmed T code

The programmed T code T○○□□ consists of tool selection No. ○○ and tool offset No.□□, which are compensated separately and executed as T●●■■.

The compensation value to tool selection numbers and the compensation value to offset numbers are determined by parameters.

(1) Before reaching the first life (when life count = 0)
Tool offset No. ■■ = □□
Tool selection No. ●● = ○○

(2) After the first life (When life count = 1)
Tool offset No. ■■ = □□ + (Offset No. compensation value)
Tool selection No. ●● = ○○ + (Tool selection No. compensation value)

(3) After the second life (When life count = 2)
Tool offset No. ■■ = □□ + (Offset No. compensation value) × 2
Tool selection No. ●● = ○○ + (Tool selection No. compensation value) × 2

(4) After the n-th life (When life count = n)
Tool offset No. ■■ = □□ + (Offset No. compensation value) × n
Tool selection No. ●● = ○○ + (Tool selection No. compensation value) × n

(Note 1) No compensation is made for the tool offset number and tool selection number, if the programmed value is 0.

(Note 2) If the tool offset number ■■ or tool selection number ●● after compensation exceeds the maximum value, ■■ or ●●becomes a remainder value of the maximum value.

(Note 3) If the tool offset number ■■ after compensation exceeds the number of offset groups, PS alarm No. 030 sounds. Accodingly, the maximum value of the tool offset number must be set to be less than the number of offset pairs.

(Note 4) T ●●■■after compensation is displayed on the T code indicator at all times.

11.4 Automatic Tool Length Measurement (G37)

By Commanding :

G37Z_;

The tool starts moving to the measurement position, and keeps on moving till the measuring position reach signal from the measurement device is output.

Moving of the tool is stopped when the tool head reaches the mesurement position.

Difference between coordinate values when tool has reached the measurement position and coordinate value commanded by G37 is added to the tool length compensation amount currently used.

12. MISCELLANEOUS FUNCTIONS

12.1 Miscellaneous Function (M2 digit)

When a 2 or 3-digit number after address M is commanded, BCD 2/3-digit code signal and M code read signal are output. This code is cleared by M,S and T function complete signal. This function is used for on/off of various functions at the machine side. A single M code can be commanded in one block.
M98 (sub program call) and M99 (return from sub program) are always processed in the CNC so, signal will not be output.

12.2 Second Miscellaneous Function (B-Function)

(1) T series

(a) Command range: 0 to ± 99999999

(b) Command method

(i) The decimal point can be used for input. (when parameter No. 16 NPRD: 0)

(Input)	(Output)
B10.	10000
B10.	10

(ii) It is possible to change over the scale factor of B output, x 1000 or x 1 when the decimal point input is omitted, using the parameter.

	(Input)	(Output)
For x 1000	B1	1000
For x 1	B1	1

(iii) It is possible to change over the scale factor of B output x 1000 or x 10000 when the decimal point input is omitted in the inch input system, usint the parameter.

(2) M series

Indexing of the table is performed by address B and a following 3-digit or 6-digit number. The relationship between B codes and the corresponding indexing differs to the machine tool builders.
You can select either B3-digit or B6-digit by parameter setting.

13. PROGRAM CONFIGURATION

13.1 Program Number

A program number is given to each program to distinguish a program from other programs. The program number is given at the head of each program, with a 4-digit number after the address O.

Program number of the program currently under execution is always displayed on the CRT screen. Program search of programs registered in the memory is done with the program number. The program number can be used in various other ways.

13.2 Main Program

A program is divided into the main program and the sub program. The CNC normally operates according to the main program, but when a command calling a sub program is encountered in the main program, control is passed to the sub program. When a command indicating to return to the main program is encountered in the sub program, control is returned to the main program.

13.3 Sub Program

When there are fixed sequences or frequently repeated patterns in a program, programming can be simplified by entering these pattern as sub programs to the memory. Sub program is called by M98, and M99 commands return from the sub program. The sub program can be nested 2 folds.

Format

Sub program call

M98P○○○ L○○○○;

P○○○: Program number of sub program to be called

L○○○○: How many times to repeat the sub program

Return from sub program

Format

M99;

13.4 Sequence Number

Sequence number can be given in a 4-digit number after the address N at the head of the program block.

The sequence number of the program under execution is always displayed on the CRT screen. The sequence number can also be searched in the program by the sequence number search function.

13.5 Tape Codes

Either the EIA or the ISO code can be used as tape code. The input program code is distinguished with the first end of block code (EIA: CR, ISO: LF).

See the List of Tape Codes for tape codes used.

13.6 Basic Addresses and Command Value Range

Function	Address	Input in mm Output in mm	Input in inch Output in mm	Input in mm Output in inch	Input inch Output in inch
Program number	:(ISO) O (EIA)	1 –9999	1 –9999	1 –9999	1 –9999
Sequence number	N	1 –9999	1 –9999	1 –9999	1 –9999
Preparatory function	G	0 –255	0 - 99	0 - 99	0 - 99
Dimension word	X, Y, ZA, B, CI, J, KQ, R	±99999.999mm (±9999.9999mm)	±9999.9999inch (±393.70068inch)	±99999.999mm (±9999.9999mm)	±9999.9999inch (±393.70068inch)
Feed per minute	F	1 –100000 mm/min (1 –12000 mm /mm)	0.01–4000.00 inch /min (0.01–480 inch /min)	1 –100000 mm/min (1 –12000 mm /min)	0.01–4000.00 inch/min (0.01–480 inch /min)
Feed per revolution	F	0.0001–500.0000mm/rev	0.000001–9.999999 inch /rev	0.0001 –500.0000 mm/rev	0.000001–9.999999 inch/rev
Spindle speed function	S	0 –19999	0 –19999	0 –19999	0 –19999
Tool functions	T	0 –9999	0 –9999	0 –9999	0 –9999
Miscellaneous functions	M	0 –999	0 –999	0 –999	0 –999
Dwell	X, P	0 –99999.999 sec. (0 –9999.9999 sec.)	0 –99999.999 sec. (0 –9999.9999 sec.)	0 –99999.999 sec. (0 –9999.9999 sec.)	0 –99999.999 sec. (0 –9999.9999 sec.)
Designation of sequence number, number of repetitions	P	1 –99999999sec.	1 –99999999sec.	1 –99999999sec.	1 –99999999sec.
Specification of sequence number	P,Q	1 –9999	1 –9999	1 –9999	1 –9999
Specification of macro arithmetic operation and control commands	H	0 –99	0 –99	0 –99	0 –99
Offset number	H	0 –64	0 –64	0 –64	0 –64
2nd miscellaneous function	B	0 –999999	0 –999999	0 –999999	0 –999999

(Note 1) In case of increment system 1/10, command value range is shown in parenthesis.

13.7 Tape Format

(1) T series

Metric input

N04·G03· { XL + 053 / UD053 }·{ XL + 053 / UD053 }.

{ RD053 / CD053 / ID053·KD053 }.{ F034 / F060 }.{ S02 / S04 / S05 }.{ T02 / T04 }.

M03;

Inch input

N04·G03· { XL + 044 / UD044 }·{ XL + 044 / UD044 }.

{ RD044 / CD044 / ID044·KD044 }.{ F016 / F042 }.{ S02 / S04 / S05 }.{ T02 / T04 }.

M03;

(2) M series

Metric input

N04.G03.XL + 053.YL + 053.ZL + 053

{ RD053 / ID053.JD053.KD053 }.{ F034 / F060 }.

H02. { S02 / S04 / S05 }{ T02 / T04 }{ B03 / B06 } .M03:

Inch input

N04.G03.XL + 044.YL + 044.ZL + 044.

{ RD044 / ID044.JD044.KD044 }.{ F016 / F042 }.

H02. { S02 / S04 / S05 }{ T02 / T04 }{ B03 / B06 } .M03:

In increment system 1/10

Metric input

N04·G03· { XL + 044 / UD044 }·{ XL + 044 / UD044 }.

{ RD044 / CD044 / ID044·KD044 }.{ F034 / F050 }.{ S02 / S04 / S05 }.{ T02 / T04 }.

M03;

Inch input

N04·G03· { XL + 035 / UD035 }·{ XL + 035 / UD035 }.

{ RD035 / CD035 / ID035·KD035 }.{ F016 / F032 }.{ S02 / S04 / S05 }.{ T02 / T04 }.

M03;

In increment system 1/10

Metric input

N04.G03.XL + 044.YL + 044.ZL + 044.

{ RD044 / ID044.JD044.KD044 }.{ F034 / F050 }.

H02. { S02 / S04 / S05 }{ T02 / T04 }{ B03 / B06 } .M03:

Inch input

N04.G03.XL + 035.YL + 035.ZL + 035.

{ RD035 / ID035.JD035.KD035 }.{ F016 / F032 }.

H02. { S02 / S04 / S05 }{ T02 / T04 }{ B03 / B06 } .M03:

(Note 1) The above addresses and numerical values have the following meanings.

(Note 2) The above format omits addresses H, P, A, and Q, shince they have various meanings.

13.8 Label Skip

Label skip function becomes valid in the following cases, and "LSK" is displayed on the CRT screen.

(1) When power is put on.

(2) When the CNC is reset.

When label skip function is valid, all codes to the first end of block (EOB) code encountered are ignored.

The ignored part is called "Leader part", and section after the first end of block (EOB) code, "significant information".

13.9 Control-in/Control-out

	ISO code	EIA code
Control out	)	Channel 2-4-5 on
Control in	(	Channel 2-4-7 on

Information between the control-in and the control-out are regarded as notes and are ignored.

The reset codes (ISO code: %, EIA code: ER) cannot be used in this part.

The ignored part is called "Notes".

13.10 Optional Block Skip

```
  ⋮
N100 X__Y__;
/N101 X__Y__;  } Ignored when optional
/N102 X__Y__;  } block skip on.
N103 X__Y__;
  ⋮
```

When a slash (/) is programmed at the head of a program, and when the machine is operated with the optional block skip switch on the machine operator's panel on, information in the block commanded is ignored.

If the optional block skip switch is turned off, information in the / commanded block will not be ignored. The block with / commanded can be skipped by the operator's selection.

Also, an optional skip can be extended up to nine with an optional function.

/1 (or /)	.....	optional skip 1
/2	.....	optional skip 2
:		:
/9	.....	optional skip 9

13.11 Program Name

```
O1234 (ABCD·····)
        |
        PROGRAM NAME (MAX. 31 CHARACTERS)
```

To discriminate from other programs when the registered program directory is displayed, the program name can be assigned subsequent to Program No.
Insert the program name between Control Out "(" and Control In ")" just after Program No.
For the program name, any code can be used if it is a code which can be used in CNC.
This program name is displayed together with Program No. displayed on the registered program directory. However, only 31 characters from the head are displayed for the program name.

13.12 F10/11 Tape Format

TAPE FOR F10/11 ⟶ 0·C

F10/11 tape can be used within the range of F0 function.

13.13 Directory Display of the Floppy Cassette

```
DIRECTORY (FLOPPY)        O001 N001
 NO.  FILE NAME          (METER )VOL.
0001.  PARAMETER            0. 5
0002.  OFFSET              40. 7
0003.  ALL ·PROGRAM        32. 2
0004.  O0002                7. 5
0005.  O0003               10. 2
0006.  O0004                2. 6
0007.  O0005                4. 9
```

SCREEN

This function displays the directory of floppy cassette, permitting file input and output.
The directory is displayed and the file is input and output in the foreground and background editing modes.

14. FUNCTIONS TO SIMPLIFY PROGRAMMING

14.1 Canned Cycles (G73, G74, G76, G80 - G89, G98, G99) (M series)

G code	Operation		Function
G73	G98 mode 	G99 mode 	Peck drilling cycle (Note)
G76	 	 	Fine boring cycle
G81	 	 	Drilling cycle (spot boring)
G82	 	 	Drilling cycle (counter boring)

G code	Operation		Function
G83	R, I, Q, d, Q, Q	R, I, Q, d, Q, Q	Peck drilling cycle (Note)
G84 G74	I, R, Spindle CW, Z, Spindle CCW, Ⓟ	I, R, Spindle CW, Z, Spindle CCW, Ⓟ	Tapping cycle (G74 is CCW tapping cycle)
G85	I, R, Z	I, R, Z	Boring cycle
G86	I, Spindle CW, R, Spindle CCW, Z	I, Spindle CW, R, Spindle CCW, Z	Boring cycle
G87	Canned cycle II (Note) oss, I, oss, Z, Spindle CW, R, Q	G99 mode cannot be used in canned cycle G87 (Canned cycle II).	Back Boring cycle

G code	Operation		Function
G88	I, R, Z, ⓟ, Spindle CW, Spindle CCW	I, R, Z, ⓟ, Spindle CW, Spindle CCW	Boring cycle
G89	I, R, Z, ⓟ	R, Z, ⓟ	Boring cycle

Cutting feed		OSS	Oriented spindle stop (Spindle stops at constant rotation position)
Rapid traverse			
Manual feed			Shift
ⓟ	Dwell	I	Initial point
Z	Z point	R	R point

(Note) "d" of G73 and G83 is set by parameters.

When the drilling axis is Z axis, machining data in the canned cycle is commanded as follows:

G00X___Y___Z___R___Q___P___F___;

Drilling mode G ..... See previous table.

Drilling position data

X, Y ..... Command position of the hole.

Z ..... Specify hole end position shown in the previous table.

R ..... Specify R point position shown in the previous table.

Q ..... Specify cutting quantity with G73, G83, and shift quantity with G76, G87.

P ..... Specify dwell time at the hole bottom.

F ..... Specify feed rate for cutting.

(1) R point level return (G99)

By specifying G99, return point in canned cycle is specified to R point.

The drilling starts from the end point of the previous block. If the previous block has ended in the initial point, it begins from the initial point and returns to the R point.

(Example) When G81 was commanded under G99 mode

(2) Initial level return (G98)

By specifying G98, return point in canned cycle is specified to the initial level. The drilling starts from the end point of the previous block. If the previous block has ended in the R point, it begins from the R point and returns to the initial point.

(Example) When G81 was commanded under G98 mode

14.2 Canned Cycles

14.2.1 Cutting cycle A (G90)

When using the incremental command, the sign of a numeric value following address U or W is determined by the directions of paths 1 and 2. If the direction of path 1 meets the negative direction of the X axis , the U value is negative.

In the above figure, if the direction of the path 1 is negative in X axis, the sign of the number following address I is negative.

If the sign is reversed, the reversed taper is cut.

14.2.2 Thread cutting cycle (G92)

(Note) When chamfering signal CDZ is "1", chamfering amount (r) is not available.
When chamfering signal CDZ is "0", chamfering amount (r) is available.

14.2.3 Cutting cycle B (G94)

In the above figure, if the direction of the path 1 is negative in Z axis, the sign of the number following address K is negative.

14.3 Multiple Repetitive Canned Cycle (G70-G76)

14.3.1 Rough cutting cycle of outer diameter (G71)

G71 U(Δd) R(e) :
G71 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;

Δd	:	Cutting depth
e	:	Return relief
ns	:	Sequence number of the first block of the program which specifies the finish figure
nf	:	Sequence number of the last block of the program which specifies the finish figure
Δu	:	Distance and dirtection of the finish allowance in the X-axis direction (diameter/radius specification)
Δw	:	Distance and dirtection of the finish allowance in the Z-axis direction
f,s,t	:	F, S, T code

When a finish figure of A → A' → B is programmed as illustrated, the machine tool cuts off the area commanded by cutting depth Δd, while leaving a finish allowance of Δu/2, Δw uncut.

A monotonous incremental or decremental pattern should be provided between A' and B in both X axis and Z axis directions.

14.3.2 Rough facing cycle (G72)

G72 U(Δd) R(e) :
G72 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;

Meanings of Δd, e, ns, nf, Δu, Δw, f, s, t are equal to those in G71.

The machine tool functions in the same way as in G71, but cutting is executed by the motion parallel to the X axis as shown in the following figure.

A monotonous incremental or decremental pattern should be provided between A' and B in both X axis and Z axis directions.

14.3.3 Pattern repeating (G73)

G73 U(Δi) W(Δk) R(d) :
G73 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;

Δi	:	Distance and direction of return relief in the X direction... Radius specification
Δk	:	Distance and direction of return relief in the Z direction
d	:	Number of division times (equal to the number of rough cutting times)
ns	:	Sequence number of the first block of the program which specifies the finish figure
nf	:	Sequence number of the last block of the program which specifies the finish figure
Δu	:	Finish allowance in the X axis direction (diameter/radius speciction)
Δw	:	Finish allownance in the Z axis direction
f, s, t	:	F, S, T codes

A constant cutting pattern can be repeatedly operated, while relocating its position bit by bit. By using this cycle, a workpiece can be cut efficiently after it has been roughly profiled in the previous machining process, such as forging, casting, etc.

14.3.4 Finish cycle (G70)

G70 P(ns) Q(nf) ;
ns : Sequence number of the first block of the program specifies the finish figure.
nf : Sequence number of the last block of the program specifies the finish figure.

Finishing can be done by the above command after rough cutting according to G71, G72 and G73.

14.3.5 Deep hole drill cycle (G74)

G74 R(e) ;
G74 X(U)__ Z(W)__ P(Δi) Q(Δk) R(Δd) F(f) ;

e	:	Return relief ;
X	:	X component at point B
U	:	A → B incremental quantity
Z	:	Z component at point C
W	:	A → C incremental quantity
Δi	:	Move amoutn in the X axis direction (unsigned specification)
Δk	:	Cutting depth in the Z axis direction (unsigned specification)
Δd	:	Return relief of tool in cutting bottom
f	:	Feed rate

The motion as illustrated above is executed by the above program command. By this cycle, chip processing is possible in outer figure cutting. If X(U) and P are omitted, the Z axis only moves in the deep hole drill cycle.

14.3.6 Outer diameter grooving cycle (G75)

G75 R(e) ;

G75 X(U)__ Z(W)__ P(Δi) Q(Δk) R(Δd) F(f) ;

The motion as illustrated above is executed by the above program command. This cycle corresponds to G74 in which X and Z are replaced with each other. Chip processing is possible durig facing in this cycle.

Also, the grooving and cutting off can be done in outer diameter cutting (by omitting Z, W and Q)

14.3.7 Threading cycle (G76)

G76 P (m) (r) (a) Q (Δd min) R (d) ;
G76 X (U)__ Z (W)__ R (i) P (k) Q (Δd) F (ℓ) ;

m : Number of repetition times of final finish1～99
This modal specification remains effective until the next specification is made. This value can also be specified by parameter setting.

r : Chamfering quantity of thread

a : Tool nose angle (Thread angle)
Six kinds of angles (80°, 60°, 55°, 30°, 29°, 0°) are selectable.
This angle is specified by a 2-digit numeric value.

m, r, a are specified at address P at a time.

m, r, a are specified at address P at a time.
P02/m 12/r 60/a, if m=2, r=1.2ℓ , and a=60°, for example.

Δbmin : Minimum cutting depth

d : Finish allowance

i : Radius difference in threaded part
(Threading is done straight, if i=0)

k : Height of thread (The distance in the X axis direction is commanded by a radius value)

Δd : First cutting depth (specified by radius value)

ℓ : Lead of thread (same as in threading by G32)

If PMC is provided, whether the chamfering is executed or not is switchable by the external signal.

14.4 Chamfering and Corner R

Item	Command	Tool movement
Chamfering Z→X	G01Z (W) bC (I) ±i ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	
Chamfering X→Z	G01X (U) bC (k) ±k ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	
Corner R Z→X	G01Z (W) bR ±r ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	
Corner R X→Z	G01X (U) bR ±r ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

A chamfer or corner arc can be inserted between two blocks which intersect at a right angle. Amount of chamfering or corner arc is specified by address C or R.

14.5 Optional Angle Corner Rounding

By adding :

, C_

to the end of blocks commanding linear or circular interpolation, optional angle corner rounding can be automatically inserted.

Example)

① G91 G01 X100.0, C10.0 ;

② X100.0 Y100.0 ;

14.6 Optional Angle Chamfering

By adding :

, R_

to the end of blocks commanding linear or circular interpolation, optional angle corner rounding can be automatically inserted.

Specify a numeral following address C, which indicates the distance between the imaginary corner and start or end of chamfering.

Example)

① G91 G01 X100.0, R10.0 ;

② X100.0 Y100.0 ;

14.7 Mirror Image for Double Turrets (G68, G69)

G code	Meaning
G68	Double turret mirror image on
G69	Mirror image cancel

Double turret programming

X40 Z180 T0101 ;	Position turret A at ①.
G68 ;	Shift the coordinate system by the distance A to B (120mm), and turn mirror image on.
X80 Z120 T0202 ;	Position turret B at ②.
G69 ;	Shift the coordinate system by the distance B to A, and turn mirror image on.
X120 Z60 T0101 ;	Position turret A at ③.

Mirror image can be applied to the X axis by a G code.

When G68 is designated, the coordinate system is shifted to the mating turret side, and the X-axis sign is reversed from the programmed command to perform symmetrical cutting.

To use this function, set the distance between the two turrets by parameter.

14.8 Direct Drawing Dimensions Programming

Angles of straight lines, chamfering values, corner rouding values, and other dimensional values on machining drawings can be programmed by directly inputting these values. In addition, the chamfering and corner rounding can be inserted between straight lines having an optional angle.

Command table(1/2)

	Commands	Movement of tool
1	X_2 _ (Z_2 _) A _ ;	
2	A_1 _ ; X_3 _ Z_3 _ A_2 _ ;	
3	X_2 _ Z_2 _ R_1 _ ; X_3 _ Z_3 _ ; or A_1 _ R_1 _ ; X_3 _ Z_3 _ A_2 _ ;	
4	X_2 _ Z_2 _ C_1 _ ; X_3 _ Z_3 _ ; or A_1 _ C_1 _ ; X_3 _ Z_3 _ A_2 _ ;	

Command table (2/2)

	Commands	Movement of tool
5	X_2 _ Z_2 _ R_1 _ ; X_3 _ Z_3 _ R_2 _ ; X_4 _ Z_4 _ ; or A_1 _ R_1 _ ; X_3 _ Z_3 _ A_2 _ R_2 _ ; X_4 _ Z_4 _ ;	
6	X_2 _ Z_2 _ C_1 _ ; X_3 _ Z_3 _ C_2 _ ; X_4 _ Z_4 _ ; or A_1 _ C_1 _ ; X_3 _ Z_3 _ A_2 _ C_2 _ ; X_4 _ Z_4 _ ;	
7	X_2 _ Z_2 _ R_1 _ ; X_3 _ Z_3 _ C_2 _ ; X_4 _ Z_4 _ ; or A_1 _ R_1 _ ; X_3 _ Z_3 _ A_2 _ C_2 _ ; X_4 _ Z_4 _ ;	
8	X_2 _ Z_2 _ C_1 _ ; X_3 _ Z_3 _ R_2 _ ; X_4 _ Z_4 _ ; or A_1 _ C_1 _ ; X_3 _ Z_3 _ A_2 _ R_2 _ ; X_4 _ Z_4 _ ;	

14.9 External Motion Function (G80, G81)

An external motion signal (EF) is output to the machine tool after positioning the X and Y axes by G81 X___Y___; command. When the motion end signal (FIN) returns from the machine tool, CNC starts the next positioning motion.

Whether the canned cycle (drill cycle) accompanied with the Z axis motion is done or the EF signal is output by this function in G81 mode after positioning the X, Y axes is determined by parameter setting.

G80: Cancel of G81 made.

14.10 Conversational programming with Graphic Function

(T series)

When G01 is selected

(M series)

When G01 is selected

As the help of programming in EDIT mode, menu and example drawing is displayed on CRT screen. You can program easly to have conversation with the screen.

14.11 Menu Programming

The assist programming in EDIT mode, the following menu is indicated on the CRT.

(1) When G is keyed in, the G code menu is indicated on the lower half of the CRT screen.

(2) When the G code selected from the menu in (1) is input, the standard format of the one block corresponding to the G code is indicated.

For example, when G is keyed in after inserting N102, the G code menu as shown below is indicated on the CRT.

The G code menus after G04 can be sequentially indicated by pushing the ↓ or ↑ button.

For example, when selecting G01, key in 0 and 1, and then push INSRT button.

At this time, G01 is inserted to the memory as shown below, and the standard format of the G01 block is indicated on the CRT screen.

Insertion of one block is completed by keying in as follows:

X_**INSRT**, Y_**INSRT**, F _**INSRT**, T _ **EOB**

14.12 Scaling (G50, G51)

Scaling can be commanded to figures commanded in the machining programs.

G51 I_J_K_P_;

where

I, J, K: Center of scaling

P: Magnification

By this command, scaling of the magnification specified by P is commanded with the point commanded by I, J, K as its center. G50 cancels scaling mode.

G50: Scaling mode cancel

G51: Scaling mode

Commandable magnification is as follows:

0.00001 - 9.9999 times

or 0.001 - 999.999 times

If P was not commanded, the magnification set by the CRT/MDI is applied.

When I, J, K are omitted, the point where G51 was commanded becomes the center of scaling.

Scaling cannot be done to offset amounts such as tool length compensation, cutter radius compensation, or tool offset.

Parameter can be set to each axis to command scaling.

14.13 Coordinate System Rotation (G68, G69)

Patterns commanded by the program can be rotated. For examples, by using this function, when the attached workpiece comes in a position which is somewhat rotated from the machine coordinates, a rotation command can be performed to compensate the position.

If there is a pattern which is a rotation of another pattern, the program in which the pattern was programmed can be assigned as a sub program. Programming time can be greatly reduced just by calling the sub program and rotating the pattern.

Command format

$$\left\{\begin{matrix} G17 \\ G18 \\ G19 \end{matrix}\right\} G68\alpha_\ \beta_\ R_;$$

α, β: Specify two axes from X, Y, Z axes which correspond to G17, G18, G19. (G90/G91 modes are recommended)

R: Rotation (+ for the counter clockwise direction. Commanded in absolute value.)

By this command, commands thereafter are rotated in the angle commanded by R, with the point commanded by α, β as the rotation center. Rotation angle is commanded in 0.001 deg units in a range of:

$-360.000 \leq R \leq 360.000$

The rotation plane is the plane selected (G17, G18, G19) when G68 was commanded. G17, G18 and G19 may not be commanded in the same block as G68.

When α, β is omitted, the point where G68 was commanded becomes the rotation center.

G69;

Cancels the coordinate system rotation.

14.14 Canned Cycle for Drilling

G code	Drilling Axis	Hole machining operation	Operation in the hole bottom position	Retraction operation	Applications
G80					Cancel
G83	Z	Cutting feed / Intermittent	Dwell	Rapid traverse	Drill / Peck drilling
G84	Z	Cutting feed	Spindle CW	Cutting feed	Tapping
G85	Z	Cutting feed	Dwell	Cutting feed	Boring
G87	X	Cutting feed / Intermittent	Dwell	Rapid traverse	Drill / Peck drilling
G88	X	Cutting feed	Spindle CCW	Cutting feed	Tapping
G89	X	Cutting feed	Dwell	Cutting feed	Boring

The canned cycle for drilling simplifies the program normally by directing the machining operation commanded with a few blocks, using one block including G-function.

(Note 1) Spindle is rotated counterclockwise by M04 command.

(Note 2) Selection of cutting feed or intermittent feed in G83 / G87 hole machining operation depends on the command of address Q.

14.15 Rigid Tap

In tapping, the feed amount of Z axis for one rotation of spindle should be equal to the pitch of screw of tapper. Namely, the following conditions must be satisfied in the best tapping:

P = F/S, where

P : Pitch of screw of tapper (mm)

F : Feed rate of Z axis (mm/min)

S : Spindle speed (rpm)

The rotation of spindle and feed of Z axis are independently controlled in the tapping cycle (G84) and left-handed tapping cycle (G74). Therefore, the above conditions may not always be satisfied. Especially at the hole bottom, both the rotation of spindle and feed of Z axis reduce the speed and stop. After that, they move in the inverse direction while increasing the speed. However, the above conditions may not be satisfied in general since each acceleration / deceleration is performed independently. Therefore, in general, the feed is compensated by mounting a spring to the inside of holder of tapper to improve the accuracy of tap cutting.

The rotation of spindle and feed of Z axis are controlled so that they are always synchronous each other in the rigid tapping cycle. Namely, in other than rigid tapping, control for speed only is performed. In the rigid tapping, however, position control is also performed during the rotation of spindle, that is, the rotation of spindle and feed of Z axis are controlled as linear interpolation of two axes.

This allows the following condition to be satisfied also during acceleration / deceleration at the hole bottom and a tapping of improved accuracy to be made.

P = F/S

Command format

M29Sxxxx;Rigid tap mode

G84 X– Y– Z– R– P– F– S– K–

Drilling position

Coordinate value of point Z

Coordinate value of point R

Dwell time on returning to points Z and R (however, when P is valid with parameter selection)

Cutting feed rate

Spindle speed rpm

Number of repetition

M29SO;Rigid tap reset

When the rigid tap is commanded in the feed per revolution mode (G95), the unit of cutting feed rate F__ becomes mm/rev or inch/rev. Therefore, the pitch of screw tap can be directly specified.

Fig. 14.15 The control system of spindle during rigid tapping

Gear ratio of spindle to position coder (1:p)	Least command increment (detection unit) deg
1:1	0.088(1×360/4096)
1:2	0.176(2×360/4096)
1:4	0.352(4×360/4096)

Even use of the spindle motor incorporating the position coder enables rigid tapping. In this case, the gear ratio of the spindle motor and the spindle is set by the parameter. If, however, there are multiple gears between the spindle motor and the spindle, i.e. , if three speed gears (for high, midium, and low speeds) are mounted between them, only one of these gears enables rigid tapping.

In addition, use of the spindle motor incorporating the position coder enables rigid tapping but disables threading and per revolution dwell.

14.16 Grinding Canned Cycle

14.16.1 Griding canned cycle (GCC)

1) Traverse indirect gauge grinding cycle (G71)

A : The first cutting depth
B : The second cutting depth
W : Grinding range
U : Dwell time Maximum command time 9999.999 sec.
I : Feed rate of A and B
K : Feed rate of W
H : Repetition frequency
Setting value 1-9999

2) Traverse direct gauge grinding cycle (G72)

P : Gauge number (1-4)

If the option of the multi-step skip is employed, gauge number can be specified. The specifying means of the gauge number is the same as the multi-step skip. If the option of the multi-step skip is not employed, a conventional skip signal becomes effective.

Other commands are similar to G71.

3) Oscillation indirect gauge grinding cycle (G73)

A : Cutting depth
B : Cutting depth
W : Grinding range
U : Dwell time
K : Feed rate
H : Repetition frequency
Setting value 1-9999

4) Oscillation direct gauge grinding cycle (G74)

P : Gauge number (1-4)
If the option of the multi-step skip is employed, gauge number can be specified. The specifying means of the gauge number is the same as the multi-step skip. If the option of the multi-step skip is not employed, a convertional skip signal becomes effective.
Other commands are similar to G73.

14.16.2 Grinding canned cycle (GSC)

In the grinding canned cycle, repeated cutting peculiar to grinding machining normally commanded by a number of blocks, is simply programmed by commanding one block which includes the G function.
There are the following 4 types of grinding canned cycle.

1) Plunge grinding cycle G75
2) Plunge direct grinding cycle G77
3) Continuous feed plane grinding cycle G78
4) Intermittent feed plane grinding cycle G79

The following shows the general format of the program command of the grinding canned cycle.

G□□I−J−K−X (Z)−R−F−P−L−;

Address	Specification contents	Command units and command range
G □□	Canned cycle mode	One of G75, G77, G78, or G79
I, J	Grindstone cutting depth	Command unit : Same as minimum setting unit Command range : ±8 digits (0 to ±99999999)
K	Total cutting depth	
X (Z)	Grinding range	
R, F	Feed rate	Command unit : 0.001mm/min 0.0001 inch/min Command range : ① When standard unit system 0.001 to 100000 mm/min 0.0001 to 4000 inch/min ② When 1/10 setting unit 0.001 to 12000 mm/min 0.0001 to 480 inch/min
P	Dwell time	Command unit : 0.001 sec Command range : 0 to 99999.999 sec
L	Grindstone wear compensation number	Command range : 1 to 200

(1) Plunge grinding cycle

The plunge grinding cycle is possible by the following command.

G 75 I−J−K−X (Z)−R−F−P−L−;

I : The first cutting depth (Cutting direction is by command coding.)

J : The second cutting depth (Cutting direction is by command coding.)

K : Total cutting depth

X (Z): Grinding range (Grinding direction is by command coding.)

R : Feed rate of I and J

F : Feed rate of X (Z)

P : Dwell time

L : Grindstone wear compensation number (Note 1)

(Note 1) L is specified when performing continuous dressing.

(Note 2) X (Z), I, J and K commands are all incremental commands.

The plunge grinding cycle is made up from the following sequence of 6 operations.
The operations from ① up to ⑥ are repeated until the grindstone cutting amount reaches the total cutting amount specified by address K.

① Grindstone cutting:
Cuts in Y axis direction by cutting feed only the amount specified by the first cutting depth I. The feed rate becomes the rate specified by R.

② Dwell:
Performs dwell for only the time specified by P.

③ Grinding:
Shifts by cutting feed only the amount specified by X (or Z) in the X axis direction (or Z axis direction). The feed rate becomes the rate specified by F.

④ Grindstone cutting:
Cuts in Y axis direction by cutting feed only the amount specified by the second cutting depth J. The feed rate becomes the rate specified by R.

⑤ Dwell:
Performs dwell for only the time specified by P.

⑥ Grinding (return direction):
Sent at rate specified by F in the reverse direction only the amount specified by X (or Z).

In case of single block, the operations from ① to ⑥ are performed by one cycle start. When cutting by I or J, in the case where the total cutting depth is reached, the cycle finishes after the following sequence of operations (up to ⑥) has been executed. The cutting depth in this case reaches the total cutting depth position.

1) When total cutting depth is reached by the cutting operation of I or J.

2) When total cutting depth is reached in the middle of cutting of I or J.

(2) Plunge direct grinding cycle

The plunge direct grinding cycle is possible by the following command.
The command method is the same as the G75 case except for the G code. Further, even for the operation, the same sequence of 6 operations as the G75 case is repeated.
For G77, the cycle can be ended after discontinuing the current operation sequence (or after the current operation sequence has ended) by inputting a skip signal during the cycle.
The following shows the operation at skip signal input for each operation sequence.

1) Case of during operation sequence ① and ④ (when I and J shift)
   Cutting immediately stops and returns to X (Z) coordinate at cycle start.

2) Case of during operation sequence ② and ⑤ (during dwell)
   Dwell immediately stops and returns to X (Z) coordinate at cycle start.

3) Case of during operation sequence ③ and ⑥ (when I and J shift)
   After shift of X (Z) has finished, returns to X (Z) coordinate at cycle start.

(3) Continuous feed plane grinding cycle

The continuous feed plane grinding cycle is possible by the following command.

G 78 I−(J−)K−X −F−P−L−;

I : The first cutting depth (Cutting direction is by command coding.)
J : The second cutting depth (Cutting direction is by command coding.)
K : Total cutting depth
X : Grinding range (Grinding direction is by command coding.)
F : Feed rate
P : Dwell time
L : Grindstone wear compensation number (Note 1)

(Note 1) L is specified when performing continuous dressing.
(Note 2) X, I, J and K commands are all incremental commands.

The continuous feed plane grinding cycle is made up from the following sequence of 4 operations.
The operations from ① up to ④ are repeated until the grindstone cutting amount reaches the total cutting amount specified by address K.

① Dwell
② Grinding
③ Dwell
④ Grinding (return direction)

In case of single block, the operation from ① to ④ are performed by one cycle start.

(Note) When J is not commanded, it is regarded as J = 1.
Further, the J command effective only at the specified block. It does not remain as modal information. (Irrespective of "J" of G75, G77, and G79)

When cutting by I or J, in the case the total cutting depth is reached, the cycle finishes after the following sequence of operations (up to ④) has been executed. The cutting depth in this case reaches the total cutting depth position.

1) When total cutting depth is reached by cutting operation of I or J

2) When the total cutting amount is reached in the middle of cutting of I or J

(4) Intermittent feed plane grinding cycle

The intermitent feed plane grinding cycle is possible by the following command.

G 79 I−J−K−X−R−F−P−L−;

I : The first cutting depth (Cutting direction is by command coding.)
J : The second cutting depth (Cutting direction is by command coding.)
K : Total cutting depth
X : Grinding range (Grinding direction is by command coding.)
R : Feed rate of I and J
F : Feed rate of X
P : Dwell time
L : Grindstone wear compensation number (Note 1)

(Note 1) L is specified when performing continuous dressing.

(Note 2) I, J, K and X commands are all incremental commands.

The intermittent feed plane grinding cycle is made up from the following sequence of 6 operations.

The operations from ① up to ⑥ are repeated until the grindstone cutting amount reaches the total cutting amount specified by address K.

① Grindstone cutting:
Cuts in Z axis direction by cutting feed only the amount specified by the first cutting depth I. The feed rate becomes the rate specified by R.

② Dwell:
Performs dwell for only the time specified by P.

③ Grinding:
Shifts by cutting feed only the amount specified by X in the X axis direction. The feed rate becomes the rate specified by F.

④ Grindstone cutting:
Cuts in Z axis direction by cutting feed only the amount specified by the second cutting depth J. The feed rate becomes the rate specified by R.

⑤ Dwell:
Performs dwell for only the time specified by P.

⑥ Grinding(return direction):
Sent at rate specified by F in the reverse direction only the amount specified by X.

In the case of a single block, the operations from ① to ⑥ are performed by one cycle start.

14.17 Conversational Input Function with Figure for Cylindlical Grinding Machine (Order-made macro)

According to the contents of grinding to be programmed, select a menu as appropricate. Referring to a message indicated on the screen, input data in response to each inquiry. The machining programs will be prepared in sequence. The type of grinding includes the outer diameter/face plunge, outer diameter/face traverse and R contouring.

THE FIRST PICTURE ①

Select a required item according to the menu.

GRINDING CONDITION FILE Screen ②

Prepare the grinding conditions in advance on the file.

PROGRAMMING Screen ③

Prepare, modify and delete a program according to Parts No.

WORK FORM O D PLUNGE Screen ④

Input the profile data of the section to be ground. The grinding conditions are preset automatically according to the grinding condition file.

WHEEL DIAMOND DRESS REF Screen ⑤

Input the data of the grinding figure.

DRESS CONDITION, PROGRAM REF Screen ⑥

Input the condition data for dressing.

(Note) This function is performed by use of order-made macro, custom macro and graphic display.

14.18 Angular Axis Control (GCC, GSC)

The angular axis control function is a function that the amount of the movement of each axis is controlled according to the inclination degree if X axis is inclined to Z axis by an angle other than 90°. The program is made supposing that X axis and Z axis are orthogonalized. However, the actual movement is controlled according to the inclination degree.

14.19 Normal Direction Control (GSC)

Corresponding to the dressing figure changes, the dresser angle can be controlled to always face the normal direction to the figure (program command). By commanding G151 or G152, the dresser normal direction control axis (rotation axis) is controlled so that the dresser always faces the direction perpendicular to the advancing direction.
(In the following description, the dresser normal direction control axis is described for the A axis.)

(1) Command method

G150: Normal direction control cancel mode (normal direction control is not performed)

G151: Normal direction control left side on (controlled to face the direction perpendicular to left side for advancing direction.)

G152: Normal direction control left side on (controlled to face the direction perpendicular to right side for advancing direction.)

For the A axis angle, as shown inthe diagram on the right, looking from the A sixs rotation center, the +X direction is defined as 0 deg., and subsequently for counterclodkwise turns, the +Y direction as 180 deg., and the −Y direction as 270 deg.

(2) Operation

In the normal direction control, the tool is controlled so that it becomes perpendicular to the advancing direction of the X-Y plane. When moving from cancel mode to normal direction control mode, G151 or G152 start at the start point of the commanded block and the A axis becomes perpendicular to the advancing direction.

According to the changes in the advancing direction between blocks in the normal direction control mode (G151, G152), the A axis shift is automatically inserted so that the A axis faces the direction perpendicular to the advancing direction at the start pint of each block. Namely, first the A axis rotates to face the direction perpendicular to the shift command, and then the X axis and Y axis shifts start.

During the tool diameter compensation mode, the A axis is controlled to face the direction perpendicular to the advancing direction of the path after compensation. In the case of single block stopping, it stops at the point after shift command block finishes without stopping between the A axis rotation block and the shift command block. (Position S in the diagram on the right)

The feed rate of the A axis shift inserted at the start point of each block becomes the feed rate which was set by parameter.

However, when dry run is enabled, it becomes the dry run rate. Further, in the case that the X and Y axis shift commands are rapid traverse (G00), it becomes the rapid traverse rate.

In the case of circular interpolation, first the A axis is rotated so that the A axis faces the direction perpendicular to the circular start point, and then the circular interpolation starts. During circular interpolation execution, the A axis faces the normal direction.

The rate of the A axis during circular command is given by the equation below.

$$F * \frac{\text{A axis shift amount (deg)}}{\text{Length of circular arc (mm or inch)}} \quad \text{(deg/min)}$$

F: The feed rate commanded by circular block (mm/min or inch/min)

A axis shift amount means the angular difference between the start point and end point of the block.

(3) Precautions

(Note 1) A axis rotation direction during normal direction control becomes the direction so that it is always less than 180°. (Namely, becomes the shorter path.)

(Note 2) A axis must not be commanded during normal direction control. If it is commanded, that command will be ignored.

(Note 3) Before starting the normal direction control, it is necessary to make a correct correspondence between the A axis coordinate value and the A axis position of the actual device by the coodinate system setting (G92), etc.

(Note 4) A axis must be the rotation axis.

(Note 5) Helical cutting cannot be commanded during normal direction control.

(Note 6) Helical cutting option is necessary for using this function.

14.20 Infeed Control

Cuts a certain fixed amount along the program figure in the specified Y-Z plane for input of external signal for the swing end point.

(1) Method of use

The following shows the program command.

G161 R ——— ;

Figure program

G160 ;

G161 R ——— :

Commands the operation mode and start of start of figure program. Further, specifies the cutting amount by address R.

Figure program:

Programs the workpiece figure in the Y-Z plane by either linear interpolation (G01) or by circular interpolation (G02, G03). Multiple blocks can be commanded.

G160:

Commands cancelling of operation mode (ending of figure program).

14.21 Multi-step Skip

In the multi-step skip function, a coordinate value at the block where P1-P4 after G31 is specified when a skip signal (4 points) turns on is stored in a custom macro variable, a remaining move amount is skipped.

Moveover, G04 dwell can be skipped by parameter.
Which is effective in four skip signals can be chosen by the parameter.
One skip signal can be effective to several Pns (n = 1,2,3,4) by the setting.
A skip signal (4 Max.) from the gauging device, for example, can skip a program being executed.
For example, a series of operations can be automatically performed, including roughing to start-out by entering each skip signal upon completion of roughing, coarse finishing, fine finishing and spark-out by entering each skip signal upon completion of roughing, coarse finishing, fine finishing and speak-out during plunge grinding.

15. COMPENSATION FUNCTIONS

15.1 Tool Offset (G43, G44, G49) (T series)

By setting the difference between tool length assumed when programming and the actual tool length as offsets, workpieces can be machined according to the size commended by the program, without changing the program.

15.2 Tool Length Compensation (G43, G44, G49) (M series)

By setting the difference between tool length assumed when programming and the actual tool length as offsets, workpieces can be machined according to the size commended by the program, without changing the program.

15.2.1 Tool length compensation A

The tool position offset (tool length offset) of the Z axis only can be done by specifying G43 and G44. Offset numbers are selected by specifying an H code.

G43 specifies the tool position offset in the + direction by the offset amount, while G44 specifies it in the - direction by the offset amount. The offset command is cancelled by G49. G43, G44 and G49 modal commands remain effective once they are specified, until other G43, G44, and G49 are specified later.

15.2.2 Tool length compensation B

The tool length offset of an optional axis can be done by plane specification (G17, G18, G19) and G43/G44 commands. The tool position offset of 2 or more axes can also be done by specifying these commands in 2 - 3 blocks.

15.3 Cutter Compensation B, C (G39-G42) (M series)

In the figure above, in order to cut a workpiece indicated as A with an R-radius tool, the path for the center of the tool must be separated R distance from the A.

Tool can be offset by the tool radius value given, that is, to pass through the offset path.

G code	Function
G39	Corner offset circular interpolation (Cutter compensation B only)
G40	Cutter compensation cancel
G41	Cutter compensation left side
G42	Cutter compensation right side

Register the tool radius value related to 2-digit numeric following address H.
Offset plane can be specified by plane specification (G17, G18, G19).
For cutter compensation B, the inside of angle cannot be cut. In this case,it is necessary to specify the appropriate circular to the corner part by program.
For cutter compensation C, even in the case of cutting inside of acute angle, intersection is calculated automatically inside CNC.

Cutter compensation B and C cannot be selected at one time. You must select either one of them.

15.4 Tool Offset Number

(1) T series

Tool offset number ..... Max. 32 pairs
Offset value ..... 0-±999.999mm
0-±99.9999inch

Offset number	OFX (X axis offset amount)	OFZ (Z axis offset amount)	OFR (Tool nose radius compensation amount)	OFT (Imaginary tool node direction)
01	0.040	0.020	0	0
02	0.060	0.030	0	0
03	0	0	0.20	0
04	:	:	:	:
05	:	:	:	:
:	:	:	:	:
:	:	:	:	:

A parameter can be set to switch the X axis tool offset amount to diameter or radius designation.
It is also possible to limit the maximum value of offset amount and the incremental offset input value by the parameter.

(2) M series

Tool offset number :32/64/99/200 tool offsets
Offset number : H00-H99
Offset value : ±6 digit

15.5 Tool Offset Value Program Input (G10)

(1) T series

G10_X(U)_Z(W)_R(C)_Q_;

P : Offset number
X(U) : X axis offset (X ; Absolute, U : Incremental)
Z(W) : Z axis offset (Z ; Absolute, W : Incremental)
R(C) : Tool nose R offset amount (R : Absolute, C : Incremental)
Q : Number indicating the direction of the virtual tool nose direction

The tool offset value may be input in the form of a program. The commands are as follows.
When the offset value is specified by X, Z or R, offset value having the offset number specified by P is replaced with the commanded value.
When the offset value is specified by U, W or C, the commanded value is added to the offset value having the offset number specified by P.
(Note 1) X, Z, R and U, W, C may occur in one block.
(Note 2) This command code in a machining program is useful when machining is done by advancing a tool bit by bit.

(2) M series

G10 P_ R_;
P : Offset number
R : Offset amount

The tool offset value may be input in the form of a program.
Whether the specified offset amount is absolute or incremental is determined by G90/G91.

15.6 Tool Geometry Offset and Tool Wear Offset

Geometry Offset

Offset No.	OFGX (X-axis geometry offset amount)	OFGZ (Z-axis geometry offset amount)	OFGR (Tool nose radius geometry offset amount)	OFT (Imaginary tool nose direction)
G01	10.040	50.020	0	0
G02	20.060	20.030	0	0
G03	0	0	0.20	0
G04	:	:	:	:
G05	:	:	:	:
:	:	:	:	:
:	:	:	:	:

Wear Offset

Wear offset No.	OFWX (X-axis wear offset amount)	OFWZ (Z-axis wear offset amount)	OFWR (Tool nose radius wear offset amount)	OFT (Imaginary tool nose direction)
W01	0.040	0.020	0	0
W02	0.060	0.030	0	0
W03	0	0	0.20	0
W04	:	:	:	:
W05	:	:	:	:
:	:	:	:	:
:	:	:	:	:

It is possible to divide the tool offset to the tool geometry offset for compensating the tool shape or tool mounting position and the tool wear offset for compensating the tool nose wear.

15.7 Tool Nose Radius Compensation (G40-G42) (T series)

G code	Work position	Tool path
G40	(Cancel)	Moving along the programmed path
G41	Right side	Moving on the left side of the programmed path
G42	Left side	Moving on the right side of the programmed path

It is difficult to produce the compensation necessary to form accurate parts when using only the tool offset function due to tool nose roundness in taper cutting or circular cutting. The tool nose radius compensation function compensates automatically for the above errors.

15.8 Stored Pitch Error Compensation

Error caused by machine position, as pitch error of the feed screw, can be compensated. This function is for better machining precision and longer life of the machine.

As the offset data are stored in the memory as parameters, compensations of dogs and setttings can be omited. Offset intervals are set constant by parameters (per axis).

Offset points = 128 (per axis).

Compensation pulse

= (−7 to +7) × (magnification)

Unit : Detection unit

Magnification : ×1, ×2, ×4, ×8, set by parameter(common to all axes)

Compensation space : Set by parameter each axis independentry.

15.9 Backlash Compensation

15.10 Y axis offset (T series)

This system is 4th axis with Y axis. Y axis is offset by offset data.

Displaying and setting Y axis offset data is prepared separately from tool offset by T code.

Tool geometry and wear offset is avairable in Y axis offset with option of tool geometry and wear offset.

(Note 1) When Y axis offset is used, Y axis is as linear axis without fail.

(Note 2) Y axis offset has no functions of tool offset value direct input and too offset value measurement value direct input B.

15.11 Continuous Dressing (GSC)

In the plane grinding canned cycle (G75, G77 to G79), grindstone cutting and dresser cutting is continuously compensated according to the dressing amount by continuous dressing during the grinding execution.

(1) Command method

For the block of the plane grinding canned cycle, the offset number (grindstone wear compensation number) is specified by address L. The compensation amount set in the offset memory corresponding to the specified number becomes the dressing amount.

A maximum of 200 numbers (L01 to L200) can be commanded for the offset number. However, this 200 is the total of the numbers used for the tool diameter compensation (G41, G42) and the tool length compensation (G43, G44). The compensation amount corresponding to the offset number is set in advance by the CRT/MDI panel.

In the block of the plane grinding canned cycle, when L is not commanded, or when L0 is commanded, the compensation operation is not performed.

(2) Compensation operation

Performed for each grinding operation of the operation sequence during the grinding canned cycle (for each shift of X axis), and becomes simultaneous 3 axis interpolation simultaneously with the shift of the X axis by compensation of the Y axis (grindstone cutting) direction and V axis (dresser cutting) direction.

The Y axis shift amout (compensation amount) becomes a value twice the specified dressing amout, and the V axis shift amount becomes a value twice the specified dressing amount (diameter).

16. MEASUREMENT FUNCTION

16.1 Skip Function (G31)

Linear interpolation can be designated as in G01 by programming axis movement following G31. Input of the skip signal during execution of this command interrupts the rest of the block and executes the next block. G31 is one-shot command, effective only in the block concerned.

The custom macro can use the coordinate values of the position where the skip signal was issued, since they are stored in system variables #5061 and #5062 of the custom macro.

#5061 .... X coordinate value
#5062 .... Z/Y coordinate value
#5063 .... Z coordinate value (Only M series)

The skip function can be used when the movement amount is not clear, as in fixed amount feed by grinder.

16.2 High-speed Skip Signal Input

Delay and error of skip signal input is 0 - 2 msec at the NC side (not Considering those at the PMC side).

This high speed skip signal input function keeps this value to 0.1 msec or less, thus allowing high precision measurement. This signal is connected directly to the NC; not via the PMC.

16.3 Tool Length Measurement

Call offset value display screen on the CRT. Relative positions are also displayed on this screen. Reset the displayed relative position to zero. Set the tool for measurement at the same fixed point on the machine by hand.

The relative position display at this point shows difference between the reference tool and the tool measured. The relative position display value is then set instead of numerals as offset amounts, with the same operation as offset amount setting.

16.4 Automatic Tool Offset

When a tool is moved to the measurement position by execution of a command given to CNC, the CNC automatically measures the difference between the current coordinate value and the coordinate value of the commnad measurement position and uses it as the offset amount for the tool. When the tool is moved to the measurement position in offset mode and an amount to be further offset is generated as the result of the measurement, the current offset amount is further offset. When the tool geometry offset & tool wear offset option is selected, the tool wear amount (wear offset amount) is compensated.

(1) Coordinate system

When moving the tool to a position for measurement, the coordinate system must be set in advance. G50XxZz ; sets the coordinate system. (Automatic coordinate system setting may be designated.) (The work coordinate system for programming is used in common.)

(2) Movement to measurement position

A movement to a measurement position is performed by specifying G36Xxa or G37Zza in the MDI or AUTO mode. In this case, the measurement position should be xa or za (absolute command).

Execution of this command moves the tool at the rapid traverse rate toward the measurement position, lowers the feed rate halfway, then continues to move it untill the approach end signal from the measuring instrument is issued. When the tool tip reaches the measurement position, the measuring instrument sends a signal to the CNC which stops the tool.

(3) Offset

The current tool offset amount is further offset by the difference between the coordinate value (α, β) when the tool has reached the measurement position and the value of xa or za specified in G36 Xxa or G37Zza.

Offset amount x

= current offset amount x + (α − xa)

Offset amount z

= current offset amount z + (β − za)

These offset amounts can also be altered from the MDI keyboard.

(4) Feed rate and alarm

The tool, when moving from the starting position toward the measurement position predicted by xa or za in G36 or G37, is fed at the rapid traverse rate across area Ⓐ. Then the tool stops at point T (xa - γx or za - γz) and moves at the measurement feed rate set by parameter across areas Ⓑ, Ⓒ and Ⓓ. If the approach end signal turns on during movement across area Ⓑ, alarm No. 80 is generated. If

the approach end signal does not turn on before point V, the tool stops at point V and alarm No. 80 is generated.

16.5 Direct Measured Value Input A of Tool Offset

The following convenient method can be used to set the offset value as the difference between a standard point, such the tip of the standard tool or the turret center, and the tip of the tool actually used.

A work coordinate system is assumed to have been set in advance.

(1) Cut the workpiece with the tool used in actual machining along surface A in manual mode.
(2) Release the tool along X axis direction without Z axis movement and stop spindle rotation.
(3) Measure distance "β" between surface A and the zero point in the work coordinate system , and set it to the offset number desired as the Z offset value (MZ).
(4) Cut the workpiece along surface B in manual mode.
(5) Release the tool along Z axis direction without X axis movement and stop spindle rotation.
(6) Measure distance "α" and set it to the offset number desired as the X offset value (MX).

Repeat the above operation for each tool used.
If the coordinate value of the tool as surface B is 105.0, the measured distance α is 104.0 and 104.0 is set to offset number 3 of address X as MX, the offset value 1.0 is automatically set to offset number 3 of address X.

(Note 1) This function is effective only when the parameter DOFSI is set to 1.
(Note 2) The X axis must always be measured in diameter.
(Note 3) When the measured value is input as the geometry offset amount, and the corresponding wear offset amount becomes zero.
(Note 4) When the measured value is input as the wear offset amount becomes zero.
Geometry offset amount does not change, while the difference between the whole offset amount and the geometry offset amount is set as the wear offset amount.
(Note 5) This function can be performed by retracting both axes, when PMC and RECORD button are provided.

16.6 Offset Value Counter Input

Push the INPUT button when an address (X, Z, U or W) but not a number has been keyed in to set an offset value, of relative coordinate value U (when the address is X or U) or W(when the address is Z or W) is set as an offset value.
This is used as follows :
(a) Move a standard tool to the standard point manually.
(b) Reset the relative coordinate values U and W to 0.
(c) Move the tool to set an offset value to the standard point.
(d) A desired offset value can be set by setting the offset number and pushing 4X and INPUT button 5Z and INPUT button.

16.7 Direct measured Value Input B of Tool Offset

The tool offset amount of a tool is automatically settable into the tool offset amount memory simply by bringing the tool into contact with the touch sensor by manual operation. The work coordinate system shift amount is automatically settable .
Preset the distance up to the measuring position (contact face of the touch sensor) as a reference value by a parameter using a certain

intrinsic point on the machine tool as the reference measureing position.
By selecting a tool whose tool offset amount is to be measured and positioning it to the measuring position by bringing it into contact with the touch sensor, and sets a difference between the reference value(parameter set value) and the machine coordinate value at the time into the tool offset amount memory as a tool geometry offset value.
The corresponding tool wear offset value is "0".

17. CUSTOM MACRO

17.1 Custom Macro A

The custom macro function previously loads macro instructions created by user into memory as subprograms, and uses them from CNC command programs by a simple procedure at all times.

This custom macro function can employ macro instructions, such as CNC commands using variables, arithmetic operation, conditional branch, etc. as well as ordinary CNC commands.

The following functions are applicable when programming the custom macro body.

(1) Variables

(a) Representation of variables

Variables are represented by variable numbers following # as follows.

#i (i = 1, 2, 3, ...)

(b) Quotation of variables

Numeric values following an address can be replaced with variables.

(Examples)

F#103 ..... Specifies the feedrate of #103 value.

G00Z-#100 .. Specifies the positioning to the -#100 position.

G#130 ..... Specifies G code by the #130 value.

If variable number i is replaced with variable #j, don't describe ##j, but describe #9j.

(where #j = i)

(Example)

#9100 = -500 when #100 = 105 and #105 = -500

(2) Kinds of variables

(a) Common variables

The common variables are variables common to main programs and respective macros to be called by these main programs. Accordingly, a common variable value arithmetically operated in a certain macro can also be used in the other macros.

(b) System variables

The system variables mean the variables whose applications are predetermined by the system. The offset amount, interface input signals, position information, and other data can be known from the predetermined variables. The offset amount can also be set by substituting the offset amount.

(3) Macro instruction

The macro instruction is generally represented in the following format.

G65 Hm P #i Q #j R #k ;

#i: Name of variable to which arithmetic result is entered.

#j: Name of variable to be calculated (A constant is also allowable.)

#k: Name of variable to be calculated (A constant is also allowable.)

Hm: Operator (For details, refer to the following table.)

(Note 1) Be careful since no decimal point is applicable to a variable value.

(Note 2) The unit of angle is 0.001°.

Macro Instructions

G code	H code	Functions	Definition
G65	H01	Definition, substitution	#i = #j
G65	H02	Addition	#i = #j + #k
G65	H03	Subtraction	#i = #j − #k
G65	H04	Multiplication	#i = #j × #k
G65	H05	Division	#i = #j ÷ #k
G65	H11	OR	#i = #j . OR . #k
G65	H12	AND	#i = #j . AND . #k
G65	H13	Exclusive OR	#i = #j . XOR . #k
G65	H21	Square root	#i = $\sqrt{\#j}$
G65	H22	Absolute value	#i = I #j I
G65	H23	Remainder	#i = #j − trunc (#j / #k) × #k
			trunc : Truncation of decimal part
G65	H24	Conversion from BCD	#i = BIN (#j)
G65	H25	Conversion into BCD	#i = BCD (#j)
G65	H26	Composite multiplication and diviation	#i = (#i × #j) ÷ #k
G65	H27	Composite square root 1	#i = $\sqrt{\#j^2 + \#k^2}$
G65	H28	Composite square root 2	#i = $\sqrt{\#j^2 - \#k^2}$
G65	H31	Sine	#i = #j · SIN (#k)
G65	H32	Cosine	#i = #j · COS (#k)
G65	H33	Tangent	#i = #j · TAN (#k)
G65	H34	Arc tangent	#i = ATAN (#j / #k)
G65	H80	Unconditional branch	GO TO n
G65	H81	Conditional branch 1	IF#j = #k, Go To n
G65	H82	Conditional branch 2	IF#j ≠ #k, Go To n
G65	H83	Conditional branch 3	IF#j > #k, Go To n
G65	H84	Conditional branch 4	IF#j < #k, Go To n
G65	H85	Conditional branch 5	IF#j ≥ #k, Go To n
G65	H86	Conditional branch 6	IF#j ≤ #k, Go To n
G65	H99	P / S alarm generation	Generation of P/S alarm 500 + n

(4) Others

The following functions are available when PMC is combined as an option.

(i) Entry of 16-bit external input signals.

(ii) Output of 16-bit data to the external devices.

17.2 Custom Macro B

(1) Use of variables

Variables: #i (i = 1, 2, 3, ...)

Quotation of variables: F#33 (#33: speed expressed by variables)

(2) Operation between variables

Various operation can be done between variables and between variables and constants.

The following operations, and functions can be used:

+ (sum), − (difference), *(product),/ (quotation), OR (logical sum), XOR (exclusive logical sum), AND (logical product), SIN (sine), COS (cosine), TAN (tangent), ATAN (arc tangent), SQRT (square roots), ABS (absolute value), BIN (conversion from BCD to binary), BCD (conversion from binary to BCD), FIX (truncation below decimal point), FUP (raise fractions below decimal point), ROUND (round)

Example)

#5 = SIN #2 + #4 * 3.14 + #4 * ABS (#10)

(3) Control command

Program flow in the custom macro body is controlled by the following command.

(a) If [<Conditional expression>] GOTO n (n = sequence number)

When <conditional expression> is satisfied, the next execution is done from block with sequence number n.

When [<conditional expression>] is not satisfied, the next block is executed.

When the [<IF conditional expression>] is committed, it executes from block with n unconditionally.

The following <conditional expressions> are available:

#j EQ #k whether #j = #k

#j NE #k whether #j ≠ #k

#j GT #k whether #j > #k

#j LT #k whether #j < #k

#j GE #k whether #j ≧ #k

#j LE #k whether #j ≦ #k

(b) WHILE [<conditional expression>] DO m (m = 1, 2, 3)

:

END m

While <conditional expression> is satisfied, blocks from DO m to END m is repeated.

When <conditional expression> is no more satisfied, it is executed from the block next to

END m block.

Example #120 = 1 ;

WHILE #120 LE 10 DO 1 ;

(4) Format of custom macro body

The format is the same as the sub program.

```
OMacro number;

Custom macro body

M99;
```

(5) Custom macro instruction

(a) Simple call

G65P (macro number) L (times to repeat) <argument assignment>;

A value is set to a variable by <argument assignment>.

Write the actual value after the address.

Example A5.0E3.2M13.4

There is a regulation on which address (A - Z) corresponds to which variable number.

(b) Modal call A

G66P (macro number) L (times to repeat) <argument assignment>;

Each time a move command is executed, the specified custom macro body is called. This can be cancelled by G67.

This function is useful when drilling cycles are programmed as custom macro bodies.

(c) Macro call by G codes

The macro can also be called by the G codes set by parameters. Instead of commanding:

N_G65 P_ <argument assignment>;

or

N_G66 P_ <argument assignment>; ,

macro can be called just by commanding:

N_G_ <argument assignment>; .

G code G_ for calling the macro, and macro program number P_ to be called, are coupled togehter and set as parameter.

Maximum ten G codes from G01 to G255 can be used for macro call (G00 cannot be used).

The G code macro call cannot be used in the macro which was called by a G code. It also cannot be used in sub programs called by sub program call with M codes or T codes.

(d) Sub program call by M code

An M code can be set by parameter to call a sub program. Instead of commanding:

N_G_X_Y_....M98 P_ ; ,

the same operation can be performed simply by commanding:

N_G_X_Y_....M_ ;.

As for M98, M codes are not transmitted.

The M code M_ for calling the sub program and the sub program number P_ to be called are coupled together and set by parameter.

Maximum three M codes from M03 to M97 can be used for macro call (M30 cannot be used).

Arguments cannot be transmitted. It also cannot be commanded in the same block as the block with M98 command.

When these M codes are commanded in macro called by G code or in sub program called by M code or T code, they are regarded as ordinary M codes.

(e) Sub program call by T code

By setting parameter, sub program can be called by T codes. When commanded:

N_G_X_Y_....Tt; ,

the same operation is done as when commanded:

#149 = t;

N_G_X_Y_....M98 P90000; .

The T code t is stored as arguments of common variable #149.

This command cannot be done in the same block with a sub program calling M code, or with M98 command.

When T code is commanded in macros called by G code, or in sub programs called by M codes or T codes, the T code is treated as ordinary T codes.

(6) Types of variables

Variables are divided into local variables, common variables, and system variables, according to their variable numbers. Each type has different use and nature.

(a) Local variables #1 - #33

Local variables are variables used locally in the macro. Accordingly, in case of multiples calls (calling macro B from macro A), the local variable used in macro A is never cleared by being used in macro B.

(b) Common variables #100-#149, #500 - #531

Compared with local variables used locally in a macro, common variables are common throughout the main program, each sub program called from the main program, and each macro. The #1 used in a certain macro is the same as the #i used in other macros. Therefore, a common variable #i calculated in a macro can be used in any other macros.

Common variables #100 to #149 are cleared when power is turned off, but common variables #500 to #531 are not cleared after power is turned off.

(Note) It is possible to increase number of common variables. For details, see "Number of common variables".

(c) System variables

A variable with a certain variable number has a certain value. If the variable number

is changed, the certain value is also changed.

The certain value are the following:

(i) 48 points DI (for read only)
(ii) 48 points DO (for output only)
(iii) Tool offset amount, work zero point offset amount
(iv) Position information (actual position, skip position, block end position, etc.)
(v) Modal information (F code, G code for each group, etc.)
(vi) Alarm message (Set alarm number and alarm message, and the NC is set in an alarm status. The alarm number and message is displayed on the CRT.)
(vii) Clock (Time can be known. A time can also be preset.)
(viii) Single block stop, Miscellaneous function end wait hold
(ix) Feed hold, Feedrate override, Exact stop inhibition
(x) Mirror image status

(7) Limitations

(a) Usable variables
#1 - #33, #100 - #149, #500 - #531, and system variables.

(b) Usable variable values
$\pm 10^{38}$ - $\pm 10^{-38}$

(c) Constants usable in expression
±0.0000001 - ±99999999

(d) Arithmetic precision
8-digit decimal number (in trigonometrical functions, some value may cause fall in precision).

(e) Custom macro body call nesting
Maximum 4 folds

(f) ( ) nesting
Maximum 5 folds.

(g) The full key type CRT/MDI is used.

17.3 Custom Macro Modal Call

The macro call mode can be specified by executing the following command.

G66 P (program number);

The macro call mode can be cancelled by executing the following command.

G67;

A macro designated by address P of G66 block is called every CNC command block in the macro call mode.

All commands of each block other than address O and N serve as arguments without being executed.

(Note 1) No call is done in G66 and G67 blocks.

(Note 2) No call is done even in G66 mode in the G65 block, but the command is executed as normal macro command.

(Note 3) The macro mode and macro call cancel mode are selectable by specifying G66 and G67 in the MDI mode. However, if another command is specified, it is processed as a normal command without executing any macro call.

(Note 4) All commands other than addresses O, N and P specified in G66 and G67 blocks are neglected.

(Note 5) The number of repetition times of macro call cannot be specified. For the address P specified by G66 block, the lower significant 4 digits only are effective.

(Note 6) The multiplicity is 1 in macro call. The multiplicity of subprograms is maximum 4. The multiplicity is maximum 4 when macro call and a subprogram are combined with each other.

17.4 Pattern Data Input

This function is provided to enable users to conduct CNC machining by taking numeric data (pattern data) only out of a drawing and setting these numerical values from CRT/MDI unit without any need of programming using an existing CNC language.

With the aid of this function, a machine tool builder can freely prepare a boring cycle (boring cycle, tapping cycle, or the like), for example, by utilizing the custom macro function, and load the boring cycle into the program memory.

(1) This cycle is designated as pattern names like BOR1, TAP3, DRL2, ...
An operator can select a desired pattern out of the menu of pattern names being displayed on CRT.

(2) A cycle should be prepared by using variables (pattern data) which are specified by the operator.
The operator can identify these variables as depth, return relief, feed, blank, and other pattern data names. The operator gives values (pattern data) to these names.

17.5 Order-made Macro

A machine tools builder is requested to optionally select macros out of the standard macro prepared by FANUC and macros requested by the machine tool builder. These macros are loaded into an ROM cassette. This ROM cassette can be mounted inside CNC, and optionally called in the programming process.

(Note) The macro executer and order-made macro cannot be specified at the same time.

17.6 Macro Executer

The macro executer is a control function of reading a macro program from the macro cassette for execution.

It is possible to execute the following two macro programs at the same time.

(1) Macro program called and executed by M/T/G codes from the machining program

(2) Screen preparing macro program called by macro screen (AUX screen) selection

The macro program should be created at the machine tool builder side. Do this in the same procedure as in the custom macro program and store it in ROM cassette. Consequently, unlike the custom macro, the macro program does not occupy the tape storage area. Since the macro program is converted to an executable format for storage, the execution speed is higher than that for custom macro, thus allowing high-speed machining operation.

The macro program has the following functions:

(1) F10/11/12 format custom macro

(2) Generation of unique graphic screen

(3) Reading PMC status via PMC window and outputting it to PMC

(Note) When a macro executer is equipped, no order-made macro can be designated.

17.7 Interruption Type Custom Macro

Outline of Interruption type custom macro

Entering an interrupt signal (UNIT) from the machine during execution of a program permits call-out of another program.

This function is called the interruption type custom macro.

The interrupt command is executed by specifying, in programming,

M96Pxxxx ;(Custom macro interrupt on)

M97 ;(Custom macro interrupt off)

This function permits any execution block of the program to call out another program, thus enabling program operations according to the ever changing conditions.

18. AXIS CONTROL

18.1 Mechanical Handle Feed

It is possible to move the machine by hand, using the mechanical handle installed on the machine; not by the CNC (servo motor).
Move distance by the mechanical handle is followed up and actual position in the CNC is revised. The mechanical handle feed is done by inputting the servo off signal of the axis fed. Machine side signal desides on/off of follow up function when servo off. However, the follow up can not be applied when emergency stop is on or when servo alarm is on. Inch/metric conversion cannot be used when mechanical handle feed function is used.

18.2 Servo Off

Servo on/off control per axis is possible by signals from the machine side.

Following functions can be executed by servo off.

(1) Workpiece install/remove fixing the machine at stated position.
(2) Machine operation by operator with mechanical handle.

18.3 Mirror Image (M series)

18.4 X-Axis Mirror Image (T series)

18.5 Additional Axis Neglect

By the signal from machine, CNC with additional axis can be set as if without additional axis.
The signs of the X, Y axis motion can be inverted in automatic operation mode.
This mirror image is effective or ineffective according to setting parameters.

19. AUTOMATIC OPERATION

19.1 Operation Mode

(i)Memory operation

After the program is once registered in memory, the machine can be run according to the program instructions. This operation is called memory operation.

(ii) MDI operation

After the program is entered, as an command group, from the MDI keyboard, the machine can be run according to the program. This operation is called MDI operation.MDI operation is executed by 1 block.

19.2 Selection of Execution Program

Program number search	Program number search by MDI (O .....)
	Program 0001 to 0015 search by signal from Machine side
Sequence number	Sequence number (N .....) search by MDI

19.3 Activation of Automatic Operation

Pressing the cycle start pushbutton causes automatic operation start. By pressing the feed hold or reset pushbutton, automatic running pauses or stops. By specifying the program stop or program termination command in the program, the running will stop during automatic operation. When each machining is completed, automatic operation sops.

19.4 Execution of Automatic Operation

Buffer register equivalent to one block is available for program read and control of CNC command operation intervals caused by preproces time.

19.5 Automatic Operation Stop

19.5.1 Program stop (M00, M01)

In a block where M00 (program stop) or M01 (optional stop) is specified, M code and M code read signals are sent to the machine tool in the same way as in general M codes.

Program stop processing must be done on the machine tool side.

Set CNC to be waiting for the end signal in case of M00 (program stop), and the CNC stops operating.

Send the end signal to CNC by the cycle start signal, and the program is continued.

Perform the same processing as in M00 only when the optional stop switch is turned on by the machine tool operator's panel in case of M01 (optional stop).

19.5.2 Program end (M02, M30)

In a block where M02 (end of program) or M30 (end of tape) is specified, M code and M code read signals are sent to the machine tool in the same way as in general M codes.

Program end processing must be done on the machine tool side. By returning an external reset signal to M02 or M30, processing returns to the start of the executing program to reset the program.

By returning the end signal to M02 or M30, processing returns to the start of the executing program to continue the execution from the start again.

19.5.3 Feed hold

The CNC can be brought to an automatic operation hold status by pressing the feed hold button on the operator's panel. When feed hold is commanded during motion, it decelerates to a stop.

Automatic operation can be restarted by the cycle start button.

19.5.4 Thread cutting cycle retract

When feed hold is commanded during thread cutting cycle by G76 or G78, the tool rapidly relieves to the cycle start point,like the final chamfering of the thread cutting cycle.

Thread cutting cycle restarts by cycle start command.

Without this function, if feed hold is commanded during thread cutting,it stops at the end point of the block next to the block in which thread cutting has ended.

19.5.5 Reset

The automatic operation can be ended in a reset status by the reset button on the CRT/MDI panel or by the external reset signal, etc. When reset is commanded during motion, it decelerates to a stop.
Modal information can be initialized by the parameter.

19.5.6 Sequence number comparison and stop

When a block having the same sequence number as the preset sequence number appears during plotting the machining process, the single block stop condition is set after executing the block.
This function is effective only when graphic drawing.

19.6 Manual Interruption during Automatic Operation

19.6.1 Handle interruption

During automatic operation, tool can be adjusted by the manual pulse generator without changing the mode. The pulse from the manual pulse generator is added to the automatic operation command and the tool is moved for the recommended pulses. The work coordinate system thereafter is shifted for the pulses commanded.
Movement commanded by handle interruption can be displayed on the CRT screen.

19.6.2 Position signal output

Machine tool position	-10000	0	10000	20000	30000
Position signal output	-1	0	1	2	

The full stroke of each axis of the machine tool is divided into 256 grids respectively, and the grid number is output as a binary 8-bit position signal every axis according to on which grid the machine is positioned at present.
The position signal output is effective after turning on the power supply, after cancelling the emergency stop, or after the reference point return.

19.6.3 Manual absolute ON/OFF

(1)When the switch is ON.

(2)When the switch is OFF.

This switch selects whether the amount of manual movement is to be added to the absolute value.

19.7 Program Restart

- when a tool is damaged
- power is turned OFF once
- the emergency stop button is depressed once
- the coordinate system is changed after stopping automatic operation previously

machining operation can be performed from the block to be restarted.

When a tool is damaged, or it is desired to restart machining after the stopping machining once, specity a sequence No. of the block to be restarted:
Machining will be restarted from its block.
Further, this function can be used as a high-speed tape checking function.

20. MANUAL OPERATION

20.1 Manual Feed

(1) Jog feed The jog feedrate can be selected in 15 steps as tabulated below by a rotary switch.These 15-step ratios form geometrical series.(Refer to the next table.)

(2) Manual rapid traverse The rapid traverse is also effective during manual operation. Manual rapid traverse is set as parameter, but this parameter is different from the automatic operated rapid traverse.The rapid traverse rate preset by parameters can also be overridden in 4 steps.

Rotary switch position	Feed rate	
	Metric input mm/min	Inch input inch/min
0	0	0
1	2.0	0.08
2	3.2	0.12
3	5.0	0.2
4	7.9	0.3
5	12.6	0.5
6	20	0.8
7	32	1.2
8	50	2.0
9	79	3.0
10	126	5.0
11	200	8.0
12	320	12
13	500	20
14	790	30
15	1260	50

(Note) Manual pulse generator is usable in the JOG mode by a parameter.

20.2 Incremental Feed

High-efficiency positioning can be realized in the manual mode by using the step value given below.When the manual pulse generator (option) is used,this function is not applicable.

Input	×1	×10	×100	×1000
Metric input	0.001mm	0.01mm	0.1mm	1mm
Inch input	0.000 1inch	0.00 1inch	0.01inch	0.1inch

20.3 Manual Handle Feed

By rotating the manual pulse generator, the axis can be moved for the equivalent distance.Manual handle feed is controlled 1 axis at a time.The manual pulse generator generates 100 pulses per rotation. Move amount per pulse can be specified from the following magnifications:

x1, x10, xm

With this function incremental feed function cannot be specified.

Input	×1	×10	×m
Metric input	0.001mm	0.01mm	0.001mm ×m
Inch input	0.0001inch	0.001inch	0.0001inch ×m

Up to three manual pulse generators can be connected. Manual pulse generator is usable in the Jog mode by a parameter.

21. PROGRAM TEST FUNCTIONS

21.1 Machine Lock

In machine lock condition,the machine does not move,but the position display is updated as if the machine were moving.

(Note 1) When the G27, G28 or G30 command is specified, the machine does not move to the reference point and the REFERENCE POINT RETURN lamp does not light.

(Note 2) The M,S and T functions are executed.

21.2 Auxiliary Function Lock

If automatic operation is executed in the condition of auxiliary function lock,the auxiliary functions (such as spindle rotation,tool change,coolant on/off) stop operation.

21.3 Dry Run

Remove the work,check only movement of the tool. Select the tool movement rate using the dial on the operator's panel

The feed rate specified in the program is ignored,and the machine is fed at the following feed rate.

Rapid traverse button ON/OFF	Rapid traverse	Cutting feed
Rapid traverse button ON	Rapid traverse	Max. JOG feed rate
Rapid traverse button OFF	JOG feed rate(Note)	JOG feed rate

(Note) Dry run for rapid traverse can be made effective by the parameter setting.

21.4 Single Block

When the cycle start pushbutton is pressed, the tool executes one operation then stops. By pressing the cycle start again, the tool executes the next operation then stops.The program is checked in this manner.

(Note 1) SINGLE BLOCK is effective at an intermediate point in G28,G30.

(Note 2) SINGLE BLOCK is not effective to the block M98_;,M99,and G65.However if M98,M99 blocks have the address other than O,N,P,SINGLE BLOCK is effective.

21.5 Z-axis Command Cancel

Only the Z axis can be locked by the machine side signal.

It is effective when using pen writing a movement for CNC program check.

22. CRT/MDI

22.1 CRT/MDI Panel

I1) Small 9 inch monochrome CRT/MDI (with softwares key)

(a) 0-T

(b) 0-M

(2) 9 inch monochrome CRT/MDI (without softwares key)

(a) 0-T

(b) 0-M

(3) Small 9 inch color CRT/MDI

(a) 0-T

(b) 0-M

(4) "Full key CRT/MDIpanel

(a) 0-T

(b) 0-M

(5) CRT/MDI panel for MMC

(a) 00-T

(b) 00-M

· Keyboard functions

No.	Name	Functions
(1)	RESET key	Press this key to reset the CNC, to cancel an alarm, etc.
(2)	START key	Press this key to start the MDI commands, or to start the automatic operation cycle.Since this key's operating method differs according to the machine tools, refer to the machine tool builder's operation manual.
(3)	Address and numerical key	Press these keys to input alphabetic, numeric, and other characters.
(4)	INPUT key	Press this key to input parameter or offset etc. and also to start inputting of I/O device, or to input command data under MDI mode.
(5)	Cancel (CAN) key	Press this key to delete the character or symbol input to the key input buffer. (example):When key input buffer display is N0001, press the cancel (CAN) key to cancel and delete the N0001.
(6)	Cursor shift keys	There are two kinds of cursor shift key described below. ↓ : This key is used to shift the cursor a short distance in the forward direction. ↑ : This key is used to shift the cursor a short distance in the reverse direction.
(7)	Changeover key	Two kinds of page changeover keys are described below. ↓ : This key is used to changeover the page on the CRT screen in the forward direction. ↑ : This key is used to changeover the page on the CRT screen in the reverse direction.

Function buttons

Function buttons indicate large items like chapters in a document.

POS ..... Indicates the current position.

PRGRM ..... Performs the following:

In EDIT mode ..... edit and display of the program in the memory

In MDI mode ..... input and display of the MDI data

In automatic operation .... display of command value.

MENU OFSET .. Setting and display of offset value.Display menu screen

DGNOS PARAM Setting and display of parameter, and display of self diagnostic data.

OPR ALARM ... Alarm number display and setting and display of software operator's panel.

AUX GRAPH ... Graphic display

Several pages are included in the chapter selected with each function button. The page is selected with PAGE button.

(Note) The data displayed on the screen disappears when one of the function buttons and the CAN key are pressed.Thereafter, when either function button is pressed, the corresponding screen is displayed again.When the unit is not used with the power turned on for long time, turn off the screen.This is effective to prevent the image quality from deteriorating.

22.2 Setting and Display

With keys, data stored in CNC memory is updated and displayed on the screen .

22.2.1 MDI operation / MDI operation B

(T series)

(M series)

In MDI mode, one block operation is possible. Also, plural blocks operation is possible for MDI operation B (option).

22.2.2 Setting and display of offset value

(T series)

```
OFFSET                                  O1000 N0000
  NO.        X            Z            R     T
  01      10.000      -10.000       0.000   0
  02       0.000        0.000       0.000   0
  03       0.000        0.000       0.000   0
  04      40.000      -40.000       0.000   0
  05       0.000        0.000       0.000   0
  06       0.000        0.000       0.000   0
  07       0.000        0.000       0.000   0
  08       0.000        0.000       0.000   0

ACTUAL POSITION (RELATIVE)
    U      0.000          W        0.000
ADRS.                                S     0 T
                                    MDI
 [OFFSET] [      ] [W.SHFT] [MACRO] [      ]
```

(M series)

```
OFFSET                                  O2000 N2000
 NO.      DATA          NO.      DATA
 001     10.000         009       0.000
 002     -1.000         010      10.000
 003      0.000         011     -20.000
 004      0.000         012       0.000
 005     20.000         013       0.000
 006      0.000         014       0.000
 007      0.000         015       0.000
 008      0.000         016       0.000
ACTUAL POSITION (RELATIVE)
    X     0.000          Y        0.000
    Z     0.000
NO.  001 =
                                    MDI
 [OFFSET] [MACRO] [      ] [      ] [      ]
```

22.2.3 Setting and display of setting data

(T series)

```
PARAMETER                               O1000 N0000

  (SETTING 1)
  _TVON = 0
   ISO  = 0   (0:EIA 1:ISO )
   INCH = 0   (0:MM  1:INCH)
   I/O  = 0
   SEQ  = 0

NO. TVON =                          S     0 T
                                   MDI
 [PARAM] [DGNOS] [      ] [      ] [      ]
```

(M series)

```
PARAMETER                               O2000 N2000

  (SETTING 1)
  _REVX = 0
   REVY = 0
   TVON = 0
   ISO  = 0   (0:EIA 1:ISO )
   INCH = 0   (0:MM  1:INCH)
   I/O  = 0
   ABS  = 0   (0:INC 1:ABS )
   SEQ  = 0

NO. REVX =
                                   MDI
 [PARAM] [DGNOS] [      ] [      ] [      ]
```

Apart from parameters, there is data that is set by the operator in operation. This data causes machine characteristics to change. For example, it specifies the following:

(1) Setting of TV check effective/ineffective
(2) ISO/EIA setting
(3) Inch/metric setting

The above data is called setting data.

22.2.4 Setting and display of parameter

(T series)

(M series)

The CNC functions have versatility in order to take action in characteristics of various machines.For example, CNC can specify the following:

(1) Rapid feed rate of each axis.

(2) Setting of least input increment (inch system/metric system)

(3) Setting of CMR/DMR

Data to make the above specification is called parameters. Parameters are characterized by each machine tool.

22.2.5 Setting and display of custom macro variables

(T series)

(M series)

Common variables (#100-#149, #500-#531) can be set and displayed on CRT.

22.2.6 Program display

22.2.7 Program list display

(T series)

(M series)

(a) PROGRAM NO.

USED: The number of the programs registered (including the subprograms).

FREE: The number of programs which can be registered additionally.

(b) MEMORY AREA

USED: The capacity of the program memory in which data is registered (indicated by the number of characters).

FREE: The capacity of the program memory which can be used additionally (indicated by the number of characters).

(c) PROGRAM DIRECTORY:

Program Nos. registered are indicated. When more than 48 programs are registered and P1 or P2 is designated, 48 program Nos. from the beginning are indicated. When P3 or P4 is designated, 48 program Nos. from the last are indicated.

22.2.8 Current Position Display

Position of the current tool can be displayed by coordinate values of each coordinate system.The distance from the current position to the target position can also be displayed.The actual speed can be displayed on the actual position display screen at the same time.

(1) In the work coordinate system

(T series)

(M series)

The current value in a programmed coordinate system set with G92 or G50, is displayed.

(2) In the relative coordinate system

(T series)

(M series)

The relative position is displayed in which the position reset by an operator is zero.

Reset operation

When this is displayed, press the X,Y,Z,or 4th key.The address on the display will flicker.Then press the CAN button.The relative position of the flickering address will be reset to zero.

(3)Overall display

The current position in the following coordinate system is displayed simultaneously:

(a) Position in the relative coordinate system (RELATIVE)

(b) Position in the work coordinate system (ABSOLUTE)

(c) Position in the machine coordinate system (MACHINE)

(d) Residual movement amount (DISTANCE TO GO)

(T series)

```
ACTUAL POSITION                    O0001 N0000
   (RELATIVE)                 (ABSOLUTE)
 X   437.942                X   438.004
 Z   657.006                Z   657.006

   (MACHINE)                (DISTANCE TO GO)
 X   438.004                X   561.934
 Z   657.006                Z   842.901

                      PART COUNT          15
RUN TIME        1H19M CYCLE TIME  0H 1M10S
ACT.F     1000 MM/M
20:40:15                    MDI
 ABS   REL   ALL
```

(M series)

```
ACTUAL POSITION                    O0001 N0000
   (RELATIVE)                 (ABSOLUTE)
 X   294.296                X   294.296
 Y   353.156                Y   353.156
 Z   441.445                Z   441.537

   (MACHINE)                (DISTANCE TO GO)
 X   294.358                X   705.641
 Y   353.230                Y   846.770
 Z   441.537                Z  1058.462

                      PART COUNT          15
RUN TIME        1H20M CYCLE TIME  0H 0M46S
ACT.F     1000 MM/M
21:35:38                    MDI
 ABS   REL   ALL
```

The residual movement amount indicates the movement amount remaining in one block of a command, and is displayed in AUTO or MDI mode. The unit of the machine coordinate system is the same as that of the machine system.

22.2.9 Alarm Display

(T series)

```
ALARM MESSAGE                      O1000 N1000
  520  OVER TRAVEL : +Z

                                   S    0 T
          ALARM                    MDI
 ALARM   OPR
```

(M system)

```
ALARM MESSAGE                      O2000 N2000
  510  OVER TRAVEL : +X
  530  OVER TRAVEL : +Z

          ALARM                    AUTO
 ALARM   OPR
```

When an alarm occurs, the screen automatically displays the alarm screen. Alarm numbers and simple messages are displayed.

22.2.10 Command value display

(T series)

```
PROGRAM                          O1000 N0102
  (CURRENT)                     (MODAL)
     X  2000.000                  F   11.0000
G01  Z   200.000          G01 M
     F   11.0000          G97 S
     T      0303          G69 T      0303
                          G99
                          G21 WX    0.000
                          G40 WZ    0.000
                          G25

                            SACT      0
                              S     0 T0303
                           AUTO
[PRGRM] [CURRNT] [NEXT] [ ] [ ]
```

(M series)

```
PROGRAM                          O2000 N0130
  (CURRENT)                     (MODAL)
      X     17.500                  F      2000
  G01 F       2000          G01 R
  G17 H          2          G17 P
                            G91 Q
                            G94 H          2
                            G21 M
  G41                       G41 S
                            G49 T
  G80                       G80
                            G98
                            G67

                         BUF AUTO
[PRGRM] [CURRNT] [NEXT] [ ] [ ]
```

22.2.11 Run time display / parts count display

(T series)

```
ACTUAL POSITION (ABSOLUTE)
O1000 N0002
X      100.000
Z      200.000
                          PART COUNT    5
RUN TIME      1H20M  CYCLE TIME 0H 0M 09
                              S    0 T
                           MDI
[ABS] [REL] [ALL] [ ] [ ]
```

(M series)

```
ACTUAL POSITION (ABSOLUTE)
O2000 N0190
X     -12.500
Y     -42.500
Z       0.000
                          PART COUNT    7
RUN TIME      0H12M CYCLE TIME  0H 0M48S
                           AUTO
[ABS] [REL] [ALL] [HNDL] [ ]
```

PART COUNT:

Indicates the number of parts.When M02 or M30 is executed, it is increased by +1.

To preset the PART COUNT to "0", press NO.LQP and CAN key.

When the number of parts reach a target value, the signal is sent to PMC.

RUN TIME:

Indicates the total run time during automatic operation,excluding the stop and feed hold time.To preset the RUN TIME to "0", press 3 and CAN key. (It is not preset to "0" by cutting off power.)

CYCLE TIME:

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to "0" when a cycle start is performed at reset status.

(It is preset to "0" by cutting off power.)

22.3 Software Operator's Panel

(T series)

(M series)

With this function, functions of the switches on the machine operator's panel can be conducted from the CRT/MDI panel. In other words, mode switch and jog override switch can be eliminated from the machine operator's panel. The following can be conducted from the CRT/MDI panel.

(a) Group 1:
Mode selection

(b) Group 2:
Selection of jog feed axis, jog rapid traverse

(c) Group 3:
Selection of manual pulse generator feed axis, selection of manual pulse magnification

(d) Group 4:
Jog feedrate, feedrate override, rapid traverse override

(e) Group 5:
Optional block skip, single block, machine lock, dry run

(f) Group 6:
Protect key

(g) Group 7:
Feedhold

Whether to operate these switches from the CRT/MDI or machine operator's panel can be determined by the parameter for each group.

22.4 General Purpose Switch for Software Operator's Panel

Machine tool builder can add eight optionally definable switches as an extended function of the software operator's panel.The name of these switches can be set by parameters as character strings of max. 8 charactors.

22.5 Program Protect Key

Program memory is protected from breaking according to error operation by outputting the signal from machine side to inhibit the writing to program memory.Provide this switch on machine side operator's panel

22.6 Graphic Display

Tool path can be plotted during operation.In the program debug mode, a program can be checked using tool paths plotted on the CRT screen by turning on machine lock (MLK) without moving the machine tool.

22.7Dinamic graphic display

(1) T series

The blank figure,chuck and tailstock are all realiatically described, and cutting progress can be checked as the tool moves on the CRT screen.

(2) M series

The changing profile of the workpiece is simulated according to the movements of the tool as programmed.Because the profile is shown in 3 dimensions,the operator can check the machining program in detail with great ease.

The coodinate axis and projection angle can be changed at random.

In addition to 3-dimensional drawings, it is also possible to describe 2-dimensional drawings or tri-plane drawings simultaneously.

22.8 Language

- Japanese (Kanji)
- English
- Germany
- French
- Chinese
- Korean
- Italian
- Spanish

22.9 Created Tool Post Name Display (TT)

Any character strings composed of numerals, alphabets and symbols can be displayed on the CRT screen as a name of a tool post as well as "Tool Post 1" and "Tool Post 2".

Character codes are set in the parameter Nos. 257 to 263 and the corresponding characters are displayed on the CRT screen. However, if 0 is set as a character code, the character of "Tool Post 1" or "Tool Post 2" corresponding to the position in which 0 has been set will be displayed.

23. PART PROGRAM STORAGE & EDITING

23.1 Part Program Storage & Editing

(1) A number of main programs and subprograms can be input from MDI and stored into the memory.Programs can also be input from the paper tape, etc. by using the optional reader/puncher interface.
(2) Main programs and subprograms being stored in the memory can be edited (modified, deleted, or inserted).
(3) An optional main program being stored in the memory can be called and edited.
(4) An optional program can be deleted.
(5) Several sequential blocks can be deleted at a time (program delete).

23.2 Number of Registered Programs

(Note) Part program storage length is 1 m shorter in case of 200 registered programs.

23.3 Part Program Storage Length

Program capacity	10/20/40/80/120/320m
Memory element	IC memory
Battery	Back-Up

23.4 Play Back

When selecting the playback option, the TEACH IN JOG mode and TEACH IN HANDLE mode are added.In these modes, programs can be created by loading the machine tool position obtained by manual operation into memory as a program position. Words (O, N, G, R, F, H, I, K, M, S, T, P, Q, C, EOB) other than X, Y, Z, A,B, C are also be loaded into memory by just the same operation as in the EDIT mode.

23.5 Graphic conversational Input

See item 14.10

23.6 Pattern Data Input

See item 17.4.

23.7 Menu Programming

See item 14.11.

23.8 I/O Device External Control

Part program registration can be commanded externally.A part program can be registered in memory in the edit mode through the input device selected for editing using the external read start signal .

23.9 Background Editing

Regardless of mode selection and CNC status (during automatic operation or not, etc.), background editing operation can be performed.No alarm taking place in background edit affects foreground operation.

In contrast,background editing is not affected by foreground alarm.

In background editing, the following operation becomes possible.

(1) Program registration from MDI
(2) Program registration from CNC tape
(3) CNC tape punching
(4) Program editing

23.10 Simple Synchronous Control

The travelling command of master axis is given to two motors of master and slave axes in a simple synchronous control.However,no synchronous error compensation or synchronous error alarm is detected for constantly detecting the position deviation of the master and slave axes to compensate the deviation.

In the reference point return,the master and slave axes similarly move until the deceleration operation is performed. After that, the detection of grid is performed independently.

The pitch error and backlash compensation are independently performed for the master and slave axes.

An input signal from the machine side can select whether the axis travelling is carried out based on the travelling command for that axis as in normal case or whether the axis travelling is carried out while synchronizing with the travelling of any other axis.

Simple synchronous operation is allowed in the automatic operation by tape command,manual data input,or memory command and in the manual operation such as manual continuous feed,manual handle feed,incremental feed,or manual reference point return.

23.11 Expanded Part Program Editing

The following editing is possible.

(1) Conversion

(a) Conversion of address
An address in the program can be converted to another address.
For example address X in the program can be converted to address Y

(b) Word conversion
A word in the program can be converted to another word. For example, a programmed M03 can be converted to M04

(2) Program copy
Program can be copied to make a new program

(a) Copy of all the program
(b) Copy of part of a program

(3) Program move
Part of a program can be moved to make a new program

(4) Program merge
New program can be created by merging two programs.

(5) Copy and move to the key-in buffer
Part of a program can be copied or moved to the key-in buffer.

(6) Sequence number automatic insertion
The sequence number, where a certain increment value is added to the sequence number of the previous block can be automatically inserted at the head of each block in preparation of programs by the part program editing.
The initial value of sequence number and a certain increment amount can be set.

23.12 Input/Output Simultaneous Operation

If a cycle operation is put into effect with DNC signal being ON, an external program is executed for operation and simultaneously the external program is stored in the memory of the control.

Also, a program in the memory of the control is executed and simultaneously output to an external device.

1) Input and machining simultaneous operation
   a) Search a head of a program to be executed.
   b) urn on DNC operating signal G0127 (DNCI).
   c) Turn on the input and machining simultaneous operation signal G0140 (STRD).
   d) Press the cycle start button.
   e) A program is executed for operation and input in the memory block by block.

2) Output and machining simultaneous operation
   a) Select a program to be executed.
   b) Turn on DNC operating signal G0127 (DNCI).
   c) Turn on the output and machining simultaneous operation signal G0140 (STWD).
   d) Press the cycle start button.
   e) A program is executed for operation and output to an external device block by block.

23.13 Operation When Converting the Machining Program Prepared by Graphic Conversation to an NC Format Program and Outputting Externally Via the Reader/puncher Interface

The machining program prepared by graphic conversation is stored in the tape memory areas as a special data type. Therefore, of course, the data cannot be displayed merely as it is in NC format and one part of the data cannot be changed.

However, after that machining program prepared by graphic conversation has been converted to an NC format program, it is possible to output it outside the NC via the NC reader/puncher interface.

After conversion, the NC format program outputted outside the NC will be of the form displayed by the NC program form by the NC operation command automatically prepared when executing that machining program.

Accordingly, it is normal that 1 NC operation command is converted as 1 block program. For example. one group of bar machining is not indicated by canned cycle G71.

In other words, in order to convert the canned cycle to a form can be resolved into each operation unit, after actual conversion, the outputted NC format program becomes longer in proportion to the machining operation.

When the machining program is converted to NC format and output to outside the NC, be sure to set the memory protect switch on the machine operator's panel to the lock state.

24. HIGH-SPEED MACHINING FUNCTION

The function converts the figure to be machined to a data group that can be high-speed pulse distribution by a macro compiler/macro executor,and calls and executes the data group as a machining cycle by the CNC command(G05 command)

(1) Number of controlled axes
4 axes maximum.(Simultaneous 4 axes control is possible.)

(2) Pulse distribution cycle
Set the pulse amount equivalent to this cycle as a macro variable for high-speed cycle machining by a macro compiler/macro executor.The pulse amount unit should be the least input increment.)

(3) Command format
Call and execute the high-speed cycle data by the following command.
G05 P10nnnL999;
P10001 - P10999;
First number of the machining cycle to call
L999;
Repetition count of machining cycle

The cycle data can be prepared for up to 999 cycle.Specify the machining cycle by the address "P" .Each cycle can be continuously called and executed by the connection data (in the header).Specify the repetition count of machining cycle by the address "L".The repetition count (in the header) can be specified for each cycle.The connection and repetition count are explained with an example.

Example)
Cycle 1 Connection data 2
Repetition count 1
Cycle 2 Connection data 3
Repetition count 3
Cycle 3 Connection data 0
Repetition count 1
G05P10001L2
In this condition, the cycles to be executed are:
1,2,2,2,3,1,2,2,2,3 cycles are continuously executed.

(4) Data configuration
The following data is converted to a high-speed cycle machining variable(#20000-#85535) by a macro compiler/macro executor.

Configuration of high-speed cycle machining data

25. POLIGONAL TURNING

Polygonal turning means machining a polygonal figure by rotating the workpiece and tool at a certain ratio.

(1) Rotation ratio of workpiece and tool
(2) Number of cutter

By changing these conditions,the machining figure can be changed to a square or hexagon.The machining time can be reduced as compared with polygonal figure machining using C and X axes of the polar coodinate.The machined figure however,is not exactly polygonal .Generally,polygonal turning is used for the heads square and/or hexagon bolts or hexagon nuts.

Tool rotation for polygonal turning is controlled by CNC controlled axis.

The rotary axis of tool is called Y axis in the following description.

The Y axis is controlled by G25.1 command,so that the rotation speeds of the workpiece mounted on the spindle (previously specified by S-command)and the tool become the specified ratio.

G251 P_Q_;

P ,Q : Rotation ratio of spindle and Y axis
Specify range :Integer 1 to 9 for both P and Q .
When Q is a positive value, Y axis makes positive rotation.
When Q is a negative value,Y axis makes negative ratation.

(Example)Rotation ratio of workpiece(spindle) to Y axis is 1:2,and the Y axis makes positive rotation.

G51.2P1Q2;

When simultaneous start is specified by G251,the one-rotation signal sent from the position coder set on the spindle is detected,and the Y axis rotation is controlled according to the rotation ratio(P:Q)while synchronizing with the spindle speed.Namely,the Y axis rotation is controlled so that the spindle and Y axis stand in a relation of P:Q.This relation will be maintained until the polygonal turning cancel command is executed(G50.2 or reset operation).The direction of Y axis rotation is determined by the sign of Q and not affected by that of the position coder rotation.

Synchronization of the spindle and Y axis is cancelled by the following command:

G250;

When G250 is specified,synchronization of the spindle and Y axis is cancelled and the Y axis stop.This synchronization is also cancelled in the following cases:

(i) Power off
(ii) Emergency stop
(iii) Servo alarm
(iv) Reset(external reset signal ERS, reset / rewind signal RRW,and RESET key on the CRT/MDI panel)
(v) When P/S alarm 217~221 is occured.

(Example of program)

G00X100 Z20 S1000M03;
Workpiece rotation speeed 1000rpm
G251P1Q2;
Tool rotation start (tool rotation speed 2000)rpm
G01X80.F10;
X axis infeed
G04X2;
G00X100;
X axis escape
G250;
Tool rotation stop
G05;
Spindle stop

Be specify G250 and G251 with single block.

26. SELF-DIAGNOSTIC FUNCTION

Various status can be displayed by the CRT/MDI panel.

(1) On-off signals input to the NC can be indicated on the CRT display.

(2) All on-off signals output from CNC can be sent in the unit of bit via MDI panel.

(3) Various parameter values such as acceleration/deceleration time constant,rapid traverse rate, etc. can be checked on the CRT display.

(4) Details of alarms which may occur accidentally can be classified and checked on the CRT display. These alarms comprise overheat, overtravel, servo alarm, tape check(horizontal and vertical directions) and program errors.

(5) The condition of servo system can be observed.

27. DATA INPUT / OUTPUT

27.1 I/O Device Interface

The following can be input/output via the reader/puncher interface.

(1) Part program registration
(2) Tool offset amount input
(3) Parameter input
(4) Part program punch
(5) Tool offset amount punch
(6) Parameter punch

In order to input a program into CNC memory or output the input program to an external data storage medium by using the MDI panel, a unit equipped with such FANUC reader/puncher interface as FANUC cassette/FANUC PPR/ portable tape reader or ASR33 interface can be connected. EIA RS244-A or ISO840 is employable as data codes.

These ISO/EIA codes are automatically identified from each other in case of data input. Either ISO codes or EIA codes is selectable by setting parameters in case of data output.

Since program input/output can be started by signals from the machine tool as well as by MDI panel operations, program can be loaded automatically.

Number of the reader/puncher interface channel

(1) Channel 1:Reader/puncher interface, ASR33 interface
(2) Channel 2:Reader/puncher interface

27.2 Input Devices

27.2.1 FANUC FLOPPY CASSETTE

Data can be stored in the floppy cassette and also data can be input to the CNC.
The interface is reader/puncher interface.

Refer to the Floppy Cassette Operator's Manual (B-66040E) for the operation of the floppy cassette.

27.2.2 FANUC PPR

An I/O device with paper tape reader, tape punch, and printer in one system.
Paper tape reader read speed:

150 ch./sec or more
Paper tape punch punching speed : 50 ch./sec
Printer characters per line:
40 ch. (dot impact method)
Printing speed : 1.2 line/sec

27.2.3 Portable Tape Reader

The portable tape reader is a carrying type paper tape reader.Used to load program, data, and parameter to the CNC.
Read speed :
300 ch/sec (60Hz), 250 ch/sec (50Hz)
Read method : Opto-electrical (LED)

27.2.4FANUC PROGRAM FILE Mate

The built-in hard disk enables data to be stored and it can be connected to the reader/puncher interface to input data to CNC.This hard disk has a large storage capacity of approximately 50,000m of paper tape data,so it can register maximum 1024 command programs.

It can be connected to the remote buffer to achieve high-speed transfer of maximum 86.4 kbps.
The hard disk is sealed to be continuously used under the factory environment.

27.2.5 FANUC FA Card

The FANUC FA Card ism a memory card which has nonvolatile solid state memories as memory element. By connecting the FA Card to the FANUC CNCs via the FA Card Adapter,part programs stored in the CNC can be transfered to and saved in the FA Card. Also, part programs written in the FA Card can be trancefered to the CNC as well.

- Compact, high density and easy-to-use
  20m to 160m paper tape equivalent data storage in 54mm ×85.6 mm ×5mm size
- Maintenance free
  Nonvolatile solid state memory requires no battery
- Stand alone type and Built in type FA Card Adapter are available

28. SAFETY FUNCTIONS

28.1 Emergency Stop

With the emergency stop, all commands stops, and the machine stops immediately.Connect the "emergency stop" signal both to the control unit and to the servo unit side.

When emergency stop is commanded, servo excitation is also reset, and servo ready signal will also turn off.After resetting the emergency stop, continue the operation after performing the reference point return operation.

28.2 Overtravel Functions

28.2.1 Overtravel

As this CNC's basic specifications cover the software limit detection function, it is generally unnecessary to provide the limit switch and dog function on the machine tool side for the purpose of overtravel detection.

In order to solve such a case as the stroke end position in the +Z direction changes each time the tailstock travels in keeping with the movement of a workpiece, an interface is provided to be able to receive a signal from the limit switch as an overtravel signal in the +Z direction only. In other words,the limit switch is mounted to the tailstock and its contact is connected to *+LZ terminal, so that the stroke and position can be detected correctly in keeping with the travel of the tailstock.

28.2.2 Stored stroke limit

The area outside an area preset by a parameter is defined as a forbidden area.If the machine tool enters the forbidden area, the axis motion is decelerated and stopped with overtravel alarm display.(stored stroke limit)

The distance from the reference return position is set to the parameter. Stored stroke limit function can be made to be ineffective by parameter setting.

28.2.3 External setting function of stored stroke limit

When changing a tool, this function rewrites the machine position (machine coordinate) of that time into parameter values (No. 700-702, 704-706) as the stored stroke limit by adapting its tip position to the end of the limit area and depressing the SETTING key.

Setting signals are provided for each axis and direction (+X, +Y, +Z, -X, -Y, -Z).

In addition, by turning the common LIMIT RELEASE button for all axes to the ON position, it is also possible to arrange that the stroke limit may not be checked.

28.2.4 Expanded Stored Stroke check

The area in which a tool may not travel (forbidden area) can be divided into the following two blocks for setting.The shaded portions above show the area in which no tool can move.

1st limit:

Set the boundary,using parameters.The outer side of boundary thus preset corresponds to the forbidden area.This is normally not changed once the machine tool builder sets this area.Therefore,always set the boundary to the maximum stroke of a machine.

2st limit:

Set the boundary,using parameters or program command.The inner or outer side of boundary preset corresponds to the tool entry prohibited area.The inner or outer side is determined by parameter. G22 prohibits the tool from entering the forbidden area of the 2nd limit.

28.3 Interlock

This function inhibits the axis movement of the machine tool. Whether the movement of all axes is inhibited or the movement of Z axis only is inhibited can be set by parameters. When all axes movement is inhibited, high speed interlock input can be used. Interlock also can be applied to each axis independently by parameter.

When this interlock signal is applied, the moving part of the machine tool stops after being decelerated. When this interlock signal is released, the machine tool starts moving again after being accelerated.

This interlock function is effective for both automatic and manual operation modes.

28.4 External Deceleration

Feed rate can be decelerated by an external deceleration signal from the machine side. A feed rate after deceleration can be set by parameter. External deceleration is prepared every axis and every direction.

When the tool is to be moved in the reverse direction, futile time may not be wasted since no external deceleration is applied.

Conditions to make this signal effective are set by parameters:

(1) Whether this signal is applied to rapid traverse only or to all feeds.

(2) Whether external deceleration in the + direction is made effective (each axis).

(3) Whether external deceleration in the – direction is made effective (each axis).

This function allows the maximum of valid strokes and keeps shock to the machine to a minimum, to stop at stroke end. This function is also useful when overtravel of the machine from the stroke end in, for example, ATC reference points , must be kept to a minimum.

29. STATUS OUTPUT

29.1 CNC Ready Signal

This signal is sent to the machine side when CNC power is on and control becomes possible. Sending of this signal will be stopped when CNC power is turned off.

29.2 Servo Ready Signal

This signal is sent to the machine side when the servo system becomes operatable. Axes necessary to be braked must be braked when this signal is not sent. When this signal is interrupted, NOT READY is displayed on CRT/MDI panel.

29.3 Alarm Signal

This signal is transmitted when the CNC comes under an alarm status.Alarm type signal is also sent out.

29.4 Distribution End Signal

This signal is sent out when pulse distribution of the M,S, or T functions has ended, so that they can be used after move of the commanded block ends.

29.5 Automatic Operation Signal

This signal is sent out when it is under automatic operation.

29.6 Automatic Operation Start Lamp Signal

This signal is sent out when automatic operation is being activated.

29.7 Feed Hold Signal

This signal is sent out when automatic operation is holded by feed hold.

29.8 Power Ready Signal

This signal is output when input power is supplied to the CNC, and output voltage of the stabilizer power circuit in the CNC has reached the standard value.If abnormality in power circuit is detected, this signal will no more be output. Stop power supply at once in this case.

29.9 Reset Signal

This signal is sent out to show that the CNC has been reset.

29.10 Battery Alarm Signal

This signal is output when the voltage of the battery for holding memory data during power interruption has become lower than the specified value. Since the above condition is usually indicated as the battery alarm display on the CRT/MDI panel, this signal is not required normally. If the detachable CRT/MDI is employed, inform an operator of the necessity of battery exchange by the lamp indication using this signal.

29.11 Clock function

This function displays the status indicating line on each CRT screen and the present time on the parameter setting screen. The time is corrected on the parameter setting screen.
It is aiso possible to know the current time. (year/month/hour/minute/second) through the PMC.

29.12 Position Switch Function

The position switch function outputs a signal to a specified controlled-axis when the machine coordinates of the controlled-axis are within the range specified by the corresponding parameter.

The parameter specifies an arbitrary controlled-aixs and the operating range (machine coordinates) within which the position switch signal is output.
Up to ten position switch signals can be output.

30. EXTERNAL DATA INPUT

30.1 External Tool Compensation

This function is provided to change the tool offset amount from the machine tool or other external devices. Input data are added to the offset amount having the offset number specified by the program when the tool offset amount is externally input. The offset amount may also be processed as input data by specifying an input signal.

When the machine tool provides an automatic measuring function for tools and workpieces, the offset amount can be changed to a correct value by inputting an error from the correct value into CNC by using this function.

If the tool offset amount is externally input when the offset number 0 is selected by a program, (that is, offset cancel), the work coordinate system shifts by the input data amount.

The external tool offset amount is 0 - ±7.999mm or ±0.7999 inch at a time.

30.2 External Message Function

For the message function applied by external data input signals, two functions are available. However, either the PMC-L or M is requried to use this function.

(1) External alarm message

"External alarm message" means a function which puts the CNC in the alarmed condition by sending alarm numbers externally and in addition, sends messages to the CNC externally and allows them to be displayed on the CNC's CRT screen. Four alarm numbers and four messages can be sent at the same time. The length of a message which can be sent against an alarm No. is within 15 characters. The alarm No. which can be sent will come from numbers between 0 and 999. However, when the CNC displays, it will add 1000. It is also executed using external data, to release the alarmed condition.

(2) External operator message

"External operator message" means a function which sends a message and a number for the operator to the CNC externally and allows to display the message on the CNC's CRT screen.

Only one message can be sent at one time with the message length within 127 characters. The number which can be sent is a number between 0 and 999, however, a message of up to No. 99 will be displayed with its number to which 2000 is added. As to the message of Number from 100 to 999, only characters will be displayed without its number.

30.3 External Key-Input Function

"External Key-Input" means a function which can control the CNC by informing CNC of key input data from the PMC as if an operator entered it into the CRT/MDI unit. For instance, it becomes possible to edit programs (the searching for a program No.) by PMC.

30.4 External Data Input

The following data can be entered,using external data input.

(1) External tool offset
(2) External message function
(3) External key input
(4) External program No. serach

This function inputs any one of the program number 1 to 9999 to CNC from the outside,such as the machine side and searchs the program from the memory

For a machine provided with a function of loading several types of workpieces

automatically, this function can be used to search a program automatically according to workpiece for execution.

(5) External work coodinate system shift (Only M series)

The external work coodinate system shift is a function of shifting the work coodinate system according to the external shift amount.

This shift amount for each axis is all added to the work coodinate system for use. This shift amount is not lost even when power is turned off.

This shift amount is not integrated incrementally but the input shift amount corresponds to a new shift amount. The shift amount which can be input is 0 to ±0.7999 inch.

30.5 External Machine Origin Shift

The machine coordinate system is compensated by offset amount given from outside. This offset amount always take absolute value; never an increment value.

The offset amount range is:

0 ± 9999

in detection unit.

When offset amount is input, the actual machine move distance is the difference between the previous offset amount and current offset amount.

This function is used t compensate the machine coordinate system error caused by mechanical deformation.

31. EXTERNAL WORK NUMBER SEARCH

By specifying work numbers of 01 - 15 externally (from the machine side,etc.), a program corresponding to the work number can be selected. The work number equals the program number. For example, when work number 12 is specified, program O0012 is selected. (The first 2 digits of the program number is 00.)

32. MACHINE INTERFACE

Kinds of I/O card

	Inputs	Outputs (Transistor output)	Remarks
I/O C1	40	40	
I/O C2	80	56	
I/O C3	104	72	
Additional I/O B1	104	72	Available with PMC-M

(Note) Either I/O C1, C2 or C3 can be selected.
When PMC-M is provided, either C1, C2 or C3 and additional I/O B1 can be employed

Number of Inputs/Outputs

Input (total)	Output (total)	Remarks
40	40	
80	56	With PMC-L/PMC-M
104	72	
144	112	With PMC-M
184	128	
208	144	

33. PROGRAMMABLE MACHINE CONTROLLER(PMC)

Magnetics sequence circuit can be incorporated in the CNC.
The following two types of PMC is available.

	Max.No. of inputs	Max.No. of outputs	No. of step	Time of processing	Remarks
PMC-Model L (PMC-L)	104	72	3000 steps	15μ/step	High-speed M, S and T interface is available.
PMC-Model M (PMC-M)	208	144	3000/5000steps	2μ/step	High-speed M, S and T interface and no dividing method is available.

Window Function

In this function, PMC can request CNC to transfer data to the PMC side.

The following kinds of data are transferred from the CNC.

(1) Present position

(2) Machine position

(3) Skip position

34. CONTROL UNIT

34.1 Cabinet

W:
Control unit A: 250
Control unit B: 450
Weight : About16kg
Coating color: P-LT;5GY 3.5/0.5(Dark grey)

34.2 Power

34.2.1 Input power voltage

200VAC $^{+10}_{-15}$ %,50/60Hz ± 1Hz,1 phase

or

220VAC $^{+10}_{-15}$ %,60Hz ± 1Hz,1 phase

34.2.2 Input unit

Power ON/OFF control is made easy by the input unit setting to machine side.Input power of unit is as following.

200VAC $^{+10}_{-15}$ %,50/60Hz ± 1Hz

or

220VAC $^{+10}_{-15}$ %,60Hz ± 1Hz

34.2.3 Multi-tap transformer

Multi-tap transformer may be necessary for the control unit according to input power voltage. Multi-tap transformer is not necessary when input power is:

200VAC $^{+10}_{-15}$ %,50/60Hz ± 1Hz,1ϕ

or

200VAC $^{+10}_{-15}$ %,60Hz ± 1Hz,1ϕ

When multi-tap transformer is equipped with, allowable input power becomes:
200/ 220/ 230/ 240/ 380/ 415/ 440/ 460/ 480/

550VAC $^{+10}_{-15}$ %,50/60Hz ± 1Hz,1phase

The multi-tap transformer must be mounted on the machine side.

34.3 Battery Unit

Mount the battery unit to the machine side where it is replaced easily.

34.4 Environmental Requirements

(1) Ambient temperature
0°C - 45°C when operating
-20°C - 60°C when stored or delivering
(2) Change in temperature Max. 1.1°C/min
(3) Humidity 75% or less (relative humidity)
generally Max. 95% for a short time
(4) Vibration 0.5G or less when operating
(5) Circumstances
When using in places with thick dust, cutting oil, or organic solvent, consult us.

34.5 Weight

Control unit (Panel mount type):About 16kg
CRT/MDI :About 5kg
CRT/MDI (Detachable type) :About 11kg
Weight may differ according to system configuration

35. CABLES AND CONNECTORS

Cables and connectors are offered for connection between units.

36. SERVO SYSTEM (Digital Servo)

36.1 Position Detection System

Pulse coder

36.2 Position Detector .

Incremental pulse coder .
Absolute pulse coder

Position is detected by the feed-back signal from the pulse coder. The following move distance/rotation of the pulse coder is offered.

Table36.2 Kind of pulse coder

Ball screw unit	Pulse coder	Move distance/rotation of pulse coder
Metric	2000p /rev	2,3,4,6,8mm
	2500p/rev	5,10mm
	3000p /rev	3,6,12mm
Inch	2000p /rev	0.1,0.15,0.2,0.3,0.4inch
	2500p /rev	0.25 ,0.5inch
	3000p /rev	0.15,0.3,0.6inch

36.3 Servo Motor

FANUC AC SERVO MOTOR MODEL 4-0, 3-0, 2-0, 1-0, 0, 5, 10, 20, 20M, 30, 30R

36.4 Velocity Control Unit

Various velocity control units are offered according to types of AC servo motors.

36.5 Regenerative Electric Discharge Unit

Various regenerative electric discharge unit are offered according to type of AC servo motor.

36.6 Servo Power Transformer

Various power transformers are offered according to types of AC servo motors.

37. POSITION CODER

A position coder is connected to the spindle and generates square wave voltage signal the frequency of which is proportional to the spindle speed. This synchronizes the feed with the spindle speed.

(1) Position coder A

(a) Number of square waves per revolution

Channel	Number of square waves
1ch	1024p /rev (A,Ā,B,B̄ phases)
2ch	1p /rev (A,Āphase)

(b) Maximum response revolutions 4,000 rpm

(c) Range of operating temperature0° - 50°C

(2) Position coder B

(a) Number of square waves per revolution

Channel	Number of square waves
1ch	1024p /rev (A,Ā,B,B̄phases)
2ch	1p /rev (A,Āphase)

(b) Maximum response revolutions 6,000 rpm

(c) Range of operating temperature 0° - 50°C

38. FUNCTION FOR 4-AXIS LATHES(0-TTC)

38.1 General

0-TTC can independently operate two tool posts and control a lathe which simultaneously cuts and machines on the two tool posts.
It can be applied on two tool posts and the lathe which machines two workpieces with two spindles on them.

Single spindle and two tool posts

Twin spindles and two tool posts

(1) Application to lathes with twin spindles and two tool posts
The system can be applied to turning tasks in which a single workpiece attached to a single spindle is machined using two tool posts simultaneously. For example,while one tool post performs machining of the outer dimeter, it is possible to machine the inner diameter with the other tool post. The result is a sharp savings in machining time.

(2) Application to lathes with twin spindles and two tool posts
The system can also be applied to turning tasks where two workpieces attached to twin separate spindles are simultaneously machined using the two tool posts. Because machining can thereby be performed with one lathe with equal efficiency to two conventional lathes, a dramatic increase in productivity can be achieved.

(3) Simultaneously,independent control of two tool posts
Using the tool post selection key on the NC operator's panel, the operator selects the programs for each of the two tool posts from among the programs registered in the memory. When the start signal is issued,the system simultaneously and independently executes the selected two programs, controlling the X1,Z1 axes and X2,Z2 axes respectively.
Only one operator's panel is provided for the two tool posts. Panel operation and display functions for each tool post are determined by the tool post selection key provided.

(4) Tool post waiting function
The movement of each of the two tool posts can be programmed independently, as in the case of an ordinary two-axis lathe. M codes control the first and second tool posts to wait for each other during machining. Or only one tool post to machine.

38.2 Balance Cut(G68,G69)

G68 : Balance cut mode
G69 : Balance cut cancel

When machining the workpieces shown in the figure above in a four-axis lathe, machining the workpiece with cutters applied to its sides simultaneously prevents sagging of the workpiece, which would be caused by machining from one side only. This results in more accurate machinng. If,however,the cutters do not move sysnchronously, the workpiece will chatter, resulting in no correct machining. The use of this function permits easy synchronization.

38.3 Interference Check Function

A 4-axis lathe is normally provided with two tool posts. When a blank is concurrently machined by using these two tool posts,it is possible that a part of each tool post approaches each other very closely. If two tool posts should make contact with each other due to a machining program failure or other setting failures, a serious accident like the damage of tools or the breakage of the machne tool itself, may happen.

39. MACHINE OPERATOR'S PANEL

Standard machine operator's panel	External dimensions	No. of keys	External view	Remarks
Standard machine operator's panel for small 9" monochrome CRT/MDI	400x180x150 (W)(H)(D)	46 keys 46LED(red)	Appendix4 Fig. -14	Override switch,Emergency stop button, Program protect key are provided
Standard machine operator's panel for small 9" color CRT/MDI	400x180x150 (W)(H)(D)	46 keys 46LED(red)	Appendix4 Fig. -15	Override switch,Emergency stop button, Program protect key are provided
Standard machine operator's panel A for full key 9" CRT/MDI	520x180x150 (W)(H)(D)	53key 53LED(red)	Appendix4 Fig. -16	Override switch, Emergency stop Program protect key, Manual pulse generator are provided
Standard machine operator's panel B for full key 9" CRT/MDI	520x180x150 (W)(H)(D)	53key 53LED(red)	Appendix4 Fig. -17	Program protect key is provided A switch can be equipped on the blind plate by the machine tool builder.

The standard machine operator's panel can work using DI 22 points and DO 8 points only. However, PMC-L or PMC-M is required.

40. REMOTE BUFFER

40.1 REMOTE BUFFER

The remote buffer is option and is used to allow a large number of data to be continuously supplied to the CNC at high speed by connecting it to the host computer or I/O device through a serial interface.

The following can be performed by the remote buffer.

1)It is used to perform DNC operation at high speed and with high reliability by performing on-line connection to the host computer.

2)It is used to download the NC program and parameters from the host computer.

3)It is used to perform DNC operation and download various kinds of data by connecting to the I/O device.The following I/O devices can be connected.

① FANUC PPR

② FANUC CASSETTE B1/B2(Bubble cassette)

③ FANUC CASSETTE F3(Floppy cassette)

④ FANUC PROGRAM FILE

⑤ FANUC PROGRAM FILE Mate

4)There are following 2 types as protcol.

Protcol A	Thehand shake way,that it is repeated transmission and receptopn between them
Protcol B	The way that the communication between them is controlled by control code output from remote buffer

40.2 High-speed Remote biffer A

High-speed remote buffer A for FANUC 0 series is started by a block command "G05;",thus allowing high-speed processing,using the subsequent travel data for the specified axes.

(1) Data format

The tool moving distance for 4/8 msec shall be expressed in binary data.

Travel sequence ----------→

1byte				~			
1st axis		2nd axis		~	N axis		check byte

(a) Designate the number of command axes N,using parameter No.597.

(b) One block shall be (2*N + 1)bytes.

(c) Command the moving distance of each axis in 2 bytes.However, be sure to set LSB of each byte to 0 . Using the remaining 14 bits,command the moving distance at intervals of 8 msec in the following unit. Command the moving distance in the negative direction , using the complement of two.

(Example) When the moving distance in 8msec is 700 μ/8 msec:

15							8	7							0
0	0	0	0	1	0	1	*	0	1	1	1	1	0	0	*
0															

"*" should be always 0.

(d) The check byte is the sum of (2*N) bytes data of 8 bits,but the overflown value exceeding the 8 bit value is discarded.

(2) High-speed remote buffer A mode

For high-speed remote buffer A, its mode should be designated.

High-speed remote buffer A mode ON: "G05;"

High-speed remote buffer A mode OFF: the moving distance is 0 for all the N axes.

(Note) After commanding "G05;" block with a nomal NC command format,command the moving distance in the aforementioned format. Then, command 0 for the moving distances of all axes. Therefore,the high-speed remote buffer is placed in the OFF mode;hereafter,command the moving distance again in a normal NC command format.

(3)It is remarkable speed of 15m/min under the conditions that the command program

containing continuous blocks with travel increment of 1mm per 4msec for 3-axis simultaneous control and transfer speed of 38.4Kbaud.

(4)Requirements

(a) Only the linear incremental command is used.

(b) If "G05;" is commanded in the current compensation mode, an alarm will be produced.

(c) The feedrate override is ignored.

(d) No storage is possible.

(e) The feed hold is operational.

(f) Interlock is operational.

(g) The clamp for maximum speed cutting feed is not operational.

(5) Transfer speed

The minimum transfer speed required is obtained regarding 11 bits/byte as follows:

[〖(2×N+1)byte×11 / 8m·sec×1000 〗 baudrate]

If the data is transferred at a lower rate than the above, the desired speed will not be produced.

40.3 High-speed remote buffer B

The use of NC language prepared by an automatic programming unit,for example, permits high-speed machining.

(1) Data format

Designate the high-speed machining section,using the following commands.

Command	Contents
G05P01;	High-speed machining start
G05P00;	High-speed machining end

Example)

```
O1234;
   :
G05P01; ......  High-speed machining start
X__Y__Z__;
   :
G05P00; .......  High-speed machining end
   :
M02;
```

(2) Command contents

The following can be commanded in the high-speed machining section.

Command address	Contents
X	X-axis moving distance
Y	Y-axis moving distance
Z	Z-axis moving distance
F	Cutting feedrate

No other than the above can be commanded.

(3) It is remarkable speed of 12m/min under the conditions that the command program containing continuous blocks with travel increment of 1mm for 3-axis simultaneous control and transfer speed of 38.4Kbaud.

II. CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION (0-TC, 0-TF, 0-MC, 0-MF, 0-Mate MF)

1. GENERAL

The conversational automatic function is prepared for 2-axis lathe by 0-TC / TF and for machining center by 0-MC / MF / Mate MF.
Machining programs can be prepared easily, just by following the illustrative drawings and operating guides. Program checking and correction / changing can also be carried out with remarkable ease.

2. GRAPHIC CONVERSATION FOR 2-AXES LATHE (0-TF)

(1) Illustrative drawings

Illustrative drawings are provided for each type of machining process. During input, each portion of the illustration flashes in succession to indicate where data is being asked for. This feature greatly sinmplifies data input operations.

(2) Operating guidance

Each impending operation is displayed on the screen. So even inexperienced operators can master all operations with no special training.

(3) Profile description check

Once data input is complete, the rasulting machining profile can be displayed on the screen as programmed. This feature makes it possible to detect input errors immediately for quick correction.

(4) Program copy

A copy function is provided for copying previously prepared machining programs. This feature greatly facilitates preparation of similar programs.

2.1 Menu Screen

A complete selection of menus has been prepared to accommodate all programming needs - everything from rod machining, tracing necking, facing and threading to grooving and drilling. This enables the inexperienced operator to proceed with ,effortless ease, and at the same time permits the experienced operator to make maximum use of his machining knowhow

```
CONVERSATIONAL PROGRAMMING PROGRAM -0010
*** PROCES (01) ***
MACHINING       P=[redacted]      1=CYLINDER
PATTERN         A=OUTER END       2=PTTRN RPT
START POINT     X=  120.000       3=RESIDUAL
                Z=   10.000       4=END FACNG
ROUGHNESS      SR=       1 ▾      5=THREADING
RGH TOOL NO.   RT=     102        6=GROOVING
  OUTPUT T     RO=     202        7=NECKING
  CUT-SPD.     RC=     700        8=CENTERDRL
  FEED/REV.    RF=  60.0000       9=DRILLING
  SPINDLE      RS=   NORMAL      10=TAPPING
                                 11=SINGLE
                                 12=MEASURING
DATA→INPUT       (CONTINUE       13=SUB CALL
(PRE-PR)(NX-PRC)(NEW PR)( PLOT )(GUIDE )
```

```
CONVERSATIONAL PROGRAMMING PROGRAM -0010
*** PROCES (01) ***
MACHINING       P= CYLINDER       1=OUTER END
PATTERN         A=[redacted]      2=OUTER MID
START POINT     X=  120.000       3=INNER END
                Z=   10.000       4=INNER MID
ROUGHNESS      SR=       1 ▾      5=FACE END
RGH TOOL NO.   RT=     102        6=FACE MID
  OUTPUT T     RO=     202
  CUT-SPD.     RC=     700
  FEED/REV.    RF=  60.0000
  SPINDLE      RS=   NORMAL

DATA→INPUT       (CONTINUE
(PRE-PR)(NX-PRC)(NEW PR)( PLOT )(GUIDE )
```

```
CONVERSATIONAL PROGRAMMING PROGRAM -0010
*** PROCES (01) ***
CUT TYPE (1)   P1=[redacted]      1=CONST. AMNT
         (2)   P2=SNGL EDGE       2=CONST. DPTH

NO. OF THREAD NT=        1
SPARK OUT     SO=        2
THRD HEIGHT    H=    2.000
CUT DEPTH     MD=    1.000
SPINDLE GEAR   P=    AUTO.
COOLANT        M=       ON

DATA→INPUT
(PRE-PR)(NX-PRC)(NEW PR)( PLOT )(GUIDE )
```

2.2 Arbitrary Profile Machining Programs

Arbitrary profiles can be specified easily using Symbolic input and the automatic intersection computation function. The specified profile is graphically displayed immediately, thus enabling fast and easy checking.

```
CONVERSATIONAL PROGRAMMING PROGRAM -0010
*** PROCES (03) FIGURE DATA ***
  CYLINDER OUTER END
 START   X=    0.000  Z=    0.000
 VERTCL  X=   20.000  Z=    0.000
 HORIZN  X=   20.000  Z=   15.000
[redacted]TAPER  X=   50.000  Z=   30.000

1=HORIZN  2=VERTCL  3=TAPER   4=ARC
5=CHAMFR  6=CONR-R  7=TANGNT
FIG. TYPE=[redacted]→INPUT
(CANCEL)(MCHN-E)(PRCS-E)( PLOT )(GUIDE )
```

2.3 Set up Guidance Display

Set up operations that have traditionally been difficult to perform without sufficient experience - e.g, measurement of tool offset values, designation of the zero point, etc. - can be performed automatically just by following the instructions as they appear on the screen.

```
*** TOOL PREPARATION *** [redacted]

3. BRING THE TOOL EDGE IN CONTACT WITH
   A WORK-END SURFACE IN MANUAL MODE

   AFTER THIS OPERATION,
   PUSH SOFTKEY [ END ]

(PRE-PG)(NXT-PG)( END  )(RETURN)(      )
```

2.4 Automatic Selection of Tools, Cutting Conditions and Pre-tools

The ideal tools and cutting conditions for each machining process can be automatically set during program preparation. Automatic programming for pre-machining , as required for drilling, is also possible.

(1) Tool file

Up to 40 tools in 17 types - tools for general-purpose lathe, tools for thread cutting, etc.- can be registered. The most suitable tools for each type of machinig can be selected automatically.

```
*** TOOL DATA (1) ***
TOOL TYPE        OUTTER   ( LEFT )
T-CODE           01
OFFSET NO.       01
REVOLUTION-D     NORMAL
GEOMETRY OFS  X=      0.000 Z=      0.000
WEAR OFFSET   X=      0.000 Z=      0.000
NOSE-RADIUS  RN=      5.000
CUTTING EDGE AC=    130
NOSE ANGLE   AN=     30
IMGNRY NOSE  TD=      3
TL MATERIAL  TM= HI-SPD
TOOL LIFE    TT=    150
SPARE TL NO.        0
OFFSET NO.                  →INPUT
(      )(      )(NXT-PG)(INPT-E)( LIST )
```

(2) Cutting condition file

For all tools registered in the tool file, cutting speed and feedrate can be set to each of 8 different types of machining materials. Based on the data determined here, the ideal cutting conditions for each machining can be set automatically.

```
*** MACHINING CONDITION (GENERAL) ***
                                ROUGHING
HI-SPEED
MATERIAL    FEED/REVCUT SPD.    CUT DPTH
            (MM/REV.) (M/MIN.)   (MM)
1. FC25                  50.0    2.000
2. FCD45    0.3000       60.0    1.500
3. SS41     0.0500      120.0    3.000
4. S45C     0.4000       20.0    1.200
5. SKD11    0.2500       65.0    2.000
6. SUS304   0.2000      130.0    1.200
7. AL       0.8000      110.0    3.000
8.          0.0000        0.0    0.000
  FC25                  →INPUT
(      )(      )(      )(RETURN)(      )
```

(3) Coefficient data file

With general-purpose tools, the ideal cutting speed for the given feedrate can be automatically set. Simiarly, for drill tools the ideal feedrate can be set automatically for each nominal diameter.

```
*** MACHINING CONDITION (COEFICGRV. *

GRV WIDTH                4    10    20
  (MM)
HI-SPD     0.500 0.700 0.900 1.000 1.100
CARBID     0.700 0.800 0.900 1.000 1.100
SPCIAL     0.900 0.950 1.000 1.050 1.100

                                (FEED/REV.)
GRV WIDTH                   →INPUT
(      )(      )(      )(RETURN)(      )
```

(4) Pre-tool list

For drilling tasks, it is possible to register the tools required for pre-machining prior to final machining. Based on the pre-machinig data determined here, it is possible to automatically prepare the pre-machining program needed for drilling.

```
*** PRE-TOOL LIST ***          NO. 01

            TOOL NO.   TL NAME  NOMINAL
DLAST TL                TAP      12.800

PRE-TOOL 1  701        CNTRDRL
         2  601        DRILL    11.776
         3
         4
         5

LAST TL                     →INPUT
(      )(      )(      )(RETURN)(      )
```

2.5 Dynamic Graphic Display (for lathes)

Prepared machining programs can be graphically displayed on the screen for close checking prior to actual machining. Two display methods are offered.

(1) Tool path display

The tool nose movement is described on the screen as a thin line.

(2) Animated simulation

The material profile, chuck and tailstock are all displayed on the screen in their true dimensions. An animated simulation illustrates how the material will be cut progressively according to the programmed tool movement.

3. GRAPHIC CONVERSATION FOR MACHINING CENTER (0-Mate MF /0-MF)

(1) Operating guidance

Each required operation is displayed in order on the screen. So even inexperienced operators can master all operations with no special training.

(2) Process list display

After a machining program has been prepared through conversational input, a list of its required processes , including tools to be used, can be displayed. This feature enables fast, easy verification of the machining order, tools, etc.

3.1Keytouch

(1) Automatic pre-tool selection

During drilling program preparation, once the final hole depth and tool number have been input, the system automatically prepares the preliminary processes that are required prior to the drilling desired.

(2) Automatic setting of cutting conditions

Once the operator inputs the numbers of the tools required for the machining at hand, the system automatically determines the most suitable spindle speed and feedrate.

3.2 Menu Screen

A complete selection of menus has been prepared to accommodate all programming needs - everything from drilling, facing, side cutting, grooving and pocketing to auxiliary functions and NC language menus. This enables the inexperienced operator to proceed with effortless ease, and at the same time permits the experienced operator to make maximum use of the machining knowhow.

3.3 Optimum Tool Order

The machining order is automatically adjusted to carry out the programmed machining task using the least number of tool changes from start to finish. This eliminates the need for the programmer to give consideration to tool order during program input.

```
PROGRAM                                  00002 N0050
START OF PROCESS                                 G202

  CHANGE OF TOOL OPERATION SEQUENCE IS
 AVAILABLE IN A PROCESS

  A PROCESS IS MADE UP OF THE BLOCKS
 BETWEEN START OF PROCESS AND END OF
 PROCESS

INSRT KEY DEPRESS
( MENU )(        )(        )(        )(        )
```

3.4 Multi-piece Machining

To machine idential products in multiple quantities at the same time, the operator need prepare the program only for one piece. The remaining pieces can be programmed automatically, inclusive of the optimum tool order.

```
PROGRAM                                  00002 N0050
MULTI-PIECE                                      G206
COORDINATE      *B
NUM. OF PIECE   *P        10
NUMBER(X)       *H         3
X PITCH          I      120.
Y PITCH          J      100.

DATA→INPUT→AT LAST INSRT
( MENU )(        )(        )(        )(        )
```

3.5 Arbitrary Profile Machining Programs

Arbitrary profiles can be specified easily using Symbolic input and the automatic intersection computation function. The specified profile appears graphically on the screen immediatery, thus enabling fast and easy checking and revisions. It is also possible to perform contour side cutting, contour grooving and contour pocketing of specified profiles.

```
PROGRAM                                  00002 N0050
CONTOUR FORM   (SELECT CROSSING)
CROSSING NMB.        _P

                 X    10.000 Y    70.000

SELECT CROSSING 1 OR 2

DATA→INPUT→AT LAST INSRT
( ←→ )( ↑↓ )( ↗↙ )( ○  )(CANCEL)
```

(1) Contour side cutting

Sides of arbitrary profiles which are formed from lines and circles can be programmed through conversational input.

(2) Contour grooving

Grooves of arbitrary profiles where the center line is formed from lines and circles, as shown in the illustration, can be programmed through conversational input.

(3) Contour pocketing

Pocketing of arbitrary profiles having islands such as shown in the illustration can be carried out automatically simply by preparing, through conversational input, the exterior walls of pockets and islands formed from lines and circles. Up to 6 islands can be designated, each with a different height if necessary.

3.6 Automatic Selection of Cutting Conditions and Pre-tools

The ideal cutting conditions for each machining can be automatically set during program preparation. Automatic selection for pre-machining, as required for drilling, is also possible.

(1) Tool file

Up to 45 tools in 9 types - drills, taps, end mills, etc. - can be registered.

```
TOOL FILE                    :03 00002 N0002
TOOL NO.              7          8          9
T-CODE                7          8          9
TOOL NAME   _END-ML     END-ML     CHAMFR
TL MATERIAL   HI-SPD     HI-SPD     HI-SPD
NOMINAL DIA.  12. 000    10. 000    20. 000
PITCH/S-DIA.   0. 000     0. 000     4. 000
TOOTH/C-DEP.          3          3          0
ANGLE                 0          0         90
OFST NO. (H)          7          8          9
OFST NO. (D)         17         18         19
DEPTH OFST.    0. 000     0. 000     0. 000
OTHER TOOLS. DEPRESS THE RIGHT SOFT KEY
NUM  -

(DRILL )(CENTER)( TAP  )(REAMER)(BORE  )
```

(2) Cutting condition file

For all tools registered in the tool file, cutting speed and feed rate can be set to each of 8 different types of machining materials.

```
MACHINING COND. (MILL)      :01 00002 N0002
HI-SPDEND-ML  (S. DIA   0MM0  ROUGHING
 MATERIAL  FEED F-XY  FEED F-Z  SPEED V
           (MM/TOTH) (MM/TOTH) (M/MIN.)
1. FC25  _  0. 200     0. 100    80. 000
2. FCD45    0. 100     0. 050    90. 000
3. SS41     0. 500     0. 300   120. 000
4. S45C     0. 100     0. 080    80. 000
5. SKD11    0. 300     0. 100    60. 000
6. SUS304   0. 200     0. 100    50. 000
7. AL       0. 800     0. 400   110. 000
8.          0. 000     0. 000     0. 000

NUM  -

(FACE-M)(END-ML)(      )(      )(      )
```

(3) Coefficient data file

To accommodate cutting conditions which vary according to tool diameter, it is possible to apply a coefficient for each set of cutting conditions. Used together with the tool file in(1) and the cutting condition file in (2), it is possible to automatically set the ideal cutting conditions for each machining.

```
MACHINING COND. (COEFF)     :01 00002 N0002

DRILL-DIA.   _  2      6     10     20
   (MM)
 HI-SPD   0. 500 0. 800 1. 000 1. 200 1. 500
 CARBID   0. 800 0. 900 1. 000 1. 100 1. 300
 SPCIAL   0. 900 1. 000 1. 100 1. 200 1. 300

TAP-DIA.         2      3      5      8
   (MM)
 HI-SPD   0. 500 0. 700 0. 900 1. 000 1. 200
 CARBID   0. 400 0. 600 0. 800 1. 000 1. 200
 SPCIAL   0. 500 0. 700 0. 900 1. 000 1. 200
NUM. =

(       )(       )(       )(       )(       )
```

(4) Pre-tool list

The pre-tool list can be used to register up to 20 pairs of pre-tools, finishing tools and auxiliary tools in max. 4 tools. Based on the pre-machining data deteamined here, it is possible to automatically prepare the pre-machining program needed for drilling.

```
PRE-TOOL LIST                :01 00002 N0002
           TOOL NO.  CYCL TOOL NAM NOM. DIA
LAST TL.  1 _ 3       6  TAP          3. 000
          2   1       3  DRILL        3. 000
PRE TOOL  3   2       2  CENTER       2. 000
          4   0       0               0. 000
          5   0       0               0. 000
1=DRILLING              7=COUNTER TAPPING
2=DRILLING (DWELL)      8=BORING
3=DEEP HOLE DRILING     9=RAPID BORING
4=RAPID DEEP HOLE      10=BACK BORING
5=HOLLOW DRILLING      11=BORING (DWELL)
6=TAPPING              12=FINE BORING
NUM. =

(       )(       )(       )(       )(       )
```

3.7 Dynamic Graphic Display (For machining center)

Prepared machining programs can be graphically displayed on the screen for visual checking prior to actual machining. Two display methods are offered.

(1) Tool path description

The tool path is described on the screen as a line. This feature enables a close check of the prepared program.

① Because description proceeds at high speed, the program check can be completed quickly.

② An automatic scaling feature is provided which permits the operator to maintain a suitable magnification and keep the focus of attention in the middle of the screen at all times.

③ During screen monitoring, the center of partial enlargements and the rate of magnification can be commanded when needed. This feature enables simple enlargement of sections demanding closer attention.

④ In addition to 2-dimensional drawings, it is also possible to describe isometric projections or bi-plane drawings simultaneously.

⑤ The precise current position of the tool can be displayed on tool path description. This enables the operator to know the exact machining status at any time.

(2) Machining profile description

The changing profile of the workpiece is simulated according to the movements of the tool as programmed. Because the profile is shown in 3 dimensions, the operator can check the machining program in detail with great ease.

① The coordinate axis and projection angle can be changed at random.

② The tool can be attached not only in parallel to the Z-axis, but alternately in parallel with either the X-axis or Y-axis.

③ In addition to 3-dimensional drawings, it is also possible to describe 2-dimensional drawings or tri-plane drawings simultanenously.

APPENDIX

APPENDIX 1 TABLE OF RANGE OF COMMAND VALUE

(1) T series

Linear axis (in case of metric thread for feed screw and metric input)

	Increment system	
	Standard	0.1μ
Least inpt increment	0.001 mm	0.0001 mm
Least command increment	X:0.0005 mm Z:0.001 mm	X:0.00005 mm Z:0.0001 mm
Max. programmable dimension	±99999.999 mm	±9999.9999 mm
Max.rapid traverse *1	100000 mm/min	24000 mm/min
Feedrate range *1	1~100000 mm/min 0.0001~500.0000 mm/rev	1~12000mm/min 0.0001~500.0000mm/rev
Step feed	0.001, 0.01, 0.1, 1 mm/step	0.001, 0.01, 0.1, 1 mm/step
Tool compensation	0~ ±999.999 mm	0~ ±999.999 mm
Backlash compensation	0~ ±0.255 mm	0~ ±0.255 mm
Dwell time	0~ ±99999.999 sec	0~ ±99999.999 sec

Linear axis (in case of metric thread for feed screw and inch input)

	Increment system	
	Standard	0.0000 inch
Least input increment	0.001 inch	0.00001 inch
Least command increment	X:0.000005 inch Z:0.0001 inch	X:0.000005 inch Z:0.00001 inch
Max. programmable dimension	±9999.9999 inch	±393.70078 inch
Max. rapid traverse *1	100000 mm/min	24000 mm/min
Feedrate range *1	1~100000 inch/min 0.0001~500.0000 inch/rev	1~12000 inch/min 0.0001~500.0000inch/rev
Step feed	0.0001, 0.001, 0.01, 0.1inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation	0~ ±99.9999 inch	0~ ±99.9999 inch
Backlash compensation	0~ ±0.255 mm	0~ ±0.255 mm
Dwell time	0~99999.999 sec	0~99999.999 sec

Linear axis (in case of inch thread for feed screw and inch input)

	Increment system	
	Standard	0.000001 inch
Least input increment	0.0001 inch	0.00001 inch
Least command increment	X:0.00005 inch Z:0.0001 inch	X:0.000005 inch Z:0.00001 inch
Max. programmable dimension	±9999.9999 inch	±999.99999
Max. rapid traverse *1	4000 inch/min	960 inch/min
Feedrate range *1	0.01~4000.00 inch/min 0.000001~9.999999 inch/rev	0.01~480 inch/min 0.000001~9.999999inch/rev
Step feed	0.0001, 0.001, 0.01, 0.1inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation	0~ ±99.9999 inch	0~ ±99.9999 inch
Backlash compensation	0~ ±0.0255 inch	0~ ±0.0255 inch
Dwell time	0~99999.999 sec	0~99999.999 sec

Linear axis (in case of inch thread for feed screw and metric input)

	Increment system	
	Standard	0.1μ
Least input increment	0.001 mm	0.0001 mm
Least command increment	X:0.00005 inch Z:0.0001 inch	X:0.000005 inch Z:0.00001 inch
Max. programmable dimension	±99999.999 mm	±9999.9999
Max. rapid traverse *1	4000 inch/min	960 inch/min
Feedrate range *1	1~100000 mm/min 0.0001~500.0000 mm/rev	1~12000 mm/min 0.0001~500.0000mm/rev
Step feed	0.001, 0.01, 0.1, 1 mm/step	0.0001, 0.001, 0.01, 0.1 mm/step
Tool compensation	0~ ±999.999 mm	0~ ±999.9999 mm
Backlash compensation	0~ ±0.0255 inch	0~ ±0.0255 inch
Dwell time	0~99999.999 sec	0~9999.9999 sec

Rotation axis

	Increment system	
	Standard	0.0001 deg
Least input increment	0.001 deg	0.0001 deg
Least command increment	0.001 deg	0.0001 deg
Max. programmable dimension	±99999.999 deg	±9999.9999 deg
Max. rapid traverse *1	100000 deg/min	24000 deg/min
Feedrate range *1	1～100000 deg/min	1～12000 deg/min
Step feed	0.001, 0.01, 0.1, 1 deg/step	0.0001, 0.001, 0.01, 0.1 deg/step
Tool compensation	0～±999.999 mm	0～±999.9999 mm
Backlash compensation	0～±0.255 deg	0～±0.255 deg

*1 The feed rate range shown above are limitations depending on CNC interpolation capacity. When regarded as a whole system, limitations depending on servo system must also be considered.

(2) M series

Linear axis (in case of metric thread for feed screw and metric input)

	Increment system		
	10 μ	Standard	0.1 μ
Least input increment	0.01 mm	0.001 mm	0.0001 mm
Least command increment	0.01 mm	0.001 mm	0.0001 mm
Max. programmable dimension	±999999.99 mm	±99999.999 mm	±9999.9999 mm
Max. rapid traverse *1	240000 mm/min	100000 mm/min	24000 mm/min
Feedrate range *1	1~150000mm/min	1~100000mm/min	1~12000mm/min
Step feed	0.01, 0.1, 1, 10 mm/step	0.001, 0.01, 0.1, 1 mm/step	0.0001, 0,001, 0.01, 0.1, mm/step
Tool compensation	0~ ±999.99 mm	0~ ±999.999 mm	0~ ±999.9999 mm
Backlash compensation	0~ ±2.55 mm	0~ ±0.255 mm	0~ ±0.255 mm
Dwell time	0~ ±99999.999 sec	0~ ±99999.999 sec	0~ ±99999.999 sec

Linear axis (in case of metric thread for feed screw and inch input)

	Increment system		
	10 μ	Standard	0.1 μ
Least input increment	0.001 inch	0.0001 inch	0.00001 inch
Least command increment	0.01mm	0.001mm	0.0001mm
Max. programmable dimension	±99999.999 inch	±9999.9999 inch	±393.70078 inch
Max. rapid traverse *1	240000 mm/min	100000 mm/min	24000 mm/min
Feedrate range *1	0.01~6000 inch/min	0.01~4000 inch/min	0.01~480 inch/min
Step feed	0.001, 0.01, 0.1, 1 inch/step	0.0001, 0.001, 0.01, 0.1 inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation	0~ ±99.999 inch	0~ ±99.9999 inch	0~ ±99.9999 inch
Backlash compensation	0~ ±2.55 mm	0~ ±0.255 mm	0~ ±0.255 mm
Dwell time	0~99999.999 sec	0~99999.999 sec	0~9999.999 9sec

Linear axis (in case of inch thread for feed screw and inch input)

	Increment system		
	10 μ	Standard	0.1 μ
Least input increment	0.001 inch	0.0001 inch	0.00001 inch
Least command increment	0.001 inch	0.0001 inch	0.00001 inch
Max. programmable dimension	±99999.999 inch	±9999.9999 inch	±9999.9999 inch
Max. rapid traverse *1	9600 inch/min	4000 inch/min	960 inch/min
Feedrate range *1	0.01~6000 inch/min	0.01~4000 inch/min	0.01~480 inch/min
Step feed	0.001, 0.01, 0.1, 1 inch/step	0.0001, 0.001, 0.01, 0.1 inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation	0~ ±99.999 inch	0~ ±99.9999 inch	0~ ±99.9999 inch
Backlash compensation	0~ ±0.255 inch	0~ ±0.0255 inch	0~ ±0.0255 inch
Dwell time	0~99999.999 sec	0~99999.999 sec	0~9999.9999 sec

Linear axis (in case of inch thread for feed screw and metric input)

	Increment system		
	10 μ	Standard	0.1 μ
Least input increment	0.01 mm	0.001 mm	0.0001 mm
Least command increment	0.001 inch	0.0001 inch	0.00001 inch
Max. programmable dimension	±999999.99 mm	±99999.999 mm	±9999.9999mm
Max. rapid traverse *1	9600 inch/min	4000 inch/min	960 inch/min
Feedrate range *1	1~150000 mm/min	1~100000 mm/min	1~12000 mm/min
Step feed	0.01, 0.1, 1, 10 mm/step	0.001, 0.01, 0.1, 1 mm/step	0.0001, 0.001, 0.01, 0.1 mm/step
Tool compensation	0~ ±999.99 mm	0~ ±999.999 mm	0~ ±999.9999 mm
Backlash compensation	0~ ±0.255 inch	0~ ±0.0255 inch	0~ ±0.0255 inch
Dwell time	0~99999.999 sec	0~99999.999 sec	0~9999.9999 sec

Rotation axis

	Increment system	
	Standard	0.0001 deg
Least input increment	0.001 deg	0.0001 deg
Least command increment	0.001 deg	0.0001 deg
Max. programmable dimension	±99999.999 deg	±9999.9999 deg
Max. rapid traverse *1	100000 deg/min	24000 deg/min
Feedrate range *1	1~100000 deg/min	1~12000 deg/min
Step feed	0.001, 0.01, 0.1, 1 deg/step	0.0001, 0.001, 0.01, 0.1 deg/step
Tool compensation	0~ ±999.999 mm	0~ ±999.9999 mm
Backlash compensation	0~ ±0.255 deg	0~ ±0.255 deg

*1 The feed rate range shown above are limitations depending on CNC interpolation capacity. When regarded as a whole system, limitations depending on servo system must also be considered.

APPENDIX 2 FUNCTION AND TAPE FORMAT

(1) T series
The symbols in the list represent the followings:
P—··X—Z—
As seen above, the format consists of a combination of arbitrary axis addresses among X, Z.
x·· First basic axis (X)
z·· Second basic axis (z)

Functions	Illustrations	Tape format
Positioning(G00)	Start point	G00g__ ;
Linear interpolation(G01)	Start point	G00g__ F__ ;
Circular interpolation (G02,G03)		{G02 / G03} X-Z- {R- / I-K-} F-;
Dwell(G04)		G04 {X- / P-};
Change of offset value by program		Geometry offset amount G10P_X_Z_R_Q_; P = 10000 + Geometry offset amount Wear offset amount G10P_X_Z_C_Q_; P = Wear offset amount number
Inch/metri conversion (G20,G21)		Inch input G20; Metric input G21;
Spindle speed fluctuation detection (G25,G26)		G25; G26P-Q-R-;
Reference point return check (G27)	Start point	G27g-;

Functions	Illustrations	Tape format
Reference point return (G28) 2nd Reference point return(G30)		G28g – ; G30g – ;
Tool nose R compensation (G40,G41,G42)		{ G40 G41 G42 } P – ;
Skip function(G31)		G31P – F – ;
Thread cutting(G32)		Equal lead thread cutting G32P – F – ;
Automatic tool compensation (G36,G37)		G36X xa ; G37Z za ;
Coordinate system setting Spindle speed setting(G50)		G50X – Z – ; (Coodinate system setting) G50S – ; (Spindle speed setting)
Mirror image for double turret (G68,G69)		G68; Mirror image for double turret on G69; Mirrur image cancel

Functions	Illustrations	Tape format
Custom macro(G65)	Macro instruction Format G65HmP#iQ#jR#k; m;Display function of macro by 01～99 #i;Name of variable to which arithmetic result is entered. #j;Name 1 of variable to be calculated. #k;Name 2 of variable to be calculated. (Meaning)#i = #j⊕#k ↑ Display by operator Hm	G65HmP#iQ#jR#k;
Feed per minute /feed per revolution	mm/min inch/min mm/rev inch/rev	G98 ..... F－; G99 ..... F－;
Constant surface speed control (G96,97)	m/min or feet/min 	G96S－; G97; ..... Cancel
Chamfering and corner R		X－ {C±k / R－}; X－ {C±k / R－};
Canned cycle for cutting(G71～G76) (G90,G92,G94)	See "Canned cycle".	N_G70P_Q_; G71U_R_; G71P_Q_U_W_F_S_T_; G72W_R_; G72P_Q_U_W_F_S_T_; G73U_W_R_; G73P_Q_U_W_F_S_T_; G74R_; G74X(u)_Z(w)_P_Q_R_F_; G75R_; G75X(u)_Z(w)_P_Q_R_F_; G76P_Q_R; G76X(u)_Z(w)_P_Q_R_F_; {G90 / G92}X_Z_I_F; G94X_Z_K_F_;

(2) M series

The symbols in the list represent the followings:
*P*__:X__Y__Z__A__
As seen above, the format consists of a combination of arbitrary axis addresses among
X: First basic axis (X usually)
Y: Second basic axis (Y usually)
Z: Third basic axis (Z usually)

Functions	Illustrations	Tape format
Positioning (G00)	Start point → *P*	G00*P*__ ;
Linear interpolation (G01)	Start point → *P*	G00*P*——F——;
Circular interpolation (G02,G03)	Start point G02: J, R, I, (x, y) (x, y), G03: J, R, I (In case of X-Y plane)	G17 {G02, G03} X-Y- {R-, I-J-} F-; G18 {G02, G03} X-Z- {R-, I-K-} F-; G19 {G02, G03} Y-Z- {R-, J-K-} F-;
Dwell (G04)		G04 {X-, *P*-} ;
Exact stop(G04)	Speed Time	
Change of offset value by program (G10)		G10*P*-R-;
Tool length compensation B (G39~G42) Tool length compensation C (G40~G42)	G41 G42 G40	{G17, G18, G19} {G40, G41, G42} H__; H : Tool offset number

Functions	Illustrations	Tape format
Tool length compensation A (G43, G44, G49)		{G43, G44} Z_H_; {G43, G44} H_; H : Tool offset No. G49 : Cancel
Tool length compensation B (G43, G44, G49)		{G17, G18, G19} {G43, G44} {Z_, Y_, X_} H_; {G17, G18, G19} {G43, G44} H_; H : Tool offset No. G49 : Cancel
Inch/metric conversion (G20,G21)		Inch input G20; Metric input G21;
Reference point return check(G27)		G27*P*−;
Reference point return (G28) 2nd reference point return(G30)		G28*P*−; G30*P*−;
Return from reference point(G29)		G29*P*−;
Skip function(G31)		G31*P*−F−;

Functions	Illustrations	Tape format
Custom macro A (G65, G66, G67)	Macro instruction Format G65HmP#iQ#jR#k; m;Display function of macro by 01～99 #i;Name of variable to which arithmetic result is entered. #j;Name 1 of variable to be calculated. #k;Name 2 of variable to be calculated. (Meaning)#i = #j⊕#k ↑ Display by calucurate Hm	G65HmP#iQ#jR#k; Modal call G66P__; G67: Cancel
Canned cycles (G73, G74, G80～G89)	See item "Canned cycle" in the main text.	G80; ····· Cancel G73 G74 G76 G81 : G89 } X__Y__Z__ P__Q__R__F__K__
Absolute/incremental programming (G90/G91)		G90_____ ; Absolute programming G91_____ ; Incremental programming G90__G91__ ; Combined use
Change of work coordinate (G92)		G92P____ ;
Initial point return/ R point return (G98,G99)		G98_____ ; G99_____ ;

APPENDIX 3 LIST OF CODE

(1) T series

ISO code										EIA code											
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		Meaning
0			○	○		∘				0			○			∘					Numeral 0
1	○		○	○		∘			○	1						∘			○		Numeral 1
2	○		○	○		∘		○		2						∘		○			Numeral 2
3			○	○		∘		○	○	3				○		∘		○	○		Numeral 3
4	○		○	○		∘	○			4						∘	○				Numeral 4
5			○	○		∘	○		○	5				○		∘	○		○		Numeral 5
6			○	○		∘	○	○		6				○		∘	○	○			Numeral 6
7	○		○	○		∘	○	○	○	7						∘	○	○	○		Numeral 7
8	○		○	○	○	∘				8					○	∘					Numeral 8
9			○	○	○	∘			○	9				○	○	∘			○		Numeral 9
A		○				∘			○	a		○	○			∘			○		Address A
B		○				∘		○		b		○	○			∘		○		?	Address B
C	○	○				∘		○	○	c		○	○	○		∘		○	○		Address C
D		○				∘	○			d		○	○			∘	○			?	Address D
E	○	○				∘	○		○	e		○	○	○		∘	○		○	?	Address E
F	○	○				∘	○	○		f		○	○	○		∘	○	○			Address F
G		○				∘	○	○	○	g		○	○			∘	○	○	○		Address G
H		○			○	∘				h		○	○		○	∘					Address H
I	○	○			○	∘			○	i		○	○	○	○	∘			○		Address I
J	○	○			○	∘		○		j		○		○		∘			○	?	Address J
K		○			○	∘		○	○	k		○		○		∘		○			Address K
L	○	○			○	∘	○			l		○				∘		○	○	?	Address L
M		○			○	∘	○		○	m		○		○		∘	○				Address M
N		○			○	∘	○	○		n		○				∘	○		○		Address N
O	○	○			○	∘	○	○	○	o		○				∘	○	○			Address O
P		○		○		∘				p		○		○		∘	○	○	○		Address P
Q	○	○		○		∘			○	q		○		○	○	∘					Address Q
R	○	○		○		∘		○		r		○			○	∘			○		Address R
S		○		○		∘		○	○	s			○	○		∘		○			Address S
T	○	○		○		∘	○			t			○			∘		○	○		Address T
U		○		○		∘	○		○	u			○	○		∘	○				Address U
V		○		○		∘	○	○		v			○			∘	○		○	?	Address V
W	○	○		○		∘	○	○	○	w			○			∘	○	○			Address W
X	○	○		○	○	∘				x			○	○		∘	○	○	○		Address X
Y		○		○	○	∘			○	y			○	○	○	∘				?	Address Y
Z		○		○	○	∘		○		z			○		○	∘			○		Address Z
DEL	○	○	○	○	○	∘	○	○	○	Del		○	○	○	○	∘	○	○	○	*	Delete (cancel an error punch).
NUL						∘				Blank						∘				*	Not punched. Cannot be used in significant section in EIA code.
BS	○				○	∘				BS			○		○	∘		○		*	Back space
HT					○	∘			○	Tab			○	○	○	∘	○	○		*	Tabulator
LF or NL					○	∘		○		CR or EOB	○					∘					End of block
CR	○				○	∘	○		○											*	Carriage return
SP	○		○			∘				SP				○		∘				*	Space
%	○		○			∘	○		○	ER					○	∘		○	○		Absolute rewind stop
(			○		○	∘				(2–4–5)				○	○	∘		○			Control out (a comment is started)
)	○		○		○	∘			○	(2–4–7)		○			○	∘		○			Control in (the end of a comment)
+			○		○	∘		○	○	+		○	○	○		∘				*	Positive sign
-			○		○	∘	○		○	-		○				∘					Negative sign
:			○	○	○	∘		○													Colon
/	○		○		○	∘	○	○	○	/			○	○		∘			○		Optional block skip
.			○		○	∘	○	○		.		○	○		○	∘		○	○		Period (A decimal point)
#	○		○			∘		○	○											*	Sharp.
$			○			∘	○													*	Dollar sign
&	○		○			∘	○	○		&					○	∘	○	○		*	Ampersand
'			○			∘	○	○	○											*	Apostrophe
*	○		○		○	∘		○												*	Asterisk
,	○		○		○	∘	○			,			○	○	○	∘		○	○	*	Comma
;	○		○	○	○	∘		○	○											*	Semicolon
<			○	○	○	∘	○													*	Left angle bracket
=	○		○	○	○	∘	○		○											*	Equal
>	○		○	○	○	∘	○	○												*	Right angle bracket
?			○	○	○	∘	○	○	○											*	Question mark
@	○	○				∘														*	Commercial at mark
"			○			∘		○												*	Quotation

(2) M series

ISO code										EIA code											
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		Meaning
0			○	○		∘				0			○			∘					Numeral 0
1	○		○	○		∘			○	1						∘			○		Numeral 1
2	○		○	○		∘		○		2						∘		○			Numeral 2
3			○	○		∘		○	○	3				○		∘		○	○		Numeral 3
4	○		○	○		∘	○			4						∘	○				Numeral 4
5			○	○		∘	○		○	5				○		∘	○		○		Numeral 5
6			○	○		∘	○	○		6				○		∘	○	○			Numeral 6
7	○		○	○		∘	○	○	○	7						∘	○	○	○		Numeral 7
8	○		○	○	○	∘				8					○	∘					Numeral 8
9			○	○	○	∘			○	9				○	○	∘			○		Numeral 9
A		○				∘			○	a		○	○			∘			○	?	Address A
B		○				∘		○		b		○	○			∘		○			Address B
C	○	○				∘		○	○	c		○	○	○		∘		○	○	?	Address C
D		○				∘	○			d		○	○			∘	○				Address D
E	○	○				∘	○		○	e		○	○	○		∘	○		○	?	Address E
F	○	○				∘	○	○		f		○	○	○		∘	○	○			Address F
G		○				∘	○	○	○	g		○	○			∘	○	○	○		Address G
H		○			○	∘				h		○	○		○	∘					Address H
I	○	○			○	∘			○	i		○	○	○	○	∘			○		Address I
J	○	○			○	∘		○		j		○		○		∘			○		Address J
K		○			○	∘		○	○	k		○		○		∘		○			Address K
L	○	○			○	∘	○			l		○				∘		○	○	?	Address L
M		○			○	∘	○		○	m		○		○		∘	○				Address M
N		○			○	∘	○	○		n		○				∘	○		○		Address N
O	○	○			○	∘	○	○	○	o		○				∘	○	○			Not used at significant information section in ISO code. Assumed as program No. in EIA code.
P		○		○		∘				p		○		○		∘	○	○	○		Address P
Q	○	○		○		∘			○	q		○		○	○	∘					Address Q
R	○	○		○		∘		○		r		○			○	∘			○		Address R
S		○		○		∘		○	○	s			○	○		∘		○			Address S
T	○	○		○		∘	○			t			○			∘		○	○		Address T
U		○		○		∘	○		○	u			○	○		∘	○			?	Address U
V		○		○		∘	○	○		v			○			∘	○		○	?	Address V
W	○	○		○		∘	○	○	○	w			○			∘	○	○		?	Address W
X	○	○		○	○	∘				x			○	○		∘	○	○	○		Address X
Y		○		○	○	∘			○	y			○	○	○	∘					Address Y
Z		○		○	○	∘		○		z			○		○	∘			○		Address Z
DEL	○	○	○	○	○	∘	○	○	○	Del		○	○	○	○	∘	○	○	○	*	Delete (cancel an error punch).
NUL						∘				Blank						∘				*	Not punched. Can not be used in significant section in EIA code.
BS	○				○	∘				BS			○		○	∘		○		*	Back space
HT					○	∘			○	Tab			○	○	○	∘	○	○		*	Tabulator
LF or NL					○	∘		○		CR or EOB	○					∘					End of block
CR	○				○	∘	○		○											*	Carriage return
SP	○		○			∘				SP				○		∘				*	Space
%	○		○			∘	○		○	ER					○	∘		○	○		Absolute rewind stop
(			○		○	∘				(2-4-5)				○	○	∘		○			Control out (a comment is started)
)	○		○		○	∘			○	(2-4-7)		○			○	∘		○			Control in (the end of a comment)
+			○		○	∘		○	○	+		○	○	○		∘				*	Positive sign
-			○		○	∘	○		○	-		○				∘					Negative sign
:			○	○	○	∘		○													Colon
/	○		○		○	∘	○	○	○	/			○	○		∘			○		Optional block skip
.			○		○	∘	○	○		.		○	○		○	∘		○	○		Period (A decimal point)
#	○		○			∘		○	○											*	Sharpe
$			○			∘	○													*	Dollar sign
&	○		○			∘	○	○		&					○	∘	○	○		*	Ampersand
'			○			∘	○	○	○											*	Apostrophe
*	○		○		○	∘		○												*	Asterisk
,	○		○		○	∘	○			,			○	○	○	∘		○	○	*	Comma
;	○		○	○	○	∘		○	○											*	Semicolon
<			○	○	○	∘	○													*	Left angle bracket
=	○		○	○	○	∘	○		○											*	Equal
>	○		○	○	○	∘	○	○												*	Right angle bracket
?			○	○	○	∘	○	○	○											*	Question mark
@	○	○				∘														*	Commercial at mark
"			○			∘		○												*	Quotation

(Note 1) *:When read in the significant data zone,the codes are ignored.

(Note 2) ?:The codes are invalid in the significant data zone, but are registered in the memory.

(Note 3) Codes not in this table are ignored if their parity is correct

(Note 4) Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment zone.

(Note 5) A character with all eight holes punched does not generate TH alarm even in EIA code.

APPENDIX 4 EXTERNAL DIMENSIONS

Fig.1 External view of control unit A

Fig.2 External view of control unit B

Fig.3 External view of small 9" monochrome CRT/MDI

Fig.4 External view of small 9" color CRT/MDI

Fig.5 External view of 9" monochrome CRT/MDI with full-keyboard

Fig.6 External view of 9" color CRT/MDI with full-keyboard

Fig.7: External view of EL/MDI unit (Small Type)

Fig. 8 External view of 14" CRT/MDI unit

Fig. 9 External view of 9" monochrome CRT unit

Fig. 10 External view of EL unit

Fig. 11 External view of MDI unit (Small Type)

Fig.12 External view of MDI unit (Full-keyboard)

Note: Mechanical specifications of the position coder are as follows:

(1) Input axis inertia
1.0×10^{-3} kg · cm · sec^2 or less

(2) Input axis starting torque
100 g · cm or less

(3) Input axis permissible loads

	Radial	Thrust
Operation	1 kg or less	1 kg or less
Idle	20 kg or less	10 kg or less

Attach a pulley directly to the position coder shaft and drive the timing belt. Note that the loads conform with the above allowable value.

(4) Weight: 1 kg or less

Fig. 13 External view of position coder

A86L-0026-0002#102: Max. 4000 rpm

A86L-0026-0002#002: Max. 6000 rpm

Fig.14 External view of manual pulse generator (A860-0201-T001)

Fig.15 External view of input unit

Fig. 16 External view of battery unit

3. OUTER VIEW

3.1 Outer View of CRT/MDI Unit

Fig.17 External view of 00-TC/00-MC CRT/MDI unit

Fig.18 External view of operator's panel for 0-TC

Fig.19 External view of operator's panel for 0-MC

Fig.20 Machine operator's panel for 9" CRT/MDI with full-keyboard (0-TC)

Fig.21 Machine operator's panel for 9" CRT/MDI with full-keyboard (0-MC)

Fig.22 Machine operator's for 9" monochrome CRT/MDI

Fig. 23 Machine operator's panel for 9" color CRT/MDI

Fig.24 Machine operator's panel for 9" CRT/MDI with full-keyboard

Fig. 25 Machine operator's panel for 9" CRT/MDI with full-keyboard

Fig.26 External view of additional I/O B2

Cable length: 1.5 m
Weight: about 1.2 kg
Unit: mm

Fig. 27 External view of FANUC PPR

Fig.28 External view of FANUC cassette adapter

Cable length: 1.5 m
Weight: 3.3 kg
Unit: mm

Fig.29 External view of floppy cassette adapter

Fig.19 External view of portable tape reader

INDEX

2nd, 3rd and 4th reference point return (G30) . . . . . 1 - 37

<A>

Absolute and incremental programming (G90, G91) . . . . . 1 - 41
Activation of automatic operation . . . . . 1 - 104
Actual spindle speed output . . . . . 1 - 44
Additional axis neglect . . . . . 1 - 102
Alarm display . . . . . 1 - 124
Alarm signal . . . . . 1 - 139
Angular axis control (GCC, GSC) . . . . . 1 - 82
Arbitrary profile machining programs . . . . . 2 - 3, 2 - 8
Automatic acceleration/deceleration . . . . . 1 - 34
Automatic coordinate system setting . . . . . 1 - 39
Automatic corner override (G62) . . . . . 1 - 34
Automatic operation signal . . . . . 1 - 139
Automatic operation start lamp signal . . . . . 1 - 139
Automatic operation stop . . . . . 1 - 104
Automatic operation . . . . . 1 - 103
Automatic reference point return (G28, G29) . . . . . 1 - 37
Automatic selection of cutting conditions and pre-tools . . . . . 2 - 9
Automatic selection of tools, cutting conditions and pre-tools . . . . . 2 - 4
Automatic tool length measurement (G37) . . . . . 1 - 50
Automatic tool offset . . . . . 1 - 91
Auxiliary function lock . . . . . 1 - 111
Axes control by PMC . . . . . 1 - 20
Axis control . . . . . 1 - 102

<B>

Background editing . . . . . 1 - 130
Backlash compensation . . . . . 1 - 89
Balance cut (G68, G69) . . . . . 1 - 152
Basic addresses and command value range . . . . . 1 - 53
Battery alarm signal . . . . . 1 - 139
Battery unit . . . . . 1 - 146

<C>

Cabinet . . . . . 1 - 146
Cables and connectors . . . . . 1 - 148
Canned cycle for drilling . . . . . 1 - 73
Canned cycles (G73, G74, G76, G80 - G89, G98, G99) (M series) . . . . . 1 - 57
Canned cycles . . . . . 1 - 60
Cf axis control . . . . . 1 - 19
Chamfering and corner R . . . . . 1 - 66

Change of axis name . . . . . 1 - 20
Circular interpolation (G02, G03) . . . . . 1 - 26
Clock Function . . . . . 1 - 139
CNC ready signal . . . . . 1 - 139
Command value display . . . . . 1 - 125
Compensation functions . . . . . 1 - 86
Constant surface speed control (G96, G97) . . . . . 1 - 43
Continuous dressing (GSC) . . . . . 1 - 89
Continuous thread cutting . . . . . 1 - 31
Control unit . . . . . 1 - 146
Control-in/control-out . . . . . 1 - 55
Controlled axes . . . . . 1 - 18
Conversational input function with figure for cylindlical grinding machine
(order-made macro) . . . . . 1 - 81
Conversational programming with graphic function . . . . . 1 - 70
Coordinate system rotation (G68,G69) . . . . . 1 - 72
Coordinate system setting . . . . . 1 - 39
Coordinate system shift . . . . . 1 - 39
Coordinate system . . . . . 1 - 39
Coordinate value and dimension . . . . . 1 - 41
Created tool post name display (TT) . . . . . 1 - 128
CRT/MDI panel . . . . . 1 - 113
CRT/MDI . . . . . 1 - 113
Cs axis control . . . . . 1 - 19
Cs contour control . . . . . 1 - 46
Current position display . . . . . 1 - 123
Custom macro B . . . . . 1 - 97
Custom macro modal call . . . . . 1 - 99
Custom macro A . . . . . 1 - 95
Cutter compensation B, C (G39 - G42) (M series) . . . . . 1 - 86
Cutting cycle A (G90) . . . . . 1 - 60
Cutting cycle B (G94) . . . . . 1 - 61
Cutting feed rate clamp . . . . . 1 - 32
Cutting feed rate . . . . . 1 - 32
Cutting mode (G64) . . . . . 1 - 36
Cyrindrical interpolation (G107) . . . . . 1 - 28

<D>

Data input/output . . . . . 1 - 135
Decimal point input/pocket calculator type decimal point input . . . . . 1 - 42
Deep hole drill cycle (G74) . . . . . 1 - 63
Diameter and radius programming . . . . . 1 - 42
Dinamic graphic display . . . . . 1 - 128
Direct drawing dimensions programming . . . . . 1 - 68
Direct measured value input A of tool offset . . . . . 1 - 92
Direct measured value input B of tool offset . . . . . 1 - 93

Direct measured value input for work coordinate system shift . . . . . 1 - 40
Directory display of the floppy cassette . . . . . 1 - 56
Distribution end signal1 . . . . . 1 - 139
Dry run . . . . . 1 - 111
Dwell (G04) . . . . . 1 - 35
Dynamic graphic display (for lathes) . . . . . 2 - 5
Dynamic graphic display (for machining center) . . . . . 2 - 10

<E>

Emergency stop . . . . . 1 - 137
Environmental requirements . . . . . 1 - 147
Exact stop (G09) . . . . . 1 - 35
Exact stop mode (G61) . . . . . 1 - 35
Execution of automatic operation . . . . . 1 - 104
Expanded part program editing . . . . . 1 - 130
Expanded stored stroke check . . . . . 1 - 138
External data input . . . . . 1 - 141
External deceleration . . . . . 1 - 138
External dimensions . . . . . A - 16
External key-input function . . . . . 1 - 141
External machine origin shift . . . . . 1 - 142
External message function . . . . . 1 - 141
External motion function (G80, G81) . . . . . 1 - 70
External setting function of stored stroke limit . . . . . 1 - 137
External tool compensation . . . . . 1 - 141
External work number search . . . . . 1 - 143

<F>

F1-digit feed . . . . . 1 - 33
F10/11 tape format . . . . . 1 - 56
FANUC FA card . . . . . 1 - 136
FANUC floppy cassette . . . . . 1 - 135
FANUC PPR . . . . . 1 - 135
FANUC program file mate . . . . . 1 - 136
Feed functions . . . . . 1 - 32
Feed hold signal . . . . . 1 - 139
Feed hold . . . . . 1 - 105
Feed per minute (G94) . . . . . 1 - 32
Feed per revolution (G99) . . . . . 1 - 33
Feed rate override . . . . . 1 - 34
Finish cycle (G70) . . . . . 1 - 63
Function and tape format . . . . . A - 7
Function for 4-axis lathes (0-TTC) . . . . . 1 - 151
Functions to simplify programming . . . . . 1 - 57

<G>

General purpose switch for software operator's panel ..... 1 - 127
Graphic conversation for 2-axes lathe (0-TF) ..... 2 - 2
Graphic conversation for machining center (0-Mate MF/0-MF) ..... 2 - 6
Graphic conversational input ..... 1 - 129
Graphic display ..... 1 - 127
Grinding canned cycle (GCC) ..... 1 - 74
Grinding canned cycle (GSC) ..... 1 - 75
Grinding canned cycle ..... 1 - 74

<H>

Handle interruption ..... 1 - 106
Helical cutting (G02, G03) ..... 1 - 2
High-speed machining function ..... 1 - 132
High-speed remote buffer A ..... 1 - 154
High-speed remote buffer B ..... 1 - 155
High-speed skip signal input ..... 1 - 91

<I>

I/O device external control ..... 1 - 129
I/O device interface ..... 1 - 135
Inch/metric conversion (G20, G21) ..... 1 - 41
Incremental feed ..... 1 - 109
Infeed control ..... 1 - 84
Input devices ..... 1 - 135
Input power voltage ..... 1 - 146
Input unit ..... 1 - 146
Input/output simultaneous operation ..... 1 - 131
Interference check function ..... 1 - 152
Interlock ..... 1 - 138
Interpolation functions ..... 1 - 26
Interruption type custom macro ..... 1 - 101

<J>

Jog override ..... 1 - 34

<K>

Keytouch ..... 2 - 6

<L>

Label skip ..... 1 - 55
Language ..... 1 - 128
Linear acceleration/deceleration after cutting feed interpolation ..... 1 - 35

Linear interpolation (G01) . . . . . 1 - 26
List of code . . . . . A - 13

<M>

Machine interface . . . . . 1 - 144
Machine lock . . . . . 1 - 111
Machine operator's panel . . . . . 1 - 153
Macro executer . . . . . 1 - 101
Main program . . . . . 1 - 52
Manual absolute on/off . . . . . 1 - 107
Manual feed . . . . . 1 - 108
Manual handle feed . . . . . 1 - 110
Manual interruption during automatic operation . . . . . 1 - 106
Manual operation . . . . . 1 - 108
Manual reference point return . . . . . 1 - 37
MDI operation/MDI operation B . . . . . 1 - 119
Measurement function . . . . . 1 - 91
Mechanical handle feed . . . . . 1 - 102
Menu programming . . . . . 1 - 71, 1 - 129
Menu screen . . . . . 2 - 3, 2 - 7
Mirror Image for double turrets (G68, G69) . . . . . 1 - 67
Mirror image (M series) . . . . . 1 - 102
Miscellaneous function (M2 digit) . . . . . 1 - 51
Miscellaneous functions . . . . . 1 - 51
Multi-piece machining . . . . . 2 - 8
Multi-spindle Control . . . . . 1 - 45
Multi-step skip . . . . . 1 - 84
Multi-tap transformer . . . . . 1 - 146
Multiple repetitive canned cycle (G70-G76) . . . . . 1 - 62

<N>

Normal direction control (GSC) . . . . . 1 - 82
Number of registered programs . . . . . 1 - 129

<O>

Offset value counter input . . . . . 1 - 93
Operation mode . . . . . 1 - 103
Optimum tool order . . . . . 2 - 8
Optional angle chamfering . . . . . 1 - 67
Optional angle corner rounding . . . . . 1 - 67
Optional block skip . . . . . 1 - 55
Order made macro . . . . . 1 - 100
Outer diameter grooving cycle (G75) . . . . . 1 - 64
Outline . . . . . 1 - 1
Override cancel . . . . . 1 - 34

Override . . . . . 1 - 34
Overtravel functions . . . . . 1 - 137
Overtravel . . . . . 1 - 137

<P>

Part program storage & editing . . . . . 1 - 129
Part program storage length . . . . . 1 - 129
Pattern data input . . . . . 1 - 100, 1 - 129
Pattern repeating (G73) . . . . . 1 - 63
peration when converting the machining program prepared by
graphic conversation to an NC format program and outputting externally via
the reader/puncher interface . . . . . 1 - 131
Play back . . . . . 1 - 129
Polar coordinate command (G15, G16) . . . . . 1 - 41
Polar coordinate interpolation (G112, G113) . . . . . 1 - 27
Poligonal turning . . . . . 1 - 133
Portable tape reader . . . . . 1 - 136
Position coder . . . . . 1 - 150
Position detection system . . . . . 1 - 149
Position detector . . . . . 1 - 149
Position signal output . . . . . 1 - 107
Position switch function . . . . . 1 - 139
Positioning (G00) . . . . . 1 - 26
Power ready signal . . . . . 1 - 139
Power . . . . . 1 - 146
Preparatory function (G function) . . . . . 1 - 21
Program configuration . . . . . 1 - 52
Program display . . . . . 1 - 122
Program end (M02, M30) . . . . . 1 - 105
Program list display . . . . . 1 - 122
Program name . . . . . 1 - 56
Program number . . . . . 1 - 52
Program protect key . . . . . 1 - 127
Program restart . . . . . 1 - 107
Program stop (M00, M01) . . . . . 1 - 104
Program test functions . . . . . 1 - 111
Programmable machine controller (PMC) . . . . . 1 - 145

<R>

Table of range of command value . . . . . A - 1
Rapid traverse override . . . . . 1 - 34
Rapid traverse . . . . . 1 - 32
Reference point return check (G27) . . . . . 1 - 37
Reference point . . . . . 1 - 37
Regenerative electric discharge unit . . . . . 1 - 149

Remote buffer . . . . . 1 - 154
Reset signal . . . . . 1 - 139
Reset . . . . . 1 - 106
Rigid tap . . . . . 1 - 73
Rough cutting cycle of outer diameter (G71) . . . . . 1 - 62
Rough facing cycle (G72) . . . . . 1 - 62
Run time display/parts count display . . . . . 1 - 125

<S>

S analog voltage control by PMC . . . . . 1 - 44
S analog voltage output . . . . . 1 - 43
S code output . . . . . 1 - 43
Safety functions . . . . . 1 - 137
Scaling (G50,G51) . . . . . 1 - 72
Second miscellaneous function (B function) . . . . . 1 - 51
Selection of execution program . . . . . 1 - 103
Self diagnostic functions . . . . . 1 - 134
Sequence number comparison and stop . . . . . 1 - 106
Sequence number . . . . . 1 - 52
Serial spindle . . . . . 1 - 46
Servo motor . . . . . 1 - 149
Servo off . . . . . 1 - 102
Servo power transformer . . . . . 1 - 149
Servo ready signal . . . . . 1 - 139
Servo system (digital servo) . . . . . 1 - 149
Set up guidance display . . . . . 2 - 3
Setting and display of custom macro variables . . . . . 1 - 121
Setting and display of offset value . . . . . 1 - 120
Setting and display of parameter . . . . . 1 - 121
Setting and display of setting data . . . . . 1 - 120
Setting and display . . . . . 1 - 119
Setting work coordinate system-method by G54-G59 . . . . . 1 - 40
Simple synchronous control . . . . . 1 - 130
Simplified tool life management . . . . . 1 - 48
Single block . . . . . 1 - 112
Single direction positioning (G60) . . . . . 1 - 29
Skip function (G31) . . . . . 1 - 91
Software operator's panel . . . . . 1 - 126
Spindle functions . . . . . 1 - 43
Spindle speed clamp (G50) . . . . . 1 - 44
Spindle speed fluctuation detecting function (G25, G26) . . . . . 1 - 44
Spindle speed override . . . . . 1 - 44
Spindle synchronization control . . . . . 1 - 47
Status output . . . . . 1 - 139
Stored pitch error compensation . . . . . 1 - 89
Stored stroke limit . . . . . 1 - 137

Sub program ..... 1 - 52
Synchronous feed (G95) ..... 1 - 33

<T>

T code output ..... 1 - 48
Tangential speed constant control ..... 1 - 32
Tape codes ..... 1 - 52
Tape format ..... 1 - 54
Tapping mode (G63) ..... 1 - 36
Thread cutting cycle (G92) ..... 1 - 61
Thread cutting cycle retract ..... 1 - 105
Thread cutting ..... 1 - 30
Threading cycle (G76) ..... 1 - 65
Tool functions ..... 1 - 48
Tool geometry offset and tool wear offset ..... 1 - 88
Tool length compensation (G43, G44, G49) (M series) ..... 1 - 86
Tool length compensation A ..... 1 - 86
Tool length compensation B ..... 1 - 86
Tool length measurement ..... 1 - 91
Tool life management ..... 1 - 48
Tool nose radius compensation (G40-G42) (T series) ..... 1 - 88
Tool offset (G43, G44, G49) (T series) ..... 1 - 86
Tool offset number ..... 1 - 87
Tool offset value program input (G10) ..... 1 - 87

<V>

Velocity control unit ..... 1 - 149

<W>

Weight ..... 1 - 147
Work coordinate system shift ..... 1 - 40

<X>

X-axis mirror image (T series) ..... 1 - 102

<Y>

Y axis offset (T series) ..... 1 - 89

<Z>

Z-axis command cancel ..... 1 - 112

Revision Record

Series 0 / Series 0-Mate DESCRIPTION (B-61392E)

Edition	Date	Contents	Edition	Date	Contents
03	Aug. '92	· Addition of 10.10 Serial Spindle · Addition of 10.11 Cs Contour Control · Addition of 10.12 Spindle Synchronization Control · Addition of 29.12 Position Switch Function · Addition of 30.5 External Machine Origin Shift			
02	Jan. '91	Addition of Series 0 / 00-GCC, Series 0 / 00-GSC			
01	Jan. '89	———			

- *No part of this manual may be reproduced in any form.*

- *All specifications and designs are subject to change without notice.*