GE Fanuc Automation

Motion Control Products

Power Mate i – Models D and H
Motion Controllers

Description Manual

GFZ-63172EN/01

September 1998
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 1998 GE Fanuc Automation North America, Inc. All Rights Reserved.
SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units. It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.

Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

Contents

1. DEFINITION OF WARNING, CAUTION, AND NOTE ...................... s–2
2. GENERAL WARNINGS AND CAUTIONS ............................... s–3
3. WARNINGS AND CAUTIONS RELATED TO PROGRAMMING .......... s–5
4. WARNINGS AND CAUTIONS RELATED TO HANDLING ............... s–7
5. WARNINGS RELATED TO DAILY MAINTENANCE .................... s–9
This manual includes safety precautions for protecting the user and preventing damage to the machine. Precautions are classified into Warning and Caution according to their bearing on safety. Also, supplementary information is described as a Note. Read the Warning, Caution, and Note thoroughly before attempting to use the machine.

**WARNING**

Applied when there is a danger of the user being injured or when there is a danger of both the user being injured and the equipment being damaged if the approved procedure is not observed.

**CAUTION**

Applied when there is a danger of the equipment being damaged, if the approved procedure is not observed.

**NOTE**

The Note is used to indicate supplementary information other than Warning and Caution.

Read this manual carefully, and store it in a safe place.
2 GENERAL WARNINGS AND CAUTIONS

**WARNING**

1. Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

2. Before operating the machine, thoroughly check the entered data. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

3. Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate. If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

4. When using a tool compensation function, thoroughly check the direction and amount of compensation. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

5. The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change. Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

6. Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit. Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.

7. The operator’s manual supplied with a CNC unit provide an overall description of the machine’s functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.
8. Some functions may have been implemented at the request of the machine–tool builder. When using such functions, refer to the manual supplied by the machine–tool builder for details of their use and any related cautions.

NOTE

Data such as programs, parameters, and macro variables is stored in the battery–powered SRAM in the CNC. Usually, they are retained even if the power is turned off. Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from SRAM as part of error recovery. Back up data to a memory card to enable quick recovery in the event of unexpected data loss. The Power Mate i has a function for backing up the contents of the SRAM to the internal FROM. To back up data, use this function as well.
3 WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

This section covers the major safety precautions related to programming. Before attempting to perform programming, read the supplied operator’s manual and programming manual carefully such that you are fully familiar with their contents.

**WARNING**

1. **Coordinate system setting**
   
   If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command. Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2. **Positioning by nonlinear interpolation**
   
   When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming. Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. **Function involving a rotation axis**
   
   When programming polar coordinate interpolation or normal–direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely. Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

4. **Inch/metric conversion**
   
   Switching between inch and metric inputs does not convert the measurement units of data such as the workpiece origin offset, parameter, and current position. Before starting the machine, therefore, determine which measurement units are being used. Attempting to perform an operation with invalid data specified may damage the tool, the machine itself, the workpiece, or cause injury to the user.

5. **Constant surface speed control**
   
   When an axis subject to constant surface speed control approaches the origin of the workpiece coordinate system, the spindle speed may become excessively high. Therefore, it is necessary to specify a maximum allowable speed. Specifying the maximum allowable speed incorrectly may damage the tool, the machine itself, the workpiece, or cause injury to the user.
6. Stroke check

After switching on the power, perform a manual reference position return as required. Stroke check is not possible before manual reference position return is performed. Note that when stroke check is disabled, an alarm is not issued even if a stroke limit is exceeded, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

7. Tool post interference check

A tool post interference check is performed based on the tool data specified during automatic operation. If the tool specification does not match the tool actually being used, the interference check cannot be made correctly, possibly damaging the tool or the machine itself, or causing injury to the user.

After switching on the power, or after selecting a tool post manually, always start automatic operation and specify the tool number of the tool to be used.

8. Absolute/incremental mode

If a program created with absolute values is run in incremental mode, or vice versa, the machine may behave unexpectedly.

9. Plane selection

If an incorrect plane is specified for circular interpolation, helical interpolation, or a canned cycle, the machine may behave unexpectedly. Refer to the descriptions of the respective functions for details.

10. Torque limit skip

Before attempting a torque limit skip, apply the torque limit. If a torque limit skip is specified without the torque limit actually being applied, a move command will be executed without performing a skip.

11. Programmable mirror image

Note that programmed operations vary considerably when a programmable mirror image is enabled.

12. Compensation function

If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine.

Before issuing any of the above commands, therefore, always cancel compensation function mode.
This section presents safety precautions related to the handling of machine tools. Before attempting to operate your machine, read the supplied operator’s manual and programming manual carefully, such that you are fully familiar with their contents.

**WARNING**

1. **Manual operation**

   When operating the machine manually, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and feedrate have been specified correctly. Incorrect operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

2. **Manual reference position return**

   After switching on the power, perform manual reference position return as required. If the machine is operated without first performing manual reference position return, it may behave unexpectedly. Stroke check is not possible before manual reference position return is performed. An unexpected operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. **Manual numeric command**

   When issuing a manual numeric command, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and command have been specified correctly, and that the entered values are valid. Attempting to operate the machine with an invalid command specified may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

4. **Manual handle feed**

   In manual handle feed, rotating the handle with a large scale factor, such as 100, applied causes the tool and table to move rapidly. Careless handling may damage the tool and/or machine, or cause injury to the user.

5. **Disabled override**

   If override is disabled (according to the specification in a macro variable) during threading, rigid tapping, or other tapping, the speed cannot be predicted, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

6. **Origin/preset operation**

   Basically, never attempt an origin/preset operation when the machine is operating under the control of a program. Otherwise, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the tool, or causing injury to the user.
WARNING

7. Workpiece coordinate system shift

Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully.

If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

8. Software operator’s panel and menu switches

Using the software operator’s panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator’s panel, such as mode change, override value change, and jog feed commands.

Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

9. Manual intervention

If manual intervention is performed during programmed operation of the machine, the tool path may vary when the machine is restarted. Before restarting the machine after manual intervention, therefore, confirm the settings of the manual absolute switches, parameters, and absolute/incremental command mode.

10. Feed hold, override, and single block

The feed hold, feedrate override, and single block functions can be disabled using custom macro system variable #3004. Be careful when operating the machine in this case.

11. Dry run

Usually, a dry run is used to confirm the operation of the machine. During a dry run, the machine operates at dry run speed, which differs from the corresponding programmed feedrate. Note that the dry run speed may sometimes be higher than the programmed feed rate.

12. Cutter and tool nose radius compensation in MDI mode

Pay careful attention to a tool path specified by a command in MDI mode, because cutter or tool nose radius compensation is not applied. When a command is entered from the MDI to interrupt in automatic operation in cutter or tool nose radius compensation mode, pay particular attention to the tool path when automatic operation is subsequently resumed. Refer to the descriptions of the corresponding functions for details.

13. Program editing

If the machine is stopped, after which the machining program is edited (modification, insertion, or deletion), the machine may behave unexpectedly if machining is resumed under the control of that program. Basically, do not modify, insert, or delete commands from a machining program while it is in use.
5 WARNINGS RELATED TO DAILY MAINTENANCE

WARNING

1. SRAM backup battery replacement

Only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high–voltage circuits (marked △ and fitted with an insulating cover).
Touching the uncovered high–voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator’s panel or screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the backed up data will be lost.
Refer to the maintenance manual or maintenance section of the operator’s manual for details of the battery replacement procedure.
2. Absolute pulse coder battery replacement

Only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high–voltage circuits (marked △ and fitted with an insulating cover).
Touching the uncovered high–voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The absolute pulse coder uses batteries to preserve its absolute position.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator’s panel or screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.
Refer to the FANUC Servo Amplifier α series Maintenance Manual or FANUC Servo Motor β series Maintenance Manual for details of the battery replacement procedure.
3. **Fuse replacement**

Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse.
For this reason, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing a fuse with the cabinet open, be careful not to touch the high-voltage circuits (marked Δ and fitted with an insulating cover).
Touching an uncovered high-voltage circuit presents an extremely dangerous electric shock hazard.
Refer to the Maintenance Manual for details of the fuse replacement procedure.
# Table of Contents

SAFETY PRECAUTIONS ............................................................................... s–1

I. GENERAL

1. GENERAL .......................................................................................... 3

2. LIST OF SPECIFICATIONS ................................................................. 5

II. NC FUNCTION

PREFACE ........................................................................................................ 15

1. CONTROLLED AXES ............................................................................. 16
   1.1 CONTROLLED AXES ......................................................................... 17
   1.2 AXIS NAMES .................................................................................... 18
   1.3 INCREMENT SYSTEM .......................................................................... 19
       1.3.1 Input Unit (10 Times) ................................................................. 19
   1.4 MAXIMUM STROKE ........................................................................... 19

2. PREPARATORY FUNCTIONS .................................................................. 20

3. INTERPOLATION FUNCTIONS ............................................................... 22
   3.1 POSITIONING (G00) .......................................................................... 23
   3.2 LINEAR INTERPOLATION (G01) ......................................................... 24
   3.3 CIRCULAR INTERPOLATION (G02, G03) .......................................... 25
   3.4 POLAR COORDINATE INTERPOLATION (G12.1, G13.1) .................. 26

4. FEED FUNCTIONS .................................................................................... 27
   4.1 RAPID TRAVERSE ............................................................................ 28
   4.2 CUTTING FEED RATE ....................................................................... 29
       4.2.1 Tangential Speed Constant Control ........................................... 29
       4.2.2 Cutting Feed Rate Clamp ......................................................... 29
       4.2.3 Per Minute Feed (G94) ............................................................... 29
       4.2.4 Per Revolution Feed (G95) ....................................................... 30
   4.3 OVERRIDE ....................................................................................... 31
       4.3.1 Feed Rate Override ................................................................... 31
       4.3.2 Rapid Traverse Override ........................................................... 31
       4.3.3 Override Cancel ........................................................................ 31
       4.3.4 Jog Override ................................................................................ 31
   4.4 AUTOMATIC ACCELERATION/DECELERATION .................................. 32
   4.5 RAPID TRAVERSE BELL–SHAPED ACCELERATION/DECELERATION . . . 33
   4.6 LINEAR ACCELERATION/DECELERATION AFTER CUTTING FEED INTERPOLATION . . . 34
4.7 BELL-SHAPED ACCELERATION/DECELERATION AFTER CUTTING FEED INTERPOLATION .......................... 35
4.8 LINEAR ACCELERATION/DECELERATION BEFORE CUTTING FEED INTERPOLATION .................. 36
4.9 EXACT STOP (G09) ................................................. 37
4.10 DWELL (G04) ..................................................... 37

5. REFERENCE POSITION ........................................... 38
5.1 MANUAL REFERENCE POSITION RETURN ........................................... 39
5.2 SETTING THE REFERENCE POSITION WITHOUT DOGS .................................................. 39
5.3 AUTOMATIC REFERENCE POSITION RETURN (G28, G29) .................................................. 40
5.4 REFERENCE POSITION RETURN CHECK (G27) .................................................. 41
5.5 2ND AND 3RD REFERENCE POSITION RETURN (G30) .................................................. 41
5.6 RETURN SIGNAL .......................................................... 42
5.7 BUTT-TYPE REFERENCE POSITION SETTING .................................................. 42
5.8 REFERENCE POSITION EXTERNAL SETTING .................................................. 43
5.9 LINEAR SCALE WITH ABSOLUTE ADDRESSING REFERENCE MARKS .................................. 43

6. COORDINATE SYSTEMS ........................................... 44
6.1 MACHINE COORDINATE SYSTEM .................................................. 45
6.2 WORKPIECE COORDINATE SYSTEM .................................................. 46
6.2.1 Setting a Workpiece Coordinate System (Using G92) .................................................. 46
6.2.2 Automatic Coordinate System Setting .................................................. 46
6.3 PLANE SELECTION (G17, G18, G19) .................................................. 47

7. COORDINATE VALUE AND DIMENSION ........................................... 48
7.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91) .................................................. 49
7.2 INCH/METRIC CONVERSION (G20, G21) .................................................. 50
7.3 DECIMAL POINT INPUT/POCKET CALCULATOR TYPE DECIMAL POINT INPUT .................................................. 50
7.4 LINEAR AXIS AND ROTATION AXIS .................................................. 51
7.5 ROTATION AXIS ROLL-OVER FUNCTION .................................................. 51

8. SPINDLE FUNCTIONS (ONLY FOR Power Mate–D) ........................................... 52
8.1 S CODE OUTPUT .................................................. 53
8.2 SPINDLE SPEED ANALOG OUTPUT (S ANALOG OUTPUT) .................................................. 53
8.3 SPINDLE SPEED SERIAL OUTPUT (S SERIAL OUTPUT) .................................................. 53
8.4 SPINDLE OUTPUT CONTROL BY THE PMC .................................................. 53
8.5 CONSTANT SURFACE SPEED CONTROL .................................................. 54
8.6 SPINDLE OVERRIDE .................................................. 54
8.7 SPINDLE ORIENTATION .................................................. 54
8.8 SPINDLE OUTPUT SWITCHING .................................................. 54

9. TOOL FUNCTIONS .................................................. 55
9.1 T CODE OUTPUT .................................................. 56

10. MISCELLANEOUS FUNCTIONS ........................................... 57
10.1 MISCELLANEOUS FUNCTIONS .................................................. 58
10.2 1–BLOCK PLURAL M COMMAND .................................................. 58
10.3 HIGH–SPEED M/S/T INTERFACE .................................................. 59
# 11. PROGRAM CONFIGURATION

11.1 PROGRAM NUMBER .................................................. 62
11.2 PROGRAM NAME .................................................. 62
11.3 MAIN PROGRAM .................................................. 62
11.4 SUB PROGRAM .................................................. 63
11.5 SEQUENCE NUMBER ................................................ 63
11.6 TAPE CODES ..................................................... 63
11.7 BASIC ADDRESSES AND COMMAND VALUE RANGE .............. 64
11.8 TAPE FORMAT ..................................................... 65
11.9 LABEL SKIP ....................................................... 65
11.10 CONTROL–IN/CONTROL–OUT ...................................... 65
11.11 OPTIONAL BLOCK SKIP .......................................... 65
11.12 ADDITIONAL OPTIONAL BLOCK SKIP .......................... 65
11.13 TAPE HORIZONTAL (TH) PARITY CHECK AND TAPE VERTICAL (TV) PARITY CHECK . . . 65

# 12. FUNCTIONS TO SIMPLIFY PROGRAMMING ......................... 66

12.1 CANNED CYCLES (G73, G74, G76, G80 – G89, G98, G99) (ONLY FOR Power Mate–D) .... 67
12.1.1 Canned Cycle Cancel (G80) .................................... 73
12.2 RIGID TAP (ONLY FOR Power Mate–D) .......................... 74
12.2.1 Rigid Tapping (G84) .......................................... 76
12.2.2 Left–handed Rigid Tapping Cycle (G74) ....................... 77
12.2.3 Peck Rigid Tapping Cycle (G84 or G74) ....................... 78

# 13. TOOL COMPENSATION FUNCTION ..................................... 79

13.1 TOOL LENGTH COMPENSATION (G43, G44, G49) ............... 80
13.2 TOOL COMPENSATION MEMORY ................................... 81
13.2.1 Tool Compensation Memory ................................... 81
13.3 NUMBER OF TOOL OFFSETS ...................................... 81
13.4 CHANGING OF TOOL OFFSET AMOUNT (PROGRAMMABLE DATA INPUT) (G10) ........... 81

# 14. ACCURACY COMPENSATION FUNCTION ............................. 82

14.1 STORED PITCH ERROR COMPENSATION (ONLY FOR Power Mate–D) ............. 83
14.2 BACKLASH COMPENSATION ....................................... 83
14.3 PROGRAMMABLE PARAMETER ENTRY (G10, G11) .................. 83

# 15. MEASUREMENT FUNCTIONS ......................................... 84

15.1 SKIP FUNCTION (G31) ............................................ 85
15.2 MULTI–STEP SKIP FUNCTION (G31 P1 – G31 P4) .................. 86
15.3 TOOL LENGTH MEASUREMENT .................................... 86

# 16. CUSTOM MACRO ..................................................... 87

16.1 CUSTOM MACRO .................................................. 88
16.2 INTERRUPTION TYPE CUSTOM MACRO ............................ 94
16.3 PATTERN DATA INPUT ............................................. 95
16.4 MACRO EXECUTER FUNCTION .................................... 96
16.5 C LANGUAGE EXECUTER FUNCTION ............................... 97
# TABLE OF CONTENTS

## 17. AXES CONTROL ................. 98

17.1 FOLLOW UP FUNCTION ......... 99
17.2 MECHANICAL HANDLE FEED .... 99
17.3 SERVO OFF ................. 99
17.4 MIRROR IMAGE ............... 99
17.5 CONTROL AXIS DETACH ........ 100
17.6 SIMPLE SYNCHRONOUS CONTROL ....... 100
17.7 TORQUE LIMIT FUNCTION .......... 101
17.8 FEED STOP ................. 102
17.9 AXIS CONTROL WITH PMC ............ 103
17.10 TANDEM CONTROL ............ 104

## 18. MANUAL OPERATION ............ 105

18.1 MANUAL FEED ............... 106
18.2 INCREMENTAL FEED ............. 106
18.3 MANUAL HANDLE FEED (1ST) ........ 106
18.4 MANUAL HANDLE FEED (2ND, 3RD) ....... 107
18.5 HANDLE FEED IN THE SAME MODE AS FOR JOGGING ............ 107
18.6 MANUAL ABSOLUTE ON/OFF ....... 107

## 19. AUTOMATIC OPERATION ........ 108

19.1 OPERATION MODE .............. 109
  19.1.1 DNC Operation (Only for Power Mate i–D) ............ 109
  19.1.2 Memory Operation ............ 109
  19.1.3 MDI Operation ............... 109
19.2 SELECTION OF EXECUTION PROGRAMS ............ 110
  19.2.1 Program Number Search ............ 110
  19.2.2 Sequence Number Search ............ 110
  19.2.3 Rewind ................. 110
  19.2.4 External Workpiece Number Search ............ 110
19.3 ACTIVATION OF AUTOMATIC OPERATION ............ 111
  19.3.1 Cycle Start ................. 111
19.4 EXECUTION OF AUTOMATIC OPERATION ............ 111
  19.4.1 Buffer Register ............... 111
19.5 AUTOMATIC OPERATION STOP ............ 112
  19.5.1 Program Stop (M00, M01) ............ 112
  19.5.2 Program End (M02, M30) ............ 112
  19.5.3 Feed Hold ................. 112
  19.5.4 Reset ................. 112
19.6 MANUAL INTERRUPTION DURING AUTOMATIC OPERATION ............ 112
  19.6.1 Handle Interruption ............ 112
19.7 RETRACTION FOR TAPPING ............ 113
19.8 RIGID TAPPING RETURN ............ 113
20. PROGRAM TEST FUNCTIONS ........................................... 114
  20.1 ALL-AXES MACHINE LOCK ........................................ 115
  20.2 AUXILIARY FUNCTION LOCK ...................................... 115
  20.3 DRY RUN .......................................................... 115
  20.4 SINGLE BLOCK ...................................................... 115

21. SETTING AND DISPLAY UNIT ........................................... 116
  21.1 SETTING AND DISPLAY UNIT ........................................ 117
    21.1.1 CRT/MDI ..................................................... 118
    21.1.2 LCD/MDI ..................................................... 119
    21.1.3 Picture Display CRT/MDI ................................... 120
    21.1.4 Separate MDI ................................................ 121
    21.1.5 Separate MDI for Picture Display .......................... 122
    21.1.6 Separate CRT ................................................ 122
    21.1.7 Separate LCD ................................................ 123
    21.1.8 Handy Operator’s Panel .................................... 124
    21.1.9 Detachable LCD/MDI ....................................... 126
    21.1.10 Touch Panel Display Unit ................................ 127
    21.1.11 Personal Computer (Connected via HSSB) .................. 128
    21.1.12 Personal Computer (Connected via RS-232C) .......... 129
  21.2 EXPLANATION OF THE KEYBOARD ................................ 130
    21.2.1 For English Key ............................................. 130
    21.2.2 For Symbol Key ............................................. 132
    21.2.3 Explanation of the Function Keys ......................... 132
    21.2.4 Explanation of the Soft Keys .............................. 133

22. DISPLAYING AND SETTING DATA .................................... 134
  22.1 DISPLAY ........................................................... 135
  22.2 LANGUAGE SELECTION ............................................. 138
  22.3 CLOCK FUNCTION .................................................. 138
  22.4 RUN TIME & PARTS NUMBER DISPLAY ............................. 138
  22.5 SOFTWARE OPERATOR’S PANEL .................................... 139
  22.6 DIRECTORY DISPLAY OF FLOPPY CASSETTE ..................... 141
  22.7 SCREENS FOR SERVO DATA AND SPINDLE DATA .................. 142
    22.7.1 Servo Setting Screen ....................................... 142
    22.7.2 Servo Adjustment Screen .................................. 142
    22.7.3 Spindle Setting Screen .................................... 143
    22.7.4 Spindle Adjustment Screen ................................. 143
  22.8 SYSTEM CONFIGURATION DISPLAY FUNCTION .................... 144
  22.9 HELP FUNCTION .................................................... 146
  22.10 DATA PROTECTION KEY .......................................... 148
  22.11 DISPLAYING OPERATION HISTORY ................................. 148
  22.12 PERIODIC MAINTENANCE SCREEN ................................ 149
  22.13 MAINTENANCE INFORMATION SCREEN ............................. 149
# TABLE OF CONTENTS

## 23. PART PROGRAM STORAGE AND EDITING ................................................. 150

- 23.1 FOREGROUND EDITING .............................................................. 151
- 23.2 BACKGROUND EDITING .............................................................. 151
- 23.3 EXPANDED PART PROGRAM EDITING .......................................... 152
- 23.4 NUMBER OF REGISTERED PROGRAMS ........................................... 152
- 23.5 PART PROGRAM STORAGE LENGTH .............................................. 152
- 23.6 PLAY BACK ............................................................................... 152
- 23.7 EXTERNAL CONTROL OF I/O DEVICE ......................................... 152

## 24. DIAGNOSIS FUNCTIONS ................................................................. 153

- 24.1 SELF DIAGNOSIS FUNCTIONS ................................................... 154

## 25. DATA INPUT/OUTPUT ................................................................. 155

- 25.1 RS-232-C INTERFACES .............................................................. 156
- 25.2 INPUT/OUTPUT DEVICES ........................................................... 156
  - 25.2.1 FANUC Handy File .......................................................... 156
- 25.3 DATA INPUT/OUTPUT USING A MEMORY CARD ............................ 157
  - 25.3.1 Batch Save and Restore Operations based on Memory Cards .... 158
- 25.4 DATA INPUT/OUTPUT FUNCTION BASED ON THE I/O LINK AND DATA INPUT/OUTPUT FUNCTION BASED ON THE I/O LINK .......... 159
- 25.5 POWER MATE CNC MANAGER .................................................... 160
- 25.6 ANALOG INPUT FUNCTION ........................................................ 160

## 26. SAFETY FUNCTIONS ................................................................. 161

- 26.1 EMERGENCY STOP .................................................................... 162
  - 26.1.1 General .............................................................................. 162
  - 26.1.2 Power Mate i ....................................................................... 162
  - 26.1.3 Servo/Spindle .................................................................... 163
- 26.2 OVERTRAVEL FUNCTIONS ....................................................... 165
  - 26.2.1 Overtravel .......................................................................... 165
  - 26.2.2 Stored Stroke Check 1 ........................................................ 165
  - 26.2.3 Externally Setting the Stroke Limit ........................................ 165
- 26.3 INTERLOCK .............................................................................. 166
  - 26.3.1 Interlock per Axis ............................................................... 166
  - 26.3.2 All Axes Interlock ............................................................... 166
  - 26.3.3 Interlock for Each Axis Direction ........................................ 166
- 26.4 ABNORMAL LOAD DETECTION ............................................... 166

## 27. STATUS OUTPUT ........................................................................ 167

- 27.1 NC READY SIGNAL ................................................................. 168
- 27.2 SERVO READY SIGNAL ........................................................... 168
- 27.3 REWINDING SIGNAL ............................................................... 168
- 27.4 ALARM SIGNAL ......................................................................... 168
- 27.5 DISTRIBUTION END SIGNAL ................................................... 168
- 27.6 AUTOMATIC OPERATION SIGNAL ........................................... 168
- 27.7 AUTOMATIC OPERATION START SIGNAL .................................... 168
## TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>27.8</td>
<td>FEED HOLD SIGNAL</td>
<td>168</td>
</tr>
<tr>
<td>27.9</td>
<td>RESET SIGNAL</td>
<td>168</td>
</tr>
<tr>
<td>27.10</td>
<td>IN–POSITION SIGNAL</td>
<td>168</td>
</tr>
<tr>
<td>27.11</td>
<td>MOVE SIGNAL</td>
<td>168</td>
</tr>
<tr>
<td>27.12</td>
<td>AXIS MOVE DIRECTION SIGNAL</td>
<td>169</td>
</tr>
<tr>
<td>27.13</td>
<td>RAPID TRAVERSING SIGNAL</td>
<td>169</td>
</tr>
<tr>
<td>27.14</td>
<td>TAPPING SIGNAL (ONLY FOR Power Mate i–D)</td>
<td>169</td>
</tr>
<tr>
<td>27.15</td>
<td>CONSTANT SURFACE SPEED CONTROL SIGNAL</td>
<td>169</td>
</tr>
<tr>
<td>27.16</td>
<td>INCH INPUT SIGNAL</td>
<td>169</td>
</tr>
<tr>
<td>27.17</td>
<td>DI STATUS OUTPUT SIGNAL</td>
<td>169</td>
</tr>
<tr>
<td>27.18</td>
<td>POSITION SWITCH FUNCTION</td>
<td>169</td>
</tr>
<tr>
<td>28.1</td>
<td>EXTERNAL PROGRAM NUMBER SEARCH</td>
<td>171</td>
</tr>
<tr>
<td>28.2</td>
<td>EXTERNAL ALARM MESSAGE</td>
<td>171</td>
</tr>
<tr>
<td>28.3</td>
<td>EXTERNAL OPERATOR’S MESSAGE</td>
<td>171</td>
</tr>
<tr>
<td>29.1</td>
<td>EXTERNAL DATA INPUT</td>
<td>170</td>
</tr>
<tr>
<td>30.1</td>
<td>KEY INPUT FROM PMC (EXTERNAL KEY INPUT)</td>
<td>172</td>
</tr>
<tr>
<td>30.1</td>
<td>PERSONAL COMPUTER FUNCTION</td>
<td>173</td>
</tr>
<tr>
<td>30.1</td>
<td>HIGH–SPEED SERIAL BUS (HSSB)</td>
<td>174</td>
</tr>
</tbody>
</table>

### APPENDIX

<table>
<thead>
<tr>
<th>Part</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>RANGE OF COMMAND VALUE</td>
<td>177</td>
</tr>
<tr>
<td>B</td>
<td>LIST OF FUNCTIONS AND TAPE FORMAT</td>
<td>179</td>
</tr>
<tr>
<td>C</td>
<td>TAPE CODE LIST</td>
<td>181</td>
</tr>
<tr>
<td>D</td>
<td>EXTERNAL DIMENSIONS OF EACH UNIT</td>
<td>184</td>
</tr>
</tbody>
</table>
I. GENERAL
The amount of cabling in the electrical unit of the machine can be significantly reduced by using a high-speed serial servo bus, which connects the CNC control unit and multiple servo amplifiers by a single optical fiber cable. Another innovation which simplifies the electrical unit of the machine tool is the use of compact distributed I/O modules, which can be separately mounted on the machine operator’s panel and control panel. Thus, the electrical sections of machine tools and industrial machines can be simplified.

This manual describes the following models and may use the following abbreviations.

<table>
<thead>
<tr>
<th>Model name</th>
<th>Abbreviation</th>
</tr>
</thead>
<tbody>
<tr>
<td>FANUC Power Mate i–MODEL D</td>
<td>Power Mate i–D</td>
</tr>
<tr>
<td>FANUC Power Mate i–MODEL H</td>
<td>Power Mate i–H</td>
</tr>
</tbody>
</table>

**Related manuals**

The following table lists the manuals related to the FANUC Power Mate i–MODEL D/H. This manual is indicated by an asterisk (*).

<table>
<thead>
<tr>
<th>Manuals Related to the Power Mate i–MODEL D/H</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Manual name</strong></td>
</tr>
<tr>
<td>Descriptions</td>
</tr>
<tr>
<td>Connection Manual (Hardware)</td>
</tr>
<tr>
<td>Connection Manual (Function)</td>
</tr>
<tr>
<td>Operator’s Manual</td>
</tr>
<tr>
<td>Maintenance Manual</td>
</tr>
<tr>
<td>Parameter Manual</td>
</tr>
<tr>
<td>Macro Compiler/Macro Executor, Programming Manual</td>
</tr>
</tbody>
</table>
### Manuals related to the \(\alpha/\beta\)-series servo motor

#### Manual related to the \(\alpha/\beta\)-series servo motor

<table>
<thead>
<tr>
<th>Manual name</th>
<th>Order No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>FANUC AC Servo Motor (\alpha) series, Descriptions</td>
<td>B–65142E</td>
</tr>
<tr>
<td>FANUC AC Servo Motor (\alpha) series, Parameter Manual</td>
<td>B–65150E</td>
</tr>
<tr>
<td>FANUC AC Spindle Motor (\alpha) series, Descriptions</td>
<td>B–65152E</td>
</tr>
<tr>
<td>FANUC AC Spindle Motor (\alpha) series, Parameter Manual</td>
<td>B–65160E</td>
</tr>
<tr>
<td>FANUC Servo Amplifier (\alpha) series, Descriptions</td>
<td>B–65162E</td>
</tr>
<tr>
<td>FANUC Control Motor (\alpha) series, Maintenance Manual</td>
<td>B–65165E</td>
</tr>
</tbody>
</table>
# 2 LIST OF SPECIFICATIONS

<table>
<thead>
<tr>
<th>Item</th>
<th>Specifications</th>
<th>iD</th>
<th>iD2</th>
<th>iH</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Axis control</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Maximum controlled axes</td>
<td>2 axes</td>
<td>☆</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td></td>
<td>1 axis per path</td>
<td>−</td>
<td>○</td>
<td>−</td>
</tr>
<tr>
<td></td>
<td>8 axes</td>
<td>−</td>
<td>−</td>
<td>●</td>
</tr>
<tr>
<td>Controlled axes</td>
<td>1 axis</td>
<td>○</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td></td>
<td>1 axis per path</td>
<td>−</td>
<td>○</td>
<td>−</td>
</tr>
<tr>
<td></td>
<td>2 axes</td>
<td>−</td>
<td>−</td>
<td>○</td>
</tr>
<tr>
<td>Simultaneously controlled axes</td>
<td>Simultaneously 1 axis</td>
<td>○</td>
<td>○</td>
<td>−</td>
</tr>
<tr>
<td></td>
<td>Simultaneously 4 axes</td>
<td>−</td>
<td>−</td>
<td>○</td>
</tr>
<tr>
<td>Simultaneously controlled axes expansion</td>
<td>2 axes</td>
<td>☆</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td>Controlled paths</td>
<td>1 paths</td>
<td>○</td>
<td>−</td>
<td>○</td>
</tr>
<tr>
<td></td>
<td>2 paths</td>
<td>−</td>
<td>○</td>
<td>−</td>
</tr>
<tr>
<td>Axis control by PMC</td>
<td>Up to 2 axes</td>
<td>○</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td></td>
<td>Up to 8 axes</td>
<td>−</td>
<td>−</td>
<td>○</td>
</tr>
<tr>
<td>Axis name</td>
<td>X, Y, Z, U, V, W, A, B, C</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Simple synchronous control</td>
<td>1 set</td>
<td>☆</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td></td>
<td>4 sets</td>
<td>−</td>
<td>−</td>
<td>☆</td>
</tr>
<tr>
<td>Tandem control</td>
<td></td>
<td>☆</td>
<td>−</td>
<td>☆</td>
</tr>
<tr>
<td>Torque control</td>
<td></td>
<td>○</td>
<td>−</td>
<td>○</td>
</tr>
<tr>
<td>Controlled axis detach</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Torque limit control</td>
<td>Torque control from the PMC</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td></td>
<td>Torque control by the DI signals</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td></td>
<td>Torque control by the parameter input</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Minimum input increment</td>
<td>0.001mm, 0.001deg, 0.0001inch</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Increment system 1/10</td>
<td>0.0001mm, 0.0001deg, 0.00001inch</td>
<td>○</td>
<td>○</td>
<td>−</td>
</tr>
<tr>
<td>Minimum input increment 10µm</td>
<td>0.01mm, 0.01deg, 0.001inch</td>
<td>−</td>
<td>−</td>
<td>○</td>
</tr>
<tr>
<td>Flexible feed gear</td>
<td>Optional DMR</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
</tbody>
</table>

Legend

○ : Standard
● : Standard option
☆ : Option
★ : Function included in another option
− : Unused
<table>
<thead>
<tr>
<th>Item</th>
<th>Specifications</th>
<th>iD</th>
<th>iD2</th>
<th>iH</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dual position feedback</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fine acceleration/deceleration</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HRV control</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Inch/metric switching</td>
<td>G20, G21</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interlock</td>
<td>All axes</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>each axis</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>each axial direction</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>High-speed interlock</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Machine lock</td>
<td>All axes</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Emergency stop</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Overtravel</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stored stroke check 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>External stroke limit setting</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mirror image</td>
<td>Each axis</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Follow-up</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Servo–off/mechanical handle feed</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Backlash compensation</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stored pitch error compensation</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Position switch</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Unexpected disturbance torque detection function</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Operation**

<table>
<thead>
<tr>
<th>Item</th>
<th>Specifications</th>
<th>iD</th>
<th>iD2</th>
<th>iH</th>
</tr>
</thead>
<tbody>
<tr>
<td>Automatic operation (memory)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DNC operation</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MDI operation</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Program number search</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Sequence number search</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>External workpiece number search</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rewinding</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Buffer register</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Dry run</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Single block</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Jog feed (manual continuous feed)</td>
<td>Jog override is enabled.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Jog feed (manual rapid traverse)</td>
<td>Rapid traverse override is enabled.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Manual reference position return</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reference position return setting without dog (grid method)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Butt–type reference position return setting</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Item</td>
<td>Specifications</td>
<td>iD</td>
<td>iD2</td>
<td>iH</td>
</tr>
<tr>
<td>-----------------------------------------------------------</td>
<td>---------------------------------</td>
<td>----</td>
<td>-----</td>
<td>----</td>
</tr>
<tr>
<td>External setting function of reference position</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Manual handle feed</td>
<td>1 unit</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1 unit per path</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2 or 3 units</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Manual handle feed magnification</td>
<td>$\times 1, \times 10, \times m, \times n$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Manual handle interrupt</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Incremental feed</td>
<td>$\times 1, \times 10, \times 100, \times 1000$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Jogging/handle feed mode</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>External pulse input function</td>
<td>Position coder</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interpolation functions</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Positioning (Non linear interpolation type)</td>
<td>G00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Positioning (Linear interpolation type)</td>
<td>G00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Exact stop mode</td>
<td>G09</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>G01</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Circular interpolation</td>
<td>G02, G03</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Dwell</td>
<td>For a specified number of seconds</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>For a specified number of rotations</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Polar coordinate interpolation</td>
<td>G12.1, G13.1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Skip function</td>
<td>G31</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Skip function from PMC</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Dwell skip function</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Multi–step skip function</td>
<td>G31 P1 to P4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reference position return</td>
<td>G28</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Return from reference position</td>
<td>G29</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reference position return check</td>
<td>G27</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2nd/3rd reference position return</td>
<td>G30</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reference position retract function</td>
<td>Signals RTN1n, RTN2n, RTN3n</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Feed functions</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rapid traverse</td>
<td>Up to 240 m/min (1 µm)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Up to 100 m/min (0.1 µm)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rapid traverse override</td>
<td>F0, 25, 50, 100%</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Feed per minute</td>
<td>G94</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Feed per rotation</td>
<td>G95</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Constant tangential speed control</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cutting feedrate clamp</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Automatic acceleration/deceleration</td>
<td>Rapid traverse: Linear</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Cutting feed: Exponential</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Item</td>
<td>Specifications</td>
<td>iD</td>
<td>iD2</td>
<td>iH</td>
</tr>
<tr>
<td>----------------------------------------------------------------------</td>
<td>---------------------------------------</td>
<td>----</td>
<td>-----</td>
<td>----</td>
</tr>
<tr>
<td>Rapid traverse bell–shaped acceleration/deceleration</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Linear acceleration/deceleration after cutting feed interpolation</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Bell–shaped acceleration/deceleration after cutting feed interpolation</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Linear acceleration/deceleration before cutting feed interpolation</td>
<td>Feed per minute only</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Feedrate override</td>
<td>0 to 254%</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Jog override</td>
<td>0 to 655.34%</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Override cancel</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Rapid traverse over lap</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Variable in–position</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Feed stop</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Program input</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Tape code</td>
<td>Automatic recognition of EIA and ISO</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Label skip</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Parity check</td>
<td>Horizontal parity, vertical parity</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Control in/out</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Optional block skip</td>
<td>9 blocks</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Maximum value</td>
<td>± with 8 digits</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Program number</td>
<td>O with 4 digits</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td></td>
<td>O with 8 digits</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Sequence number</td>
<td>N with 5 digits</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Absolute/incremental programming</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Decimal point input, pocket calculator type decimal point input</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Input unit (10 times)</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Plane selection</td>
<td>G17, G18, G19</td>
<td>○</td>
<td>–</td>
<td>○</td>
</tr>
<tr>
<td>Rotary axis designation</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Rotary axis roll–over</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Coordinate system setting</td>
<td>G92</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Automatic coordinate system setting</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Manual absolute on/off</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Circular interpolation by R programming</td>
<td></td>
<td>○</td>
<td>–</td>
<td>○</td>
</tr>
<tr>
<td>Programmable data input</td>
<td>G10</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Subprogram call</td>
<td>M98, M99 (4 levels of nesting)</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Custom macro B</td>
<td></td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Custom macro common variables</td>
<td>#100 to #199, #500 to #999</td>
<td>○</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>Item</td>
<td>Specifications</td>
<td>iD</td>
<td>iD2</td>
<td>iH</td>
</tr>
<tr>
<td>------------------------------------------------</td>
<td>----------------------------------------------------</td>
<td>----</td>
<td>-----</td>
<td>----</td>
</tr>
<tr>
<td>Pattern data input</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interrupt–type custom macro</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Miscellaneous/spindle functions</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Miscellaneous function</td>
<td>M with 8 digits</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Miscellaneous function lock</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>High–speed M/S/T/B interface</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Multiple miscellaneous–function commands</td>
<td>5 pieces</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Spindle function</td>
<td>S with 5 digits, binary output</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Spindle serial output</td>
<td>S with 5 digits, serial output (1 per path in iD2)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Spindle analog output</td>
<td>S with 5 digits, analog output</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Constant surface speed control</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Spindle output control by PMC</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Spindle override</td>
<td>0 to 254%</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Spindle orientation</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Spindle output switching</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Return to R point</td>
<td>G99</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Return to initial point</td>
<td>G98</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Canned cycle</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rigid tapping</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interruption function for rigid tapping</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interruption function for tapping</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool functions, tool compensation functions</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool function</td>
<td>T with 8 digits</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool compensation data</td>
<td>± with 6 digits, 99 items</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool offset memory A</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool length compensation</td>
<td>G43, G44, G49</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool length measurement</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Editing</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Part program storage length</td>
<td>10m</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20m</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>40m</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>80m</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>160m</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
## 2. LIST OF SPECIFICATIONS

### GENERAL

<table>
<thead>
<tr>
<th>Item</th>
<th>Specifications</th>
<th>iD</th>
<th>iD2</th>
<th>iH</th>
</tr>
</thead>
<tbody>
<tr>
<td>Registered programs</td>
<td></td>
<td>63</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>125</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>200</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>400</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Part program editing</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Program protection</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Background editing</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Expanded part program editing</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Playback</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Setting, display

<table>
<thead>
<tr>
<th>Item</th>
<th>Specifications</th>
<th>iD</th>
<th>iD2</th>
<th>iH</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status display</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Clock function</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Current position display</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Program display</td>
<td>31–character program name</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Parameter setting display</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Self–diagnosis function</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Alarm display</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Alarm history display</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Operation history display</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Help function</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Run time and parts number display</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Actual speed display</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Display of actual spindle speed</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Display of T code</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Floppy Cassette directory display</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Servo adjustment screen</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Spindle adjustment screen</td>
<td>Serial output only</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hardware/software system configuration display</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Periodic maintenance screen</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Maintenance information display</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Software operator’s panel</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>General–purpose switch on software operator’s panel</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Display language</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>English</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Japanese</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>German/French</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Italian</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Spanish</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Item</td>
<td>Specifications</td>
<td>iD</td>
<td>iD2</td>
<td>iH</td>
</tr>
<tr>
<td>--------------------------------------------------</td>
<td>---------------------------------------</td>
<td>----</td>
<td>-----</td>
<td>----</td>
</tr>
<tr>
<td>Data protection key</td>
<td>4 types</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Data input/output</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reader/punch interface control 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reader/punch interface control 2</td>
<td>Data line only (Function restrictions)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>External I/O device control</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>External message</td>
<td>Alarm and message</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>External program number search</td>
<td>1 to 9999</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>External key input</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>External workpiece number search</td>
<td>0 to 31</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Input/output by memory card</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PCMCIA card input/output</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Power mate CNC manager</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Others</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Acceleration/deceleration signal</td>
<td>SUP</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Distribution signal</td>
<td>IPL</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Servo position deviation motor signal</td>
<td>SVER</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Internal DO over current alarm signal</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fan 1 or 2 alarm signal</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Alarm classification signals output</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CRT/MDI</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CRT/MDI sharing</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Operation package with DPL/MDI</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Handy operator’s panel</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HSSB</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Basic operation package</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Macro executor</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>C language executor</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DI/DO translation by FANUC I/O Link Master function</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DI/DO translation by FANUC I/O Link Slave function</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Linear scale interface with absolute addressing reference mark</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Analog interface</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FANUC I/O Link–II</td>
<td>Slave function</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Internal DI/DO</td>
<td>DI : 32 points, DO : 24 points</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7–segment LED</td>
<td>Alarm and status display</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rotary switch</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Item** | **Specifications** | **iD** | **iD2** | **iH**
--- | --- | --- | --- | ---
**PMC–SB5**
  Ladder monitor display
    Basic instruction: 0.085μsec/step
      5000 steps | ○ | ○ | ○
      12000 steps | ★ | ★ | ★
      16000 steps | ★ | ★ | ★
      24000 steps | ★ | ★ | ★
**PMC–SB6**
  Ladder monitor display
    Basic instruction: 0.085μsec/step
    Step sequence function
      5000 steps | ★ | ★ | ★
      12000 steps | ★ | ★ | ★
      16000 steps | ★ | ★ | ★
      24000 steps | ★ | ★ | ★
      32000 steps | ★ | ★ | ★
**Dynamic ladder diagram display**
    Not available for DPL/MDI operation package or handy operator's panel | ○ | ○ | ○
**Ladder editing (ladder diagram)**
    Ladder diagram editing card is necessary. | ★ | ★ | ★
**Area signal (PMC function command)** | | ○ | ○ | ○
II. NC FUNCTION
This part describes the functions that can be performed on all models. For the functions available with each model, see the list of specifications in Part I.
1. CONTROLLED AXES

CONTROLLED AXES
### 1.1 CONTROLLED AXES

#### Power Mate i–D

<table>
<thead>
<tr>
<th>Item</th>
<th>Power Mate i–D</th>
<th>Power Mate i–D (two–path control)</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of basic controlled axes</td>
<td>1 axis</td>
<td>1 axis for each path (2 axes in total)</td>
</tr>
<tr>
<td>Controlled axes expansion (total)</td>
<td>1 axis (2 axes in total)</td>
<td>1 axis</td>
</tr>
<tr>
<td>Basic simultaneously controlled axes</td>
<td>1 axis</td>
<td>1 axis for each path (2 axes in total)</td>
</tr>
<tr>
<td>Simultaneously controlled axes expansion (total)</td>
<td>2 axes</td>
<td></td>
</tr>
</tbody>
</table>

**NOTE**

The number of simultaneously controllable axes for manual operation jog feed, manual reference position return, or manual rapid traverse is 1 or 2 (when Power Mate i–D is one–path control) (1 when bit 0 (JAX) of parameter 1002 is set to 0 and 2 when it is set to 1).

#### Power Mate i–H

<table>
<thead>
<tr>
<th>Item</th>
<th>Power Mate i–H</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of basic controlled axes</td>
<td>8 axes</td>
</tr>
<tr>
<td>Controlled axes expansion (total)</td>
<td></td>
</tr>
<tr>
<td>Basic simultaneously controlled axes</td>
<td>4 axes</td>
</tr>
<tr>
<td>Simultaneously controlled axes expansion (total)</td>
<td></td>
</tr>
</tbody>
</table>

**NOTE**

The number of simultaneously controllable axes for manual operation jog feed, manual reference position return, or manual rapid traverse is 1 or 4 (1 when bit 0 (JAX) of parameter 1002 is set to 0 and 4 when it is set to 1).
1. CONTROLLED AXES B–63172EN/01NC FUNCTION

1.2 AXIS NAMES

The user can assign any one of the following nine characters as the axis name: A, B, C, U, V, W, X, Y, and Z. Parameter No.1020 is used to determine the name of each axis. When this parameter is set to 0 or a character other than the valid characters is specified, an axis name from 1 to 8 is assigned by default.

Limitations

• Default axis name
  When a default axis name (1 to 8) is used, operation in the AUTO mode, MDI mode is disabled.

• Duplicate axis names
  If a duplicate axis name is specified in the parameter, operation is enabled only for the axis specified first.
1.3 INCREMENT SYSTEM

<table>
<thead>
<tr>
<th>Name of increment system</th>
<th>Least input increment</th>
<th>Least command increment</th>
<th>Maximum stroke</th>
</tr>
</thead>
<tbody>
<tr>
<td>IS–A</td>
<td>0.01 mm</td>
<td>0.01 mm</td>
<td>999999999 mm</td>
</tr>
<tr>
<td></td>
<td>0.001 inch</td>
<td>0.001 inch</td>
<td>999999.999 inch</td>
</tr>
<tr>
<td></td>
<td>0.01 deg</td>
<td>0.01 deg</td>
<td>99999.9999 deg</td>
</tr>
<tr>
<td>IS–B</td>
<td>0.001 mm</td>
<td>0.001 mm</td>
<td>999999.999 mm</td>
</tr>
<tr>
<td></td>
<td>0.0001 inch</td>
<td>0.0001 inch</td>
<td>99999.9999 inch</td>
</tr>
<tr>
<td></td>
<td>0.001 deg</td>
<td>0.001 deg</td>
<td>99999.9999 deg</td>
</tr>
<tr>
<td>IS–C</td>
<td>0.0001 mm</td>
<td>0.0001 mm</td>
<td>9999.99999 mm</td>
</tr>
<tr>
<td></td>
<td>0.00001 inch</td>
<td>0.00001 inch</td>
<td>999.999999 inch</td>
</tr>
<tr>
<td></td>
<td>0.0001 deg</td>
<td>0.00001 deg</td>
<td>9999.99999 deg</td>
</tr>
</tbody>
</table>

NOTE

IS–A is available for Power Mate i–H.
IS–A is available for Power Mate i–D.

Combined use of the inch system and the metric system is not allowed. There are functions that cannot be used between axes with different unit systems. For the increment system, see the machine tool builder’s manual.

1.3.1 Input Unit (10 Times)

The following least input increments can be set using a parameter:

<table>
<thead>
<tr>
<th>Increment system</th>
<th>Least input increment</th>
</tr>
</thead>
<tbody>
<tr>
<td>IS–B</td>
<td>0.01 mm, 0.01 deg, or 0.001 inch</td>
</tr>
<tr>
<td>IS–C</td>
<td>0.0001 mm, 0.001 deg, or 0.0001 deg</td>
</tr>
</tbody>
</table>

NOTE

1. When the IS–A increment system is used, the minimum input increment cannot be made ten times greater than the minimum command increment.
2. The minimum input increment for inch input is not affected.

1.4 MAXIMUM STROKE

The following table lists the maximum strokes of machine tools that are allowed by the control unit:

Maximum stroke = Least command increment × 99999999

See Sec. 1.3.

NOTE

1. A command that exceeds the maximum stroke is not allowed.
2. The actual stroke varies from one machine to another.
## 2. PREPARATORY FUNCTIONS

### Table 2 G code list (1/2)

<table>
<thead>
<tr>
<th>G code</th>
<th>Group</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>01</td>
<td>Positioning</td>
</tr>
<tr>
<td>G01</td>
<td></td>
<td>Linear interpolation</td>
</tr>
<tr>
<td>G02</td>
<td></td>
<td>Circular interpolation (clockwise)</td>
</tr>
<tr>
<td>G03</td>
<td></td>
<td>Circular interpolation (counterclockwise)</td>
</tr>
<tr>
<td>G04</td>
<td>00</td>
<td>Dwell, Exact stop</td>
</tr>
<tr>
<td>G09</td>
<td></td>
<td>Exact stop</td>
</tr>
<tr>
<td>G10</td>
<td></td>
<td>Data setting</td>
</tr>
<tr>
<td>G11</td>
<td></td>
<td>Data setting mode cancel</td>
</tr>
<tr>
<td>G12.1</td>
<td>21</td>
<td>Polar coordinate interpolation mode</td>
</tr>
<tr>
<td>G13.1</td>
<td></td>
<td>Polar coordinate interpolation cancel mode</td>
</tr>
<tr>
<td>G17</td>
<td>02</td>
<td>XpYp plane selection</td>
</tr>
<tr>
<td>G18</td>
<td></td>
<td>ZpXp plane selection</td>
</tr>
<tr>
<td>G19</td>
<td></td>
<td>YpZp plane selection</td>
</tr>
<tr>
<td>G20</td>
<td>06</td>
<td>Input in inch</td>
</tr>
<tr>
<td>G21</td>
<td></td>
<td>Input in mm</td>
</tr>
<tr>
<td>G27</td>
<td></td>
<td>Reference position return check</td>
</tr>
<tr>
<td>G28</td>
<td>00</td>
<td>Return to reference position</td>
</tr>
<tr>
<td>G29</td>
<td></td>
<td>Return from reference position</td>
</tr>
<tr>
<td>G30</td>
<td></td>
<td>2nd and 3rd reference position return</td>
</tr>
<tr>
<td>G31</td>
<td></td>
<td>Skip function</td>
</tr>
<tr>
<td>G43</td>
<td>08</td>
<td>Tool length compensation + direction</td>
</tr>
<tr>
<td>G44</td>
<td></td>
<td>Tool length compensation – direction</td>
</tr>
<tr>
<td>G49</td>
<td></td>
<td>Tool length compensation cancel</td>
</tr>
<tr>
<td>G65</td>
<td>00</td>
<td>Macro call</td>
</tr>
<tr>
<td>G66</td>
<td>12</td>
<td>Macro modal call</td>
</tr>
<tr>
<td>G67</td>
<td></td>
<td>Macro modal call cancel</td>
</tr>
</tbody>
</table>
## Table 2  G code list (2/2)

<table>
<thead>
<tr>
<th>G code</th>
<th>Group</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G73</td>
<td></td>
<td>Peck drilling cycle</td>
</tr>
<tr>
<td>G74</td>
<td></td>
<td>Counter tapping cycle</td>
</tr>
<tr>
<td>G76</td>
<td></td>
<td>Fine boring</td>
</tr>
<tr>
<td>G80</td>
<td></td>
<td>Canned cycle cancel</td>
</tr>
<tr>
<td>G81</td>
<td></td>
<td>Drilling cycle, spot boring</td>
</tr>
<tr>
<td>G82</td>
<td></td>
<td>Drilling cycle, counter boring</td>
</tr>
<tr>
<td>G83</td>
<td>09</td>
<td>Peck drilling cycle</td>
</tr>
<tr>
<td>G84</td>
<td></td>
<td>Tapping cycle</td>
</tr>
<tr>
<td>G85</td>
<td></td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G86</td>
<td></td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G87</td>
<td></td>
<td>Back boring cycle</td>
</tr>
<tr>
<td>G88</td>
<td></td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G89</td>
<td></td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G90</td>
<td>03</td>
<td>Absolute command</td>
</tr>
<tr>
<td>G91</td>
<td></td>
<td>Increment command</td>
</tr>
<tr>
<td>G92</td>
<td>00</td>
<td>Setting for work coordinate system/Clamp of maximum spindle speed</td>
</tr>
<tr>
<td>G94</td>
<td>05</td>
<td>Feed per minute</td>
</tr>
<tr>
<td>G95</td>
<td></td>
<td>Feed per rotation</td>
</tr>
<tr>
<td>G96</td>
<td>13</td>
<td>Constant surface speed control</td>
</tr>
<tr>
<td>G97</td>
<td></td>
<td>Constant surface speed control cancel</td>
</tr>
</tbody>
</table>

**NOTE**

1. The G codes of group 09, group 10, group 13 and G92 clamp of maximum spindle speed are only for the Power Mate i–D.
2. G02, G03, G12.1, and G13.1 are disabled for the Power Mate i–D (2–path control).
3 INTERPOLATION FUNCTIONS
3.1 POSITIONING (G00)

Positioning is done with each axis separately (Non linear interpolation type positioning).
Either of the following tool paths can be selected according to bit 1 of parameter No. 1401.

- **Non linear interpolation positioning**
  The tool is positioned with the rapid traverse rate for each axis separately. When an absolute command is used, the tool moves at the rapid traverse rate to a specified point in the workpiece coordinate system. When an incremental command is used, the tool moves a specified distance at the rapid traverse rate from the current position. The tool path is normally straight.

- **Linear interpolation positioning**
  The tool path is the same as in linear interpolation (G01). The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.

![Diagram of tool path showing linear and non-linear interpolation]

It is decelerated, to a stop at the end point, and imposition check is performed (checks whether the machine has come to the specified position). The in-position check can be suppressed using a parameter. Width of imposition can be set as a parameter.

**Format**

```
G00 IP_;  Coordinates of a tool movement end point when an absolute command is used
          Tool travel when an incremental command is used
```
Linear interpolation is done with tangential direction feed rate specified by the F code.

Programming example:

```
G01 G90 X200. Z150. F200 ;
```

Format:

```
G01 IP_F_ ;
F: Feedrate
```
Circular interpolation of optional angle from 0° to 360° can be specified.  
G02: Clockwise (CW) circular interpolation  
G03: Counterclockwise (CCW) circular interpolation

Feed rate of the tangential direction takes the speed specified by the F code.  
Planes to perform circular interpolation is specified by G17, G18, G19.  
Circular interpolation can be performed not only on the X, Y, and Z axis but also on the parallel axes of the X, Y, and Z axes.

- G17: Xp–Yp plane  
- G18: Zp–Xp plane  
- G19: Yp–Zp plane

where

- Xp: X axis or its parallel axis  
- Yp: Y axis or its parallel axis  
- Zp: Z axis or its parallel axis

Parameter is set to decide which parallel axis of the X, Y, Z axes to be the additional axis.

**Format**

Arc on the Xp–Yp plane

\[
G17 \begin{cases} 
G02 \\
G03 \\
\end{cases} Xp \_ Yp_ \_ \begin{cases} 
R_ \_ \\
I_ \_ J_ \_ \\
\end{cases} F_ \_; 
\]

Arc on the Zp–Xp plane

\[
G18 \begin{cases} 
G02 \\
G03 \\
\end{cases} Zp \_ Xp_ \_ \begin{cases} 
R_ \_ \\
K_ \_ I_ \_ \\
\end{cases} F_ \_; 
\]

Arc on the Yp–Zp plane

\[
G19 \begin{cases} 
G02 \\
G03 \\
\end{cases} Yp \_ Zp_ \_ \begin{cases} 
R_ \_ \\
J_ \_ K_ \_ \\
\end{cases} F_ \_; 
\]

I_, J_, K_:Distance of the X, Y, Z axes from the start point to the center of the circle

R:Arc radius (For an arc having a central angle of 180° or greater, specify an R value with a minus sign. A complete circumference cannot be specified.)
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 straight line groove is cut on the outer diameter of a workpiece or when a cam shaft is ground.

Whether the polar coordinate interpolation is done or not is commanded by a G code.

These G codes shall be commanded in a single block.

### Format

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G12.1;</td>
<td>Polar coordinate interpolation mode&lt;br&gt;(Polar coordinate interpolation shall be done.)</td>
</tr>
<tr>
<td>G13.1;</td>
<td>Polar coordinate interpolation cancel mode&lt;br&gt;(Polar coordinate interpolation is not done.)</td>
</tr>
</tbody>
</table>

### Explanations

- **Polar coordinate interpolation mode (G12.1)**

  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 G12.1, and a plane (hereinafter 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 made, the command of linear interpolation (G01) and circular interpolation (G02, G03) is possible. And both absolute command (G90) and incremental command (G91) are possible.
  As for feedrate, specify the tangential speed (relative speed between the workpiece and the tool) on the polar coordinate interpolation plane (cartesian coordinate system) with F.

- **Polar coordinate interpolation cancel mode (G13.1)**

  The polar coordinate interpolation cancel mode is obtained by G13.1 command.
4 FEED FUNCTIONS
4.1 RAPID TRAVERSE

Positioning of each axis 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).

<table>
<thead>
<tr>
<th>Least command increment</th>
<th>Rapid traverse rate range</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.001mm, deg</td>
<td>30 to 240000mm/min, deg/min</td>
</tr>
<tr>
<td>0.0001mm, deg</td>
<td>30 to 100000mm/min, deg/min</td>
</tr>
<tr>
<td>0.0001inch</td>
<td>3.0 to 9600.0inch/min</td>
</tr>
<tr>
<td>0.00001inch</td>
<td>3.0 to 4000.0inch/min</td>
</tr>
</tbody>
</table>

**NOTE**

The above feed rates are limits according to the NC’s interpolation capacity. When the whole system is considered, there are also limits according to the servo system. For details, refer to Appendix A.
4.2 CUTTING FEED RATE

Feed rates of linear interpolation (G01), and circular interpolation (G02, G03) are commanded with numbers after the F code.

4.2.1 Tangential Speed Constant Control

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

4.2.2 Cutting Feed Rate Clamp

Cutting feed rate upper limit can be set as parameters. 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.

NOTE
An upper limit is set in mm/min or inch/min.

4.2.3 Per Minute Feed (G94)

With the per minute feed mode G94, tool feed rate per minute is directly commanded by numerical value after F.

<table>
<thead>
<tr>
<th>Least command increment</th>
<th>Cutting feed rate range</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.001mm, deg</td>
<td>1 to 240000mm/min, deg/min</td>
</tr>
<tr>
<td>0.0001mm, deg</td>
<td>1 to 100000mm/min, deg/min</td>
</tr>
<tr>
<td>0.0001inch</td>
<td>0.01 to 9600.0inch/min</td>
</tr>
<tr>
<td>0.00001inch</td>
<td>0.01 to 4000.0inch/min</td>
</tr>
</tbody>
</table>

NOTE
The above feed rates are limits according to the NC’s interpolation capacity. When the whole system is considered, there are also limits according to the servo system. For details, see Appendix A.
4.2.4 Per Revolution Feed (G95)

With the per revolution feed mode G95, tool feed rate per revolution of the spindle is directly commanded by numeral after F. A position coder must be mounted on the Power Mate.

<table>
<thead>
<tr>
<th>Least command increment</th>
<th>Cutting feed rate range</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.001mm, deg</td>
<td>0.01 to 500.00mm/rev, deg/rev</td>
</tr>
<tr>
<td>0.0001mm, deg</td>
<td>0.01 to 500.00mm/rev, deg/rev</td>
</tr>
<tr>
<td>0.0001inch</td>
<td>0.0001 to 9.9999inch/rev</td>
</tr>
<tr>
<td>0.00001inch</td>
<td>0.0001 to 9.9999inch/rev</td>
</tr>
</tbody>
</table>

**NOTE**

The above feed rates are limits according to the NC’s interpolation capacity. When the whole system is considered there are also limits according to the servo system. For details, See Appendix A.
## 4.3 OVERRIDE

### 4.3.1 Feed Rate Override

The per minute feed (G94) and per rotation feed (G95) can be overridden by:

- 0 to 254% (per every 1%).

The feedrate override function is disabled in tapping or other operations for which the override function is prohibited. The feedrate override function is also disabled when the override cancel signal is set to 1.

### 4.3.2 Rapid Traverse Override

Rapid traverse rate can be overridden by:

- F0, 25%, 50%, 100%.

  F0: A constant speed per axis can be set by parameter

  An override of 0% to 100% can be applied in 1% steps using a signal.

### 4.3.3 Override Cancel

Feed rate override can be clamped to 100% by a signal from the machine side.

### 4.3.4 Jog Override

The manual continuous feedrate and incremental feed rate can be overridden by:

- 0% to 655.34% (in steps of 0.01%)
4.4 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 parameter set per axis)
- **Cutting feed**: Exponential acceleration/deceleration
  (time constant is parameter set per axis)
- **Jogging**: Exponential acceleration/deceleration
  (time constant is parameter set per axis)

![Diagram of rapid traverse, jog feed, and feed, dry run](image-url)
The function for rapid traverse bell–shaped acceleration/deceleration increases or decreases the rapid traverse feedrate smoothly. This reduces the shock to the machine system due to changing acceleration when the feedrate is changed. As compared with linear acceleration/deceleration, bell–shaped acceleration/deceleration allows smaller time constants to be set, reducing the time required for acceleration/deceleration.
In the linear acceleration/deceleration, the delay for the command caused by the acceleration/deceleration becomes \(1/2\) compared with that in exponential acceleration/deceleration, substantially reducing the time required for acceleration and deceleration.

Also, the radius direction error in the circular interpolation caused by the acceleration/deceleration is substantially reduced.

\[
\Delta r : \text{Maximum value of radius error (mm)} \\
v : \text{Feedrate (mm/sec)} \\
r : \text{Circular radius (mm)} \\
T_1 : \text{Acceleration/deceleration time constant (sec)} \\
T_2 : \text{Time constant of servo motor (sec)}
\]

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

\[
\Delta r = \left(\frac{1}{2} T_i^2 + \frac{1}{2} T_d^2\right)\frac{V^2}{r} \ldots \text{For exponential acceleration/deceleration}
\]

\[
\Delta r = \left(\frac{1}{24} T_i^2 + \frac{1}{2} T_d^2\right)\frac{V^2}{r} \ldots \text{For linear acceleration/deceleration after cutting feed interpolation}
\]

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.
As shown above in the quadratic curve, it is possible to accelerate and decelerate the cutting feedrate. When the acceleration and deceleration section are connected, the composed curve shapes just like a hanging bell. That is why this kind of acceleration/deceleration is called bell–shaped acceleration/deceleration. Considering a time constant as $T_c$ (time spent to accelerate from feedrate 0 up to commanded feedrate $F$ or time spent to decelerate from commanded feedrate $F$ down to feedrate 0), feedrate accelerates up to $1/2$ of the commanded feedrate ($F/2$) for $1/2$ of the time constant ($T_c/2$). The acceleration/deceleration curve $0A$ shown in the figure above can be expressed by the following equation:

$$f(t) = \frac{2F}{T_c} t^2$$

The curve $AB$ and $0A$ are symmetric with respect to point $A$. The feature of this acceleration/deceleration is that the feedrate change is small near feedrate 0 and the commanded feedrate.
### 4.8 LINEAR ACCELERATION/DECELERATION BEFORE CUTTING FEED INTERPOLATION

In response to the cutting feed command, the feedrate before interpolation, the command feedrate can be directly accelerated/decelerated. This enables a machined shape error caused by the delay of acceleration/deceleration to be eliminated.

- **Exponential acceleration/deceleration after cutting feed interpolation**

  ![Exponential Acceleration/Deceleration Diagram]

- **Linear acceleration/deceleration after cutting feed interpolation**

  ![Linear Acceleration/Deceleration Diagram]

- **Linear acceleration/deceleration before cutting feed interpolation**

  ![Linear Acceleration/Deceleration Diagram]
4.9 EXACT STOP (G09)
Move command in blocks commanded with G09 decelerates at the end point, and in-position check is performed. G09 command is not necessary for deceleration at the end point for positioning (G00) and in-position check is also done automatically. This function is used when sharp edges are required for workpiece corners in cutting feed.

4.10 DWELL (G04)
With the G04 command, shifting to the next block can be delayed. Shifting to the next block can be delayed for the commanded minutes.

Format

Per second dwell

\[
\text{G94 G04}\ \{\text{P}_-\text{X}_0\};
\]

X: Specify a time. (A decimal point can be used.)
P: Specify a time. (A decimal point cannot be used.)

Per revolution dwell

\[
\text{G95 G04}\ \{\text{P}_-\text{X}_0\};
\]

P_ or X_: Spindle rotation angle commanded in rev. (0.001–99999.999 rev)
5 REFERENCE POSITION
Positioning to the reference position can be done by manual operation. With jogging mode (JOG), manual reference position return (ZR) signals, and signal for selecting manual reference position return axis (±J1 to ±J8) on, the tool the machine is turned on, it decelerates, and when it is turned off again, it stops at the first grid point, and reference position return end signal is output. This point is the reference position. By performing manual reference position return, the machine coordinate system and the work coordinate system is established.

There is only one method available to perform manual reference point return:

In the grid method, a certain grid of the position detection is appointed as the reference position. The reference position position can be shifted by the grid shift function.

This function moves the machine to around the reference position set for each axis in the manual continuous feed mode. Then it sets the reference position for the machine in the manual reference position return mode without the deceleration signal for reference position return. With this function, the machine reference position can be set at a given position without the deceleration signal for reference position return.

1. Place the machine in the manual continuous feed mode, and perform positioning to a position near but not exceeding the reference position from reference position return direction (setting by parameter).

2. Enter the manual reference position return mode, then input the feed axis direction select signal (+) or (–) for the axis.

3. Positioning is made at the grid point located nearest from the current point to reference position return direction. This point is recorded as the reference position. If the absolute-position detector is provided, the set reference position is retained after the power is turned off. In this case, when the power is turned on again, there is no need for setting the reference position again.

1. In reference position return mode after reference position establishment, when the feed axis selection signal for the axis is set to 1, reference position return operation is performed by rapid traverse, regardless of the direction of the feed axis direction selection signal.
5.3 AUTOMATIC REFERENCE POSITION RETURN (G28, G29)

- Return to reference position (G28)

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

```
G28 IP_;  
  IP : Command intermediate point
```

- Return from reference position (G29)

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

```
G29 IP_;  
  IP : Specify a return destination position.
```

Example of use of G28 and G29
5.4 REFERENCE POSITION RETURN CHECK (G27)

This function is used to check whether the reference position return command was performed correctly. When G27 is commanded, the commanded axis is positioned to the specified position, reference position return end signal is output if reference position return is performed to the correct position, and alarm arises it is not positioned correctly to the reference point. This function is available after power is turned on an reference point return is performed.

Format

\[
\text{G27 IP} _\_ ;
\]

IP : Command reference position

5.5 2ND AND 3RD REFERENCE POSITION RETURN (G30)

With the G30 command, the commanded axis is positioned to the 2nd or 3rd reference position, via the commanded point. 2nd or 3rd reference position return end signal is output when positioning ends. Set the 2nd and 3rd reference position position as parameters. This function is available after power is turned on and reference position return is performed.

G29 can be used to return from the 2nd and 3rd reference point (same as reference position return, G28)

Format

\[
\text{G30 } \left\{ \begin{array}{c}
\text{P2} \\
\text{P3}
\end{array} \right\} \text{ IP } _\_ ;
\]

IP : Command intermediate position

P2, P3 : Select from 2nd or 3rd reference positions. If not selected, 2nd reference position return is automatically selected.
5.6 RETURN SIGNAL

If the jog feed (JOG) mode is selected, and a return signal (RTN1n, RTN2n, or RTN3n) is set to 1, rapid traverse occurs to the reference position specified by the return signal. If the signal becomes 0 during rapid traverse, axis movement is decelerated to a stop. When the reference position is reached, axis movement is stopped completely, and the reference position return completion output signal (ZPn, ZP2n, or ZP3n) signal is set to 1.

Before starting to use this function, it is necessary to set up the origin; a reference position return should be made previously.

5.7 BUTT–TYPE REFERENCE POSITION SETTING

The butt–type reference position setting function automates the setting of a reference position by butting the tool against a mechanical stopper on an axis. This function is provided to eliminate the variations in reference position setting that arise when the procedure is performed by different operators, and to minimize the amount of work involved in making fine adjustments after reference position setting.

Select the axis for which the reference position is to be set, then perform cycle start. The following operations are performed automatically:

1. The torque (force) of the selected axis is reduced so that the butting feedrate is constant. The tool is butted against the mechanical stopper. Then, the tool is drawn back a parameter–set amount from the mechanical stopper.

2. Again, the torque (force) of the selected axis is reduced, then the tool is butted against the mechanical stopper. Then, the tool is drawn back a parameter–set amount from the mechanical stopper.

3. The point on the axis to which the tool is drawn back is set as the reference position.
5.8 REFERENCE POSITION EXTERNAL SETTING

Basic procedure for reference position external setting

The machine is moved to a position to be specified as its reference position, for example, in the jog feed mode. Setting the reference position external setting signal to 1 specifies the current position of the relevant axis as the reference position.

1. Move the machine to a position to be specified as the reference position, for example, by jogging.
2. Select the jog feed (JOG) mode, and set the ZRN (bit 7 of G043) signal to 1.
3. Set the reference position external setting signal (ZPEXTn) corresponding to the axis for which the reference position is to be set up to 1. The reference position is set up for the specified axis.

5.9 LINEAR SCALE WITH ABSOLUTE ADDRESSING REFERENCE MARKS

The linear scale with absolute addressing reference marks has reference marks (one-rotation signals) at intervals that change at a constant rate. By determining the reference mark interval, the corresponding absolute position can be deduced. The CNC makes a small movement along an axis to measure the one-rotation signal interval, then calculates the absolute position. The reference position can be established without performing positioning to the reference position.

![Sample linear scale with absolute addressing reference marks](image)

**Fig. 5.9** Sample linear scale with absolute addressing reference marks
By teaching the CNC the position the tool is to arrive, the CNC moves the tool to that position. The position is specified using coordinates on a certain coordinate system.

There are two types of coordinate systems.

- **Machine coordinate system**
- **Workpiece coordinate system**

As necessary, one of the above coordinate systems is used for specifying coordinates for the target position of the tool.
6.1 MACHINE COORDINATE SYSTEM

Machine coordinate system is a coordinate system set with a zero point proper to the machine system. A coordinate system in which the reference position becomes the parameter-preset coordinate value when manual reference point return is performed, is set.
6. COORDINATE SYSTEMS

6.2 WORKPIECE COORDINATE SYSTEM

A coordinate system in which the zero point is set to a fixed point on the workpiece, to make programming simple. A workpiece coordinate system may be set by using one of the following methods:

1. Using G92
2. Automatic setting

When (1) is used, a workpiece coordinate system is established using the numeric value programmed after G92.

When (2) is used, a workpiece coordinate system is automatically established upon a manual reference position return, as specified in a parameter.

6.2.1 Setting a Workpiece Coordinate System (Using G92)

Format

\[ \text{G92 } \_ ; \]

Examples

By using the above command, a workpiece coordinate system can be set so that the current machine position is at a specified position.

6.2.2 Automatic Coordinate System Setting

When manual reference position return is performed, a workpiece coordinate system can be set automatically so that the current tool position at the reference position becomes a desired position which is set using a parameter in advance. This functions as if G92IP__; were specified at the reference position.
6.3
PLANE SELECTION
(G17, G18, G19)

A plane subject to circular interpolation or drilling can be selected by specifying a G code.

<table>
<thead>
<tr>
<th>G code</th>
<th>Selected plane</th>
<th>Xp</th>
<th>Yp</th>
<th>Zp</th>
</tr>
</thead>
<tbody>
<tr>
<td>G17</td>
<td>Xp–Yp plane</td>
<td>X axis or an axis parallel to the X axis</td>
<td>Y axis or an axis parallel to the Y axis</td>
<td>Z axis or an axis parallel to the Z axis</td>
</tr>
<tr>
<td>G18</td>
<td>Zp–Xp plane</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>G19</td>
<td>Yp–Zp plane</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Explanations

One of the existing parallel axes is determined by an axis address that appears in the block for which G17, G18, or G19 is specified.

- **Example 1**

  When X and U, Y and V, and Z and W are parallel to each other, respectively

  - G17 X_ Y_ ...... XY plane
  - G17 U_ Y_ ...... UY plane
  - G18 X_ W_ ...... WX plane
  - G18 U_ W_ ...... WU plane

- **Example 2**

  Planes remain unchanged in blocks for which G17, G18, or G19 is not specified.

  - G18 X_ Z_ ...... ZX plane
    - X_ Y_ ...... Plane not changed (ZX plane)

- **Example 3**

  If G17, G18, or G19 is specified for a block, and no axis address is specified in that block, the axis addresses for the basic three axes are assumed to be omitted.

  - G17 ............... XY plane
  - G17 X_ ............... XY plane
  - G17 U_ ............... UY plane

**NOTE**

A parameter is used to specify which axis, X, Y, or Z the additional axis is parallel to. The move command functions regardless of the plane selection.

For example, suppose that the following is specified:

```gCode
G17 Z_;
```

Axis Z does not exist on the XpYp plane. The XY plane is just selected, and the Z axis is moved regardless of the plane.
7 COORDINATE VALUE AND DIMENSION
7. COORDINATE VALUE AND DIMENSION

7.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)

There are two ways to command travels to 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. G90 and G91 are used to command absolute or incremental command.

**Format**

<table>
<thead>
<tr>
<th>Format</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>Absolute command</td>
<td>G90 IP_ ;</td>
</tr>
<tr>
<td>Incremental command</td>
<td>G91 IP_ ;</td>
</tr>
</tbody>
</table>

**Example**

For the above figure, incremental command programming results in:

G91 X60.0 Y40.0 ;

while absolute command programming results in:

G90 X40.0 Y70.0 ;
7.2 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 parameter-set when the machine is installed.

Command G20, G21 at the head of the program.

Inch/metric conversation can also be done by MDI setting.

The contents of setting data differs depending on whether G20 or G21 is commanded.

**WARNING**

Never specify G20 or G21 for switching in the middle of a program.

7.3 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 time. The position of the decimal point is at the mm, inch, deg position.

There are two types of decimal point notation: calculator-type notation and standard notation.

When calculator-type decimal notation is used, a value without decimal point is considered to be specified in millimeters, inch or deg. When standard decimal notation is used, such a value is considered to be specified in least input increments.

Use parameters to select input method; whether to input by pocket calculator type input, or by the usual decimal point input.

Values can be specified both with and without decimal point in a single program.

<table>
<thead>
<tr>
<th>Program command</th>
<th>Pocket calculator type decimal point programming</th>
<th>Usual decimal point programming</th>
</tr>
</thead>
<tbody>
<tr>
<td>X1000 Command value without decimal point</td>
<td>1000mm Unit : mm</td>
<td>1mm Unit : Least input increment (0.001 mm)</td>
</tr>
<tr>
<td>X1000.0 Command value with decimal point</td>
<td>1000mm Unit : mm</td>
<td>1000mm Unit : mm</td>
</tr>
</tbody>
</table>
7.4 LINEAR AXIS AND
ROTATION AXIS

A linear axis refers to an axis moving linearly, and for it values are specified in mm or inches.
A rotation axis refers to a rotating axis, and for it values are specified in degrees.
For rotation axes, note the following:

- Inch–metric switching is not performed.
- The machine coordinate system is always normalized to the range from 0deg to 360deg.

7.5 ROTATION AXIS
ROLL–OVER
FUNCTION

The rotation axis roll–over function rounds the absolute coordinate value and relative coordinate value of a rotation axis to a coordinate value within one rotation. This prevents coordinate values to overflow.
In an incremental command, the specified value is regarded as the amount of travel.
In an absolute command, the specified value is rounded to within one rotation. The resulting coordinate value is used as the end point. A parameter is used to specify whether to determine the move direction by the sign of the specified value or by the move distance (the shortest move distance to the end point is selected).
8. SPINDLE FUNCTIONS
(ONLY FOR Power Mate–D)

8.1 S CODE OUTPUT

Specify the spindle speed with up to five digits immediately after address S. The 5–digit numeric value is output to the PMC as a 32–bit binary code. The code is maintained until another S is specified. The maximum number of input digits for S can be specified using a parameter.

8.2 SPINDLE SPEED ANALOG OUTPUT (S ANALOG OUTPUT)

The speed of the analog interface spindle is controlled. Specify the spindle speed with up to five digits immediately after address S. According to the specified spindle speed, a speed command is output to the spindle motor in a form of analog voltage. During constant surface speed control, an analog voltage is output so that it matches the spindle speed reached after constant surface speed control.

8.3 SPINDLE SPEED SERIAL OUTPUT (S SERIAL OUTPUT)

The speed of the serial interface spindle is controlled. Specify the spindle speed with up to five digits immediately after 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.

8.4 SPINDLE OUTPUT CONTROL BY THE PMC

If a speed command for the spindle motor is input in a form of [sign + 12–bit binary code], the command is output to the spindle motor according to the input.
8.5  CONSTANT SURFACE SPEED CONTROL

Whether to perform constant surface speed control is specified using G96 or G97.

- **G96**: Constant surface speed control mode
- **G97**: Constant surface speed control cancel mode

If the surface speed is specified with an S code (S followed by a numeric value) in the constant surface speed control mode, the spindle speed is controlled so that a constant surface speed can be maintained while the tool position is changing.

The axis on which the calculation for constant surface speed control is based can be specified with either a parameter or the following command:

```
G96 P\alpha ;
```

- `P0`: Axis specified with a parameter
- `P\alpha`: \( \alpha \)th axis (\( \alpha = 1 \) to 2)

The specifiable range of the S code is as follows:

- 1 to 99999 m/min or feet/min

In the constant surface speed control cancel mode, the spindle speed is specified using an S code.

In the constant surface speed control mode, a constant surface speed control on signal is output.

By specifying the following command, the maximum spindle speed can be set:

```
G92 S-- ;
```

(Where, S-- is the maximum spindle speed in rpm)

The spindle speed is clamped when it reaches the specified maximum spindle speed.

8.6  SPINDLE OVERRIDE

To the spindle speed specified by S, an override from 0% to 254% can be applied (in steps of 1%).

8.7  SPINDLE ORIENTATION

The spindle orientation function stops the spindle at a certain position.

You can perform spindle orientation simply by mounting a position coder on the spindle. Stoppers or pins for physically stopping the spindle at a specified position are not necessary. A spindle can be instantly oriented, even when rotating at high speed, thereby greatly reducing the orientation time.

8.8  SPINDLE OUTPUT SWITCHING

Spindle output switching switches between the two windings, one for low speed and the other for high speed, incorporated into the special spindle motors. This ensures that the spindle motor demonstrates stable output characteristics over a wide range.
9. TOOL FUNCTIONS

9.1 T CODE OUTPUT

A tool can be selected by specifying a tool number of up to eight digits immediately after address T. The tool number is output to the PMC in a 32-bit binary code. This code is kept till the next T code is commanded. Maximum input digits are set by parameters.
10. MISCELLANEOUS FUNCTIONS

When up to eight digits immediately after address M are specified, a 32-bit binary code is output. The maximum number of input digits can be specified with a parameter. This binary code is used for on/off control of the machine. A block can usually contain up to three M codes although only one of them is effective.

The following M codes are used for special purposes:

- M00 : Program stop
- M01 : Optional stop
- M02 : End of program
- M30 : End of program and tape rewind

The above M codes can also be output in binary codes.

M98 (subprogram call), M99 (return from a subprogram), and an M code (set with a parameter) for calling a subprogram or custom macro are processed in the CNC, so that no signal is output.

10.2 1–BLOCK PLURAL M COMMAND

Up to five M codes can be simultaneously specified in one block. As these M codes are simultaneously sent to PMC side, the machining cycle time compared with the conventional 1–block single M command is reduced.

Example:

(i) 1–block single M command
   M40;
   M50;
   M60;
   G28G91X0Y0Z0;
   :

(ii) 1–block plural M command
   M40M50M60;
   G28G91X0Y0Z0;
   :

NOTE
1. The maximum input value of the first M code is 99999999, while the maximum input values of the second and five M codes are 65535.
2. A strobe signal is provided for each of the first to five M codes (MF, MF2, MF3, MF4, and MF5). When all the operations for the first to third M codes are completed, completion signal FIN is output.
10.3 HIGH–SPEED M/S/T INTERFACE

The communication of execution command signal (strobe signal) and completion signal is the M/S/T function were simplified to realize a high–speed execution of M/S/T function.

The time required for cutting can be minimized by speeding up the execution time of M/S/T function.

The following describes an example of auxiliary function M code command. The same applies to the T and S functions.

When an M code is specified, the CNC inverts the logical level of strobe signal MF. Thus, when the signal is 0, it becomes 1. When it is 1, it becomes 0. After inverting strobe signal MF, the CNC assumes the completion of PMC operation once the logical level of completion signal MFIN from PMC has become the same as the logical level of strobe signal MF.

In the usual system, if the leading edge (from “0” to “1”) of the completion signal FIN of M/S/T is received and then the trailing edge (from “1” to “0”) of the signal FIN is received, it is considered that the operation has been completed. However, in this system, the operation is considered to have been completed by a single change of completion signal MFIN.

Example) M10;
        M20;

<table>
<thead>
<tr>
<th>High–speed System Time Chart</th>
</tr>
</thead>
<tbody>
<tr>
<td>M command Mxx</td>
</tr>
<tr>
<td>code signal</td>
</tr>
<tr>
<td>M10 M20</td>
</tr>
<tr>
<td>Strobe signal MF</td>
</tr>
<tr>
<td>M function completion signal MFIN</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Conventional System Time Chart</th>
</tr>
</thead>
<tbody>
<tr>
<td>M command Mxx code signal</td>
</tr>
<tr>
<td>M10 M20</td>
</tr>
<tr>
<td>Strobe signal MF</td>
</tr>
<tr>
<td>Operation at RMC side</td>
</tr>
<tr>
<td>Completion signal FIN</td>
</tr>
</tbody>
</table>
NOTE
1 Either the conventional system or the high-speed system can be selected for communication of strobe signal and completion signal.
2 In the conventional system, only one completion signal is available for all functions of M/S/T. However, in the high-speed system, one completion signal is available for each of M/S/T functions.
11 PROGRAM CONFIGURATION
11. PROGRAM CONFIGURATION

11.1 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 (when the 8-digit program number option is used, however, eight digits following address O) 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 ways.

11.2 PROGRAM NAME

A program name can be given to the program to distinguish the program from other programs when displaying all the registered program on a screen. Register the name between the control-out and the control-in. Any codes usable in the CNC can be used for the program name. The program name is displayed with the program number in the directory display of registered programs. Note that the program name displayed is within 31 characters.

Example) O1234 (PROGRAM FOR ATC);

11.3 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.
11.4 SUB PROGRAM

When there are fixed sequences or frequently repeated patterns in a program, programming can be simplified by entering these pattern as subprograms to the memory. Sub program is called by M98, and M99 commands return from the sub program. The sub program can be nested 4 folds.

<table>
<thead>
<tr>
<th>Main program</th>
<th>Sub program</th>
<th>Sub program</th>
<th>Sub program</th>
<th>Sub program</th>
</tr>
</thead>
<tbody>
<tr>
<td>O0001 ;</td>
<td>O1000 ;</td>
<td>O2000 ;</td>
<td>O3000 ;</td>
<td>O4000 ;</td>
</tr>
<tr>
<td>M98P1000 ;</td>
<td>M98P2000 ;</td>
<td>M98P3000 ;</td>
<td>M98P4000 ;</td>
<td></td>
</tr>
<tr>
<td>M30 ;</td>
<td>M99 ;</td>
<td>M99 ;</td>
<td>M99 ;</td>
<td></td>
</tr>
</tbody>
</table>

1–loop nesting 2–loop nesting 3–loop nesting 4–loop nesting

Format

Sub program call

\[
\text{M98 P } \underbrace{\ldots}_{\text{Number of repetitive calls}} \underbrace{\ldots}_{\text{Subprogram number}} ;
\]

If the number of repetitive calls is omitted, 1 is assumed.

Return from sub program

\[
\text{M99 ;}
\]

11.5 SEQUENCE NUMBER

Sequence number can be given in a 5–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 screen. The sequence number can also be searched in the program by the sequence number search function.

11.6 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.
The following table shows the basic addresses and the range of values to be specified. The range, however, is that of CNC. Note that the range of the machine is different from this.

<table>
<thead>
<tr>
<th>Function</th>
<th>Address</th>
<th>Metric input</th>
<th>Inch input</th>
</tr>
</thead>
<tbody>
<tr>
<td>Program number</td>
<td>O (*1)</td>
<td>1–9999 (1 to 99999999 (when eight digits is selected))</td>
<td>1–9999 (1 to 99999999 (when eight digits is selected))</td>
</tr>
<tr>
<td>Sequence number</td>
<td>N</td>
<td>1–99999</td>
<td>1–99999</td>
</tr>
<tr>
<td>Preparatory function</td>
<td>G</td>
<td>0–999</td>
<td>0–999</td>
</tr>
<tr>
<td>Dimension word,</td>
<td>IS–B</td>
<td>±99999.999mm</td>
<td>±99999.999inch (Note2)</td>
</tr>
<tr>
<td>Setting unit</td>
<td>IS–C</td>
<td>±99999.999deg</td>
<td>±99999.999deg</td>
</tr>
<tr>
<td>Feed per minute,</td>
<td>IS–B</td>
<td>1–240000mm/min</td>
<td>0.01–9600.00inch/min</td>
</tr>
<tr>
<td>Setting unit</td>
<td>IS–C</td>
<td>1–100000mm/min</td>
<td>0.01–4000.00inch/min</td>
</tr>
<tr>
<td>Feed per rotation,</td>
<td>F</td>
<td>0.01–500.00mm/rev</td>
<td>0.0001–9.9999inch/rev</td>
</tr>
<tr>
<td>Setting unit</td>
<td>F</td>
<td>0.01–500.00mm/rev</td>
<td>0.0001–9.9999inch/rev</td>
</tr>
<tr>
<td>Spindle function</td>
<td>S</td>
<td>0–20000</td>
<td>0–20000</td>
</tr>
<tr>
<td>Tool function</td>
<td>T</td>
<td>0–99999999</td>
<td>0–99999999</td>
</tr>
<tr>
<td>Miscellaneous function</td>
<td>M</td>
<td>0–99999999</td>
<td>0–99999999</td>
</tr>
<tr>
<td>Dwell,</td>
<td>IS–B</td>
<td>0–99999.999 (sec or rev)</td>
<td>0–99999.999 (sec or rev)</td>
</tr>
<tr>
<td>Setting unit</td>
<td>IS–C</td>
<td>0–99999.999 (sec or rev)</td>
<td>0–99999.999 (sec or rev)</td>
</tr>
<tr>
<td>Program number specification</td>
<td>P</td>
<td>1–9999</td>
<td>1–9999</td>
</tr>
<tr>
<td>Number of repeats</td>
<td>P</td>
<td>1–999</td>
<td>1–999</td>
</tr>
<tr>
<td>Offset number</td>
<td>H, D</td>
<td>0–400</td>
<td>0–400</td>
</tr>
</tbody>
</table>

**NOTE**
1. “:*” can be used for 0 in ISO Code.
2. Coordinates maximum command value for inch input/metric output is limited to: ±3937.0078 inch (IS–B)/ ±393.70078 inch (IS–C).
11.8 TAPE FORMAT

The variable block word address format with decimal point is adopted as tape format. See List of Tape Format in Appendix C for details on tape formats.

11.9 LABEL SKIP

Label skip function is valid in the following cases, and “LSK” is displayed on the screen.
- When power is put on.
- When the NC is reset.

When label skip function is in valid, all codes to the first encountered end of block (EOB) code are ignored. The ignored part is called “Reader part”, and section after the first end of block (EOB) code, “significant information”.

11.10 CONTROL–IN/CONTROL–OUT

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”.

<table>
<thead>
<tr>
<th>ISO code</th>
<th>EIA code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Control–out</td>
<td>Channel 2–4–5 on</td>
</tr>
<tr>
<td>Control–in</td>
<td>Channel 2–4–7 on</td>
</tr>
</tbody>
</table>

11.11 OPTIONAL BLOCK SKIP

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

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

I can be used for n. The 1 to /1 can be omitted.

Example) /1 N12345 G00 X100.Y200.;

2 to 9 can also be used for the n of the /n.

11.12 ADDITIONAL OPTIONAL BLOCK SKIP

A parity check is made on the number of punch holes for each input tape character. If the parity does not match, an alarm occurs (TH check). A parity check is made on each input data block. If the number of characters in one block (from the code next to EOB to another EOB) is odd, an alarm occurs (TV check). The TH or TV check cannot be made on the area skipped by the label skip function. The TH check is not made on the command field. A parameter can be used to specify whether the characters constituting comments are to be counted when obtaining the number of characters for TV check. The TV check function is validated or invalidated according to the value set on the MDI panel.
12 FUNCTIONS TO SIMPLIFY PROGRAMMING
12.1 CANNED CYCLES (G73, G74, G76, G80 – G89, G98, G99) (ONLY FOR Power Mate i–D)

Canned cycle is a function to simplify commands for machining (boring, drilling, or tapping, etc.). The canned cycle has the positioning plane and the drilling axis. The positioning plane is specified with the plane selection of G17, G18, and G19. The drilling axis is the basic axis X, Y or Z (that does not compose the positioning plane) or its parallel axis.

<table>
<thead>
<tr>
<th>G code</th>
<th>Positioning plane</th>
<th>Drilling axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>G17</td>
<td>Xp–Yp plane</td>
<td>Xp</td>
</tr>
<tr>
<td>G18</td>
<td>Zp–Xp plane</td>
<td>Yp</td>
</tr>
<tr>
<td>G19</td>
<td>Yp–Zp plane</td>
<td>Zp</td>
</tr>
</tbody>
</table>

Xp : X axis or its parallel axis
Yp : Y axis or its parallel axis
Zp : Z axis or its parallel axis

The drilling axis address commanded in the same block as the G codes, G73 – G89, decides whether the drilling axis is the basic axis or its parallel axis. If the drilling axis address was not commanded, the basic axis becomes the drilling axis.

Axis other than the drilling axis becomes the positioning axis.

Example)
When U, V, W axes are set as parallel axes for X, Y, Z axes respectively.
- G17G81 .... Z_ ; Drilling axis is Z axis.
- G17G81 .... W_ ; Drilling axis is W axis.
- G18G81 .... Y_ ; Drilling axis is Y axis.
- G18G81 .... V_ ; Drilling axis is V axis.
- G19G81 .... X_ ; Drilling axis is X axis.
- G19G81 .... U_ ; Drilling axis is U axis.

It is not always necessary to command G17, G18, G19 in the same block as G73 – G89.

NOTE
X axis can always be appointed the drilling axis by parameter setting.

A canned cycle can be canceled by G80 or a G code (G00 to G03) of group 01.
Positioning can be commanded with optional axes other than the drilling axis. The drilling cycle starts after the positioning.
The description below assumes that the hole machining axis is the X–axis.
The following 12 types of canned cycles are available.
## 12 types of canned cycles (1/4)

<table>
<thead>
<tr>
<th>G code</th>
<th>Operation</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>G73</strong></td>
<td><img src="image1" alt="Diagram" /></td>
<td>High-speed peck drilling cycle (Note 1)</td>
</tr>
<tr>
<td><strong>G74</strong></td>
<td><img src="image2" alt="Diagram" /></td>
<td>Counter tapping cycle</td>
</tr>
<tr>
<td><strong>G76</strong></td>
<td><img src="image3" alt="Diagram" /></td>
<td>Fine boring cycle</td>
</tr>
</tbody>
</table>
### 12 types of canned cycles (2/4)

<table>
<thead>
<tr>
<th>G code</th>
<th>Operation</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>G81</strong></td>
<td><img src="image" alt="G81 Diagram" /></td>
<td>Drilling cycle (Spot drilling)</td>
</tr>
<tr>
<td><strong>G82</strong></td>
<td><img src="image" alt="G82 Diagram" /></td>
<td>Drilling cycle (Counter boring)</td>
</tr>
<tr>
<td><strong>G83</strong></td>
<td><img src="image" alt="G83 Diagram" /></td>
<td>Peck drilling cycle (Note 1)</td>
</tr>
</tbody>
</table>
### 12 types of canned cycles (3/4)

<table>
<thead>
<tr>
<th>G code</th>
<th>Operation</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>G84</strong></td>
<td><img src="image1" alt="Tapping cycle" /></td>
<td>Tapping cycle</td>
</tr>
<tr>
<td><strong>G85</strong></td>
<td><img src="image2" alt="Boring cycle" /></td>
<td>Boring cycle</td>
</tr>
<tr>
<td><strong>G86</strong></td>
<td><img src="image3" alt="Boring cycle" /></td>
<td>Boring cycle</td>
</tr>
<tr>
<td><strong>G87</strong></td>
<td><img src="image4" alt="Back boring cycle" /></td>
<td>Back boring cycle</td>
</tr>
</tbody>
</table>

- **G98 mode**
- **G99 mode**

---

[12. FUNCTIONS TO SIMPLIFY PROGRAMMING](#)
### 12 types of canned cycles (4/4)

<table>
<thead>
<tr>
<th>G code</th>
<th>G98 mode</th>
<th>G99 mode</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G88</td>
<td>Initial level</td>
<td>R point</td>
<td>Boring cycle</td>
</tr>
<tr>
<td></td>
<td>Z point</td>
<td>P</td>
<td>Dwell</td>
</tr>
<tr>
<td>G89</td>
<td>Initial level</td>
<td>R point</td>
<td>Boring cycle</td>
</tr>
<tr>
<td></td>
<td>R point</td>
<td>P</td>
<td>Z point</td>
</tr>
</tbody>
</table>

Cutting feed  
Rapid traverse  
Manual feed  
P  
Dwell  
Z  
Z point  
(Hole bottom position)  
OSS  
 Oriented spindle stop (Spindle stops at constant rotation position)  
Shift  
I  
Initial position level  
R  
R point  
Note 1 “d” of G73 and G84 is set by parameters.
When the drilling axis is Z axis, machining data in the canned cycle is commanded as follows:

**Format**

```
G○○ X_ Z_ R_ Q_ P_ K_ F_ ;
```

- **Drilling mode G○○**: See previous table.
- **Drilling position data X**: 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.
- **K**: Specify how many times to repeat. When specified K0, drilling data will be set, but no drilling will be done.
- **F**: Specify feed rate for cutting.

**Explanations**

- **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

  ![Diagram of R point level return](image)

- **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.

  ![Diagram of Initial level return](image)
12.1.1 Canned Cycle Cancel (G80)

G80 cancels canned cycles.

Format

```
G80 ;
```

Explanations

All canned cycles are canceled to perform normal operation. Point R and point Z are cleared. This means that $R = 0$ and $Z = 0$ in incremental mode. Other drilling data is also canceled (cleared).

Examples

- `M3 S100 ;` Cause the spindle to start rotating.
- `G90 G88 X300. Z–150. R–120. F120. ;
  Position, drill hole 1, then return to point R.
- `X1000. ;
  Position, drill hole 2, then return to point R.
- `G80 G28 G91 X0 Z0 : Return to the reference position return, canned cycle cancel
- `M5 ;` Cause the spindle to stop rotating.
12.2 RIGID TAP (ONLY FOR Power Mate i–D)

In tapping, the feed amount of drilling 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:

\[ P = \frac{F}{S}, \]

where 
- \( P \): Pitch of screw of tapper (mm)
- \( F \): Feed rate of drilling axis (mm/min)
- \( S \): Spindle speed (rpm)

The rotation of spindle and feed of Z axis are independently controlled in the tapping cycle G74/G84. Therefore, the above conditions may not always be satisfied. Especially at the hole bottom, both the rotation of spindle and feed of drilling 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 drilling 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 drilling 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.

\[ P = \frac{F}{S} \]

The pitch of screw tap can be directly specified.

For rigid tapping, specify M29 S○○○○○ before tapping command G74/G84.

Rigid tapping can be canceled by G80 (canned cycle cancel).

![Diagram of Spindle Control System]

- Spindle control (voltage calculation of spindle speed rpm)
- Distributed pulse
- Error counter
- D/A converter
- Spindle amplifier
- Spindle motor
- Gear ratio
- Position coder
- Gear ratio
- Spindle

\[ \times 4 \]

DMR

CMR

\[ \times 4 \]
The Control System of Spindle during Rigid Tapping

<table>
<thead>
<tr>
<th>Gear ratio of spindle to position coder (1 : p)</th>
<th>Least command increment (detection unit) deg</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 : 1</td>
<td>0.088 (1x360 / 4096)</td>
</tr>
<tr>
<td>1 : 2</td>
<td>0.176 (2x360 / 4096)</td>
</tr>
<tr>
<td>1 : 4</td>
<td>0.352 (4x360 / 4096)</td>
</tr>
<tr>
<td>1 : 8</td>
<td>0.703 (8x360 / 4096)</td>
</tr>
</tbody>
</table>

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.

In addition, use of the spindle motor incorporating the position coder enables rigid tapping but disables threading and per revolution dwell.
12.2.1 Rigid Tapping (G84)

When the spindle motor is controlled in rigid mode as if it were a servo motor, a tapping cycle can be sped up.

Format

\[ G84 \ X_\_ \ Z_\_ \ R_\_ \ P_\_ \ F_\_ ; \]

- **X_**: Hole position data
- **Z_**: The distance from point R to the bottom of the hole and the position of the bottom of the hole
- **R_**: The distance from the initial level to point R level
- **P_**: Dwell time at the bottom of the hole and at point R when a return is made
- **F_**: Cutting feedrate

Explanations

After positioning along the X–axis, rapid traverse is performed to point R.
Tapping is performed from point R to point Z. When tapping is completed, a dwell is performed and the spindle is stopped. The spindle is then rotated in the reverse direction, the tool is retracted to point R, then the spindle is stopped. Rapid traverse to initial level is then performed. While tapping is being performed, the feedrate override and spindle override are assumed to be 100%.

- **Rigid mode**
  - Specify M29 S***** before a tapping command.

- **Thread lead**
  - In feed–per–minute mode, the thread lead is obtained from the expression, feedrate \times spindle speed. In feed–per–revolution mode, the thread lead equals the feedrate speed.
  - If a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.
12. FUNCTIONS TO SIMPLIFY PROGRAMMING

12.2.2 Left–handed Rigid Tapping Cycle (G74)

Format

```
G74 X_ Z_ R_ P_ F_; 
```

- **X_**: Hole position data
- **Z_**: The distance from point R to the bottom of the hole and the position of the bottom of the hole
- **R_**: The distance from the initial level to point R level
- **P_**: Dwell time at the bottom of the hole and at point R when return is made.
- **F_**: Cutting feedrate

Explanations

After positioning along the X–axis, rapid traverse is performed to point R. Tapping is performed from point R to point Z. When tapping is completed, a dwell is performed and the spindle is stopped. The spindle is then rotated in the normal direction, the tool is retracted to point R, then the spindle is stopped. Rapid traverse to initial level is then performed. While tapping is being performed, the feedrate override and spindle override are assumed to be 100%.

- **Rigid mode**
- **Thread lead**

Specify **M29 S***** before a tapping command.**

In feed–per–minute mode, the thread lead is obtained from the expression, feedrate ÷ spindle speed. In feed–per–revolution mode, the thread lead equals the feedrate.

If a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.
12. FUNCTIONS TO SIMPLIFY PROGRAMMING

12.2.3 Peck Rigid Tapping Cycle (G84 or G74)

Tapping a deep hole in rigid tapping mode may be difficult due to chips sticking to the tool or increased cutting resistance. In such cases, the peck rigid tapping cycle is useful.

In this cycle, cutting is performed several times until the bottom of the hole is reached. Two peck tapping cycles are available: High-speed peck tapping cycle and standard peck tapping cycle. These cycles are selected using the PCP bit (bit 5) of parameter 5200.

Format

G84 (or G74) X_ Z_ R_ P_ Q_ F_ ;

- X_ : Hole position data
- Z_ : The distance from point R to the bottom of the hole and the position of the bottom of the hole
- R_ : The distance from the initial level to point R level
- P_ : Dwell time at the bottom of the hole and at point R when a return is made
- Q_ : Depth of cut for each cutting feed
- F_ : The cutting feedrate

• High-speed peck tapping cycle (Parameter PCP(No.5200#5=0))

(1) The tool operates at a normal cutting feedrate. The normal time constant is used.
(2) Retraction can be overridden. The retraction time constant is used.

• Peck tapping cycle (Parameter PCP(No.5200#5=1))

(1) The tool operates at a normal cutting feedrate. The normal time constant is used.
(2) Retraction can be overridden. The retraction time constant is used.
(3) Retraction can be overridden. The normal time constant is used.

During a rigid tapping cycle, CNC checks whether the pulse distribution is performed or not and start the next operation at the end of each operation of (1) and (2) in the peck tapping cycle.
13 TOOL COMPENSATION FUNCTION
13.1 TOOL LENGTH COMPENSATION (G43, G44, G49)

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

Explanations

- Tool length compensation and its cancellation (G43, G44, G49)

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G43</td>
<td>Tool length compensation +</td>
</tr>
<tr>
<td>G44</td>
<td>Tool length compensation –</td>
</tr>
<tr>
<td>G49</td>
<td>Tool length compensation cancel</td>
</tr>
</tbody>
</table>

In G43 mode, the tool is offset to the + direction for the preset tool length offset amount. In G44 mode, it is offset to the – direction for the preset tool length offset amount. G49 cancels tool length compensation.

- Tool length compensation axis

  Tool length compensation can be performed for three types of axes. Compensation for the Z axis is tool length compensation A. That for the axis vertical to the selected plane is tool length compensation B. That for the axis specified by the G43 or G44 block is tool length compensation C. Which compensation to perform can be selected by a parameter.

- Assignment of offset amount (H code)

  The offset amount can be set in the tool length compensation memory. By specifying an offset number with the H code, offset amount loaded in corresponding tool length compensation memory is used as tool length compensation amount.

Format

<table>
<thead>
<tr>
<th>Tool length offset A</th>
<th>G43 Z_ H_ ;</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>G44 Z_ H_ ;</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Tool length offset B</th>
<th>G17 G43 Z_ H_ ;</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>G17 G44 Z_ H_ ;</td>
</tr>
<tr>
<td></td>
<td>G18 G43 Y_ H_ ;</td>
</tr>
<tr>
<td></td>
<td>G18 G44 Y_ H_ ;</td>
</tr>
<tr>
<td></td>
<td>G19 G43 X_ H_ ;</td>
</tr>
<tr>
<td></td>
<td>G19 G44 X_ H_ ;</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Tool length offset C</th>
<th>G43 α_ H_ ;</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>G44 α_ H_ ;</td>
</tr>
</tbody>
</table>

| Tool length offset cancel | G49 ; or H0 ; |

Explanation of each address

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G43</td>
<td>Positive offset</td>
</tr>
<tr>
<td>G44</td>
<td>Negative offset</td>
</tr>
<tr>
<td>G17</td>
<td>XY plane selection</td>
</tr>
<tr>
<td>G18</td>
<td>ZX plane selection</td>
</tr>
<tr>
<td>G19</td>
<td>YZ plane selection</td>
</tr>
<tr>
<td>α</td>
<td>Address of a specified axis</td>
</tr>
<tr>
<td>H</td>
<td>Address for specifying the tool length offset value</td>
</tr>
</tbody>
</table>
13.2 TOOL COMPENSATION MEMORY

13.2.1 Tool Compensation Memory

Tool offset amount range which can be set is as follows:

<table>
<thead>
<tr>
<th>Increment system</th>
<th>Metric input</th>
<th>Inch input</th>
</tr>
</thead>
<tbody>
<tr>
<td>IS–B</td>
<td>± 999.999mm</td>
<td>± 99.9999inch</td>
</tr>
<tr>
<td>IS–C</td>
<td>± 999.9999mm</td>
<td>± 99.9999inch</td>
</tr>
</tbody>
</table>

99 tool offsets
Offset numbers (H code) 0 – 99 can be used.
H00 – H99

13.3 NUMBER OF TOOL OFFSETS

13.4 CHANGING OF TOOL OFFSET AMOUNT (PROGRAMMABLE DATA INPUT) (G10)

Tool offset amount can be set/changed with the G10 command.
When G10 is commanded in absolute input (G90), the commanded offset amount becomes the new tool offset amount. When G10 is commanded in incremental input (G91), the current tool offset amount plus the commanded offset amount is the new tool offset amount.

Format

G10 L11 P_ R_;

P_: Offset number
R_: Tool offset amount
14. ACCURACY COMPENSATION FUNCTION
14. ACCURACY COMPENSATION

14.1 STORED PITCH ERROR COMPENSATION (ONLY FOR Power Mate i–D)

The errors caused by machine position, as pitch error of the feed screw, can be compensated. This function is for better machining precision. As the offset data are stored in the memory as parameters, compensations of dogs and settings can be omitted. Offset intervals are set constant by parameters (per axis). Total offset points are:

Total offset points = 128 × controlled axes.

Optional distribution to each axis can be done by parameter setting. As each position:
Offset pulse = (-7 to +7) x (magnification)
Where Offset pulse unit is the same as detection unit
Magnification: 0 to 100 times, set by parameter (per axis)

14.2 BACKLASH COMPENSATION

This function is used to compensate lost motions proper to the machine system. Offset amounts come in a range of 0 to ±9999 pulses per axis, and is set as parameters in detection unit.

14.3 PROGRAMMABLE PARAMETER ENTRY (G10, G11)

Parameters and pitch errors data can be set by programs. therefore, following uses can be done example.

- Parameter setting such as pitch errors compensation data, etc. when the attachment is replaced.
- Parameters such as max. cutting speed and cutting feed time constant can be changed according to the machining conditions.

Format

G10 L50 ;
N_ R_ ; Input of parameters except axis type
N_ P_ R_ ; Input of axis type parameters
;
G11 ;

G10 L50 : Parameter input mode
G11 : Parameter input mode cancel
N_ : Parameter No. (or pitch error data No.+10000)
P_ : Axis No. (in the case of axis type parameter)
R_ : Parameter setting value (or pitch error data)

NOTE
Some parameters cannot be set.
15 MEASUREMENT FUNCTIONS
By commanding axis move after G31, linear interpolation can be commanded like in G01. If an external skip signal is input during this command, the remainder of this command is cancelled, and program skips to the next block.

G31 is a one–shot command and is valid for the commanded block only.

Coordinate value when skip signal is on, is stored in the system variables #5061 – #5068 of the customer macro, so this function can also be read with the customer macro function.

#506n : ntn axis skip signal position (n=1–8)

As the skip function can be used when move amount is not clear, this function can be used for:

- Tool measurement with tactile sensor.
15.2 MULTI-STEP SKIP FUNCTION (G31 P1 – G31 P4)

In blocks with either of P1 to P4 following G31 commanded, the coordinate value where skip signals (4 types) were input is stored in the custom macro variables, and at the same time, the remaining movement of the block is skipped. It is also possible to skip the remaining dwell with the skip signal by parameter, in a block where: G04 is commanded (dwell).

Parameters decide which skip command or dwell command is valid to which of the four skip signals. The skip signal is not necessarily unique to a single skip command or dwell command; it is also possible to set a skip signal to multiple skip command or dwell commands.

15.3 TOOL LENGTH MEASUREMENT

The value displayed as a relative position can be set in the offset memory as an offset value by a soft key.

Call offset value display screen. 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 and the relative position display value is then set as offset amounts.
16 CUSTOM MACRO
16.1 CUSTOM MACRO

A function covering a group of instructions is stored in the memory like the sub program. The stored function is represented by one instruction and is executed by simply writing the represented instruction. The group of instructions registered is called the custom macro body, and the representative instruction, the custom macro instruction.

The programmer need not remember all the instructions in the custom macro body. He needs only to remember the representative, custom macro instruction.

The greatest feature in custom macro is that variables can be used in the custom macro body. Operation between the variables can be done, and actual values can be set in the variables by custom macro instructions.

Bolt hole circle as shown above can be programmed easily. Program a custom macro body of a bolt hole circle; once the custom macro body is stored, operation can be performed as if the CNC itself has a bolt hole circle function. The programmer need only to remember the following command, and the bolt hole circle can be called any time.
Format

G65 Pp Rr Aa Bb Kk ;

- p  :  Macro number of the bolt hole circle
- r  :  Radius
- a  :  Initial angle
- b  :  Angle between holes
- k  :  Number of holes

With this function, the CNC can be graded up by the user himself. Custom macro bodies may be offered to the users by the machine tool builder, but the users still can make custom macro himself.

The following functions can be used for programming the custom macro body.

Explanations

- **Use of Variable**
  - Variables: #1 (i=1, 2, 3,........)
  - Quotation of variables: F#33 (#33: speed expressed by variables)

- **Operation between variables**
  - Various operation can be done between variables and constants.
  - The following operands, and functions can be used:
    - + (sum), – (difference), * (product), / (quotient), 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 = \text{SIN} \left[ \frac{\#2 + \#4}{3.14 + \#4} \right] \text{ABS (\#10)}

- **Control command**
  - Program flow in the custom macro body is controlled by the following command.
  - ![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 \geq \#k
      - \#j LE \#k  whether \#j \leq \#k
• Format of custom macro body

  The format is the same as the sub program.

```
0 Macro number ;

Custom macro body

M99 ;
```

• Custom macro instruction

  • Simple call

    G65 P (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.

  • Modal call A

    G66 P (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 canceled by G67.
    This function is useful when drilling cycles are programmed as custom macro bodies.
Macro call by G codes
The macro can also be called by the parameter–set G codes. Instead of commanding:
\[
\text{N\_G65 P\ldots <argument assignment> ;}
\]
macro can be called just by commanding:
\[
\text{N\_G** <argument assignment> ;}
\]
G code for calling the macro, and macro program number **** to be called, are coupled together and set as parameter.
Maximum ten G codes from G01 to G9999 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.

Macro call by M code
Custom macros can be called by pre–determined M codes which are set by parameters.
The following command
\[
\text{N\_G65 P\ldots <Argument assignment> ;}
\]
is equivalent to the following command:
\[
\text{N\_Mxx <Argument assignment> ;}
\]
The correspondence between M codes (Mxx) and program number (delta delta delta delta) of a macro shall be set by a parameter.
Signal MF and M code are not sent out the same as the subprogram call by M code.
Also when this M code is specified in a program called a macro calling G code or a subprogram calling M or T code, the M code is regarded as a normal M code.
Up to ten M codes from M01 to M99999999 can be used for custom macro calling M codes.

Sub program call by M code
An M code can be set by parameter to call a sub program. Instead of commanding:
\[
\text{N\_G\_X\_Y\_ ... M98 P\ldots ;}
\]
the same operation can be performed simply by commanding:
\[
\text{N\_G\_X\_Y\_ ... MXX ;}
\]
As for M98, M codes are not transmitted.
The M code XX for calling the sub program and the sub program number delta delta delta delta to be called are coupled together and set by parameter.
Maximum ten M codes from M01 to M99999999 can be used for macro call.
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 subprogram called by M code or T code, they are regarded as ordinary M codes.
Sub program call by T code

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

\[ \text{N}_\text{G}_\text{X}_\text{Y}_\text{…} \text{Tt ;} \]

the same operation is done as when commanded:

\[ \text{#149} = t; \]

\[ \text{N}_\text{G}_\text{X}_\text{Y}_\text{…} \text{M98 P9000 ;} \]

The T type 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. The T code is not output.

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.

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.

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 destroyed by being used in macro B.

Common variables #100 – #199, #500 – #699

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 common variable #1 used in a certain macro is the same as the common variable #i used in other macros. Therefore, a common variable #1 calculated in a macro can be used in any other macros.

Common variables #100 to #199 are cleared when power is turned off or reset, but common variables #500 to #699 are not cleared after power is turned off.

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:

- PMC interface signals (16 points DI and 48 points DO)
- D/R area information for PMC
- Tool offset amount
- Position information (actual position, skip position, block end position, etc.)
- Modal information (F code, G code for each group, etc.)
- Alarm message (Set alarm number and alarm message, and the CNC is set in an alarm status. The alarm number and message is displayed.)
- A date (year, month, day) and time (hour, minute, second) are indicated.
- Clock (Time can be known. A time can also be preset.)
- Single block stop, Miscellaneous function end wait hold
- Feed hold, Feed rate override, Exact stop inhibition
- The number of machining parts is indicated. It can be preset.
Value of variables or characters can be output to external devices via the reader/puncher interface with custom macro command. Results in measurement is output using custom macro.

- **External output commands**

- **Limitations**

  - **Usable variables**
    See Item ”Types of variables” above.

  - **Usable variable values**
    Maximum : ± 10^47
    Minimum : ± 10^-29

  - **Constants usable in <expression>**
    Maximum : ± 99999999
    Minimum : ± 0.0000001
    Decimal point allowed

  - **Arithmetic precision**
    8–digit decimal number

  - **Macro call nesting**
    Maximum 4 folds.

  - **Repeated ID numbers**
    1 – 3

  - **( ) nesting**
    Maximum 5 folds.

  - **Sub program call nesting**
    8 folds (including macro call nesting)
16.2
INTERRUPTION TYPE
CUSTOM MACRO

When custom macro interruption signal is input during automatic operation, the block currently under execution is interrupted and the specified custom macro is activated. After execution of this custom macro, it returns to the interrupted block and continues execution of the remaining commands.

M96P_;

When custom macro interruption signal is input between M96 block and M97 block, custom macro specified by P is activated.

M97;
16.3 PATTERN DATA INPUT

With this function, custom macro interruption signal can be input on detection of tool break, tool change cycle can be executed by custom macro, and machining is continued. This function simplifies program creation for CNC machining. Instead of programming in the NC format, the program can be created by selecting a menu and entering data according to the menu displayed on the CRT screen. A menu is provided for each type of drilling such as boring and tapping. A programmer can select data necessary for actual machining from these menus. Machining data such as hole position and hole depth is also provided in menus. The programmer can create a program simply by entering data from the menus. This function is basically executed by the custom macro created by a machine tool builder. What menus and machining data to prepare totally depends on a machine tool builder. Therefore, a machine tool builder can incorporate their own know-how into this function.

MENU: HOLE PATTERN 09505 N0001

1. TAPPING
2. DRILLING
3. BORING
4. POCKET
5. BOLT HOLE
6. LINEANGLE
7. GRID
8. PECK
9.
10.

>_< S 0 T0000

EDIT **** *** *** 15:56:32
[ MACRO ] [ MENU ] [ OPR ] [ ] [OPRT] ]

Pattern menu display

VAR. : BOLT HOLE 09505 N0001

<table>
<thead>
<tr>
<th>NO.</th>
<th>NAME</th>
<th>DATA</th>
<th>COMMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>500</td>
<td>TOOL</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>501</td>
<td>KIJUN X</td>
<td>0.000</td>
<td>*BOLT HOLE</td>
</tr>
<tr>
<td>502</td>
<td>KIJUN Y</td>
<td>0.000</td>
<td>CIRCLE*</td>
</tr>
<tr>
<td>503</td>
<td>RADIUS</td>
<td>0.000</td>
<td>SET PATTERN</td>
</tr>
<tr>
<td>504</td>
<td>S. ANGL</td>
<td>0.000</td>
<td>DATA TO VAR.</td>
</tr>
<tr>
<td>505</td>
<td>HOLES NO.</td>
<td>0.000</td>
<td>NO.500–505.</td>
</tr>
<tr>
<td>506</td>
<td></td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>507</td>
<td></td>
<td>0.000</td>
<td></td>
</tr>
</tbody>
</table>

ACTUAL POSITION (RELATIVE)

<table>
<thead>
<tr>
<th>X</th>
<th>0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y</td>
<td>0.000</td>
</tr>
<tr>
<td>Z</td>
<td>0.000</td>
</tr>
</tbody>
</table>

>_< S 0 T0000

MDI **** *** *** 15:56:32
[OFFSET ][SETTING ][ ] [OPRT] ]

Pattern data display
There are two types of NC programs; those which, once created, are scarcely changed, and those which are changed for each machining type. The former are programs created by the custom macro, and the latter are machining programs. If programs of these types are executed simultaneously, a battery may run out or the custom macro may be destroyed by error operation. Such problems can be solved by this function. The custom macro created by a machine tool builder is converted to an execute–form program, be cataloged in the Flash ROM module, and be executed.

Since the program is cataloged after converted to an execute–form program, the execution speed is high. The machining time is then reduced, and the precision is improved.

Since the program is cataloged in Flash ROM, there is no problem of battery extinction or custom macro destruction by error operation. The reliability is improved.

Since the cataloged program is not displayed on a program screen, the know–how of the machine tool builder is protected.

Since the custom macro is cataloged in Flash ROM, the program edit memory can be used efficiently.

The user can call the macro easily without knowing the cataloged program. A custom macro can be created and executed in the program edit memory as usual.

An original screen can be created by using the selecting screens by the soft key. The machine tool builder can extend the control function by using such functions as machining program creation and edit control, reader/punch interface control, and PMC data read/write functions.
16. CUSTOM MACRO FUNCTION

As with the conversational macro function of macro executors/compilers, the C language executor function is used to customize screens and include unique operations. Application programs for display and operation can be created in standard C language, in the same way as programs are made for normal personal computers. A program compiled on a personal computer is transferred and stored in flash ROM in the CNC via a memory card. The program is read into memory upon activation of the CNC, and executed by the C language executor.

NOTE
This function can not used for Power Mate i–D (2 path control).

Features

• Low–cost customization

No special additional hardware is needed to run the C language executor and application programs (*). CRT/MDI and handy operator’s panel are supported. User applications can be included in the current CNC system.

NOTE
(*): The flash ROM/DRAM capacity may have to be increased.

• Application development on a personal computer

Application programs can be developed using an ordinary personal computer. Program development, from program creation and editing to compilation/linkage, can also be performed on a personal computer. And, to a certain extent, debugging is also possible on the personal computer.

• High compatibility with C language application programs for personal computers

Microsoft Corporation’s C compiler (MS–C) is employed as the C language compiler. It is the de–facto standard C compiler for personal computers. The function library provided by the C language executor has excellent compatibility with the ANSI standards and MS–C. Therefore, application programs for ordinary personal computers can be transported to the CNC, except when they are dependent on particular hardware.

• Integration of CNC software and applications

An application program created by the machine tool builder is executed as one task of the CNC software. The application program can display its own screens in place of existing CNC screens. In addition, the application program can read and write CNC system data via libraries provided by the C language executor. This enables operation of the application program to be integrated with CNC software.

• Using the C language executor with the macro executor

The C language executor can be used with the macro executor. Not only executable macros, but also conversational macros can be used together. The screen display portion of a macro program already created by the machine tool builder can be replaced with a program coded in C. This can prevent existing software resources from becoming useless.

16.5 C LANGUAGE EXECUTER FUNCTION

Features

• Low–cost customization

No special additional hardware is needed to run the C language executor and application programs (*). CRT/MDI and handy operator’s panel are supported. User applications can be included in the current CNC system.

NOTE
(*): The flash ROM/DRAM capacity may have to be increased.

• Application development on a personal computer

Application programs can be developed using an ordinary personal computer. Program development, from program creation and editing to compilation/linkage, can also be performed on a personal computer. And, to a certain extent, debugging is also possible on the personal computer.

• High compatibility with C language application programs for personal computers

Microsoft Corporation’s C compiler (MS–C) is employed as the C language compiler. It is the de–facto standard C compiler for personal computers. The function library provided by the C language executor has excellent compatibility with the ANSI standards and MS–C. Therefore, application programs for ordinary personal computers can be transported to the CNC, except when they are dependent on particular hardware.

• Integration of CNC software and applications

An application program created by the machine tool builder is executed as one task of the CNC software. The application program can display its own screens in place of existing CNC screens. In addition, the application program can read and write CNC system data via libraries provided by the C language executor. This enables operation of the application program to be integrated with CNC software.

• Using the C language executor with the macro executor

The C language executor can be used with the macro executor. Not only executable macros, but also conversational macros can be used together. The screen display portion of a macro program already created by the machine tool builder can be replaced with a program coded in C. This can prevent existing software resources from becoming useless.
AXES CONTROL
17.1 FOLLOW UP FUNCTION

 Normally, the machine is controlled to move to a commanded position. However, when the follow up function is applied, actual position in the CNC is revised according to the move of the machine. Follow up function is activated when:

 – Emergency stop is on

 Because machine movement during the emergency stop is reported, the actual position of the machine is reflected in the CNC. Therefore, machining can be resumed after the emergency stop has been deactivated, without performing the reference point return again.

 However, when a trouble has generated in the position detection system, the system cannot follow up correctly. So present position in CNC does not become correct value.

 By input signal (follow up signal) from PMC follow up function can also be applied to:

 – Servo off status. It is also valid in cases when the machine is moved with a mechanical handle.

17.2 MECHANICAL HANDLE FEED

 It is possible to move the machine by hand, using the mechanical handle installed on the machine; not by the NC (servo motor). Move distance by the mechanical handle is followed up and actual position in the NC is revised. The mechanical handle feed is done by inputting the servo off signal of the axis fed. It is necessary, however, to specify following up of the movement in the servo off status with the follow up signal.

17.3 SERVO OFF

 Servo on/off control per axis is possible by input signals from PMC. This function is generally used with the machine clamp.

17.4 MIRROR IMAGE

 The MDI–commanded or the program–commanded move direction of each axis can be reversed and executed. Mirror image is set by CRT/MDI setting or by input signals from PMC. Mirror image can be applied to each axis.
17.5 CONTROL AXIS DETACH

It is possible to detach or attach rotary tables and attachments with this function. Switch control axis detach signal according to whether the rotary tables and attachments are attached or detached. When this signal is on, the corresponding axis is excluded from the control axes, so the servo alarm applied to the axis are ignored. The axis is automatically regarded as being interlocked. This signal is not only accepted when power turned is on, so automatic change of attachments is possible any time with this function. The same switching as with this signal can also be performed with the MDI setting.

17.6 SIMPLE SYNCHRONOUS CONTROL

The traveling 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. Simple synchronous operation is allowed in the automatic operation and in the manual operation such as manual continuous feed, manual handle feed, incremental feed, or manual reference point return. In the manual 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 PMC can be select whether the slave axis traveling is carried out based on the traveling command for that axis as in normal case or whether the slave axis traveling is carried out while synchronizing with the traveling of the master axis.

NOTE
This function can not used for Power Mate i–D (2–path control).
Positioning can be performed by butting against a mechanical stop while applying a torque limit to the servo motor. A torque limit can be set using three methods: parameter input, DI signal input, and input from the PMC. A torque limit set using any of these methods is valid in all modes (EDIT, AUTO, JOG, and so forth).

With the torque control function, a torque limit can be set using three methods:

1: Torque control based on parameter input
2: Torque control based on DI signal input
3: Torque control based on input from the PMC

See Table 17.7 for information about which torque control method is used.

A torque control method can be selected separately for each axis.

<table>
<thead>
<tr>
<th>Parameter TRQLTn (No.1855)</th>
<th>Torque limit enable signal (TRQnE)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td><strong>ON</strong></td>
</tr>
<tr>
<td>When set to 0</td>
<td>Torque control method based on DI signal input</td>
</tr>
<tr>
<td>When set to a value other than 0</td>
<td>Torque control method based on parameter input</td>
</tr>
</tbody>
</table>

**n**: 1 or 8

(1) Position and alarm check when a torque limit is applied
An excessive position error and stop/move error can be ignored by setting parameter TIP, TAL (No. 1803#0, #1) only when a torque limit is applied.
17.8 FEED STOP

This function usually checks position deviation amount during motion. If the amount exceeds the parameter set “feed stop position deviation amount”, pulse distribution and acceleration/deceleration control is stopped for the while exceeding, and move command to the positioning control circuit is stopped.

The overshoot at rapid feed acceleration is thus kept to a minimum.
The PMC can directly control any given axis, independently of the CNC. In other words, moving the tool along axes that are not controlled by the CNC is possible by entering commands, such as those specifying moving distance and feedrate, from the PMC. This enables the control of turrets, pallets, index tables and other peripheral devices using any given axes of the CNC.

Whether the CNC or PMC controls an axis is determined by the input signal provided for that particular axis.

The PMC can directly control the following operations:

1. Rapid traverse with moving distance specified
2. Cutting feed–feed per minute, with moving distance specified
3. Cutting feed–feed per revolution, with moving distance specified
4. Skip–feed per minute, with moving distance specified
5. Dwell
6. Continuous feed
7. Reference position return
8. 1st reference position return
9. 2nd reference position return
10. 3rd reference position return
11. External pulse synchronization–Main spindle
12. External pulse synchronization–first manual handle
13. External pulse synchronization–second manual handle
14. Feedrate control
15. Auxiliary function
16. Selection of the machine coordinate system
17. Torque control command

The PMC is provided with number of axis these operations using input and output signals.

By issuing commands through two paths (for Power Mate i–D) or eight paths (for Power Mate i–H), the PMC can simultaneously control multiple axes separately. Use parameter to determine which path controls which axis. Commands may be issued through one path to two or more axes, thus allowing the PMC to control multiple axes using one path.

NOTE
This function can not used for Power Mate i–D (2–path control).
17.10 TANDEM CONTROL

When enough torque for driving a large table cannot be produced by only one motor, two motors can be used for movement along a single axis. Positioning is performed by the main motor only. The sub motor is used only to produce torque. With this tandem control function, the torque produced can be doubled.

Example of operation

In general, the NC regards tandem control as being performed for one axis. However, for servo parameter management and servo alarm monitoring, tandem control is regarded as being performed for two axes.

NOTE
This function can not used for Power Mate i–D (2–path control).
18 MANUAL OPERATION
18.1 MANUAL FEED

- **Jogging**
  Each axis can be moved in the + or – direction for the time the button is pressed. Feed rate is the parameter set speed with override of: 0 – 655.34%, 0.01% step. The parameter set speed can be set to each axis.

- **Manual rapid feed**
  Each axis can be fed in a rapid feed to the + or – direction for the time the button is pressed. Rapid traverse override is also possible.

18.2 INCREMENTAL FEED

Specified move amount can be positioned to the + or – direction with the button.
Move amount of:
(least command increment) x (magnification)
can be specified. The feed rate is that of manual feed.
The possible magnifications to be specified are as follows.
×1, ×10, ×100, ×1000.

18.3 MANUAL HANDLE FEED (1ST)

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:
×1, ×10, ×M, ×N.
N is parameter set values of 0 – 1000. M is parameter set values of 1–127.
Move distance is :
(Least command increment) x (magnification)

<table>
<thead>
<tr>
<th>Increment system</th>
<th>Metric input</th>
<th>Inch input</th>
</tr>
</thead>
<tbody>
<tr>
<td>IS–B</td>
<td>0.001, 0.01, M/1000, N/1000 mm</td>
<td>0.0001, 0.001, M/10000, N/10000 inch</td>
</tr>
<tr>
<td>IS–C</td>
<td>0.0001, 0.001, M/10000, N/100000 mm</td>
<td>0.000001, 0.00001, M/1000000, N/10000000 inch</td>
</tr>
</tbody>
</table>
18.4 MANUAL HANDLE FEED (2ND, 3RD)

A 2nd, as well as 3rd manual pulse generator can be rotated to move the axis for the equivalent distance. Manual handle feed of 3 axes can be done at a time. Multiplier is common to 1st, 2nd and 3rd manual pulse generators. The third one can be connected only via the I/O Link.

18.5 HANDLE FEED IN THE SAME MODE AS FOR JOGGING

Manual handle feed can be performed in the fog feed mode. However, manual continuous–feed and manual handle–feed cannot be performed simultaneously. Manual handle–feed can be performed only when manual continuous–feed is in progress (i.e., an axis is moving).

18.6 MANUAL ABSOLUTE ON/OFF

When tool is moved by manual operation, whether to add the move distance to the absolute coordinate value in the workpiece coordinate system is selected depending on the input signal *ABSM.

When tool is moved by manual operation when *ABSM is set to 0, the move distance is added to the absolute coordinate value.

When tool is moved by manual operation when *ABSM is set to 1, the move distance is ignored, and is not added to the absolute coordinate value. In this case, the work coordinates is shifted for the amount tool was moved by manual operation.
19 AUTOMATIC OPERATION
## 19.1 OPERATION MODE

### 19.1.1 DNC Operation (Only for Power Mate i–D)

The part program can be read and executed block by block from the input device connected to the reader/puncher interface.

### 19.1.2 Memory Operation

Program registered in the memory can be executed.

### 19.1.3 MDI Operation

Multiple blocks can be input and executed by the MDI unit.
19.2
SELECTION OF EXECUTION PROGRAMS

19.2.1
Program Number Search

Program number currently in need can be searched from the programs registered in memory operating the MDI.

19.2.2
Sequence Number Search

The sequence number of the program on the currently selected memory can be searched using the MDI unit. When executing the program from half-way (not from the head) of the program, specify the sequence number of the half-way program, and the program can be executed from the half-way block by sequence number search.

19.2.3
Rewind

After program execution has ended, the program in the memory or the tape reader can be reminded to the program head, with this reset & rewind signal on. (When a portable tape reader with reels is in use)

19.2.4
External Workpiece Number Search

By specifying work numbers of 01 - 255 externally (from the machine side, etc.), 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.
19.3 ACTIVATION OF AUTOMATIC OPERATION

19.3.1 Cycle Start

Set operation mode to memory operation, MDI operation, or DNC operation, press the cycle start button, and automatic operation starts.

19.4 EXECUTION OF AUTOMATIC OPERATION

19.4.1 Buffer Register

Buffer register in CNC equivalent to one block is available for program read and control of CNC command operation intervals caused by preprocess time.
19.5 AUTOMATIC OPERATION STOP

19.5.1 Program Stop (M00, M01)
Automatic operation is stopped after executing the M00 (program stop) commanded block. When the optional stop switch on the operator’s panel is turned on, the M01 (optional stop) commanded block is executed and the automatic operation stops. The automatic operation can be restarted by the cycle start button.

19.5.2 Program End (M02, M30)
M02 or M30 marks the end of a main program. After a block in which M02 (end of program) or M30 (end of tape) is specified is executed, automatic operation is stopped, and the CNC is placed in the reset state.

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

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 pulse commanded value. Movement commanded by handle interruption can be displayed.
19.7 RETRACTION FOR TAPPING

When tapping is stopped, either as a result of an emergency stop or a reset, the tap may cut into the workpiece. The tap can subsequently be drawn out by using a PMC signal. This function automatically stores information relating to the tapping executed most recently. When a tap retraction signal is input, only drawn out of the tapping cycle is executed, based on the stored information. The tap is pulled toward the R point.

![Diagram of Retraction for Tapping]

19.8 RIGID TAPPING RETURN

When rigid tapping is stopped, either by an emergency stop or by a reset, the tap may cut into the workpiece. The tap can subsequently be drawn out by using a PMC signal. This function automatically stores information relating to the tapping executed most recently. When a tap return signal is input, only the rigid tapping cycle return operation is executed, based on the stored information. The tap is pulled toward the R point. When a return value $\alpha$ is set in a corresponding parameter, the pulling distance can be increased by $\alpha$.

![Diagram of Rigid Tapping Return]
PROGRAM TEST FUNCTIONS
20.1 ALL-AXES MACHINE LOCK

In machine lock condition, the machine does not move, but the position display is updated as if the machine were moving. Machine lock is valid even in the middle of a block.

20.2 AUXILIARY FUNCTION LOCK

This function inhibits transmitting of M, S, T function code signals and strobe signals to PMC. Miscellaneous functions M00, M01, M02, and M30 are executed even when miscellaneous function lock is applied, allowing the code signal, strobe signal, and decode signal to be transmitted normally.

20.3 DRY RUN

In the dry run mode, the tool moves at the speed obtained by multiplying the dry run speed by the override value for manual feeding, regardless of the specified cutting federate. When the rapid traverse command (G00) is used, dry run is enabled for the rapid traverse command, or rapid traverse is performed as specified, depending on the parameter settings.

20.4 SINGLE BLOCK

The program can be executed block by block under automatic operation.
The available operational devices include the setting and display unit attached to the CNC, the machine operator’s panel, and external input/output devices such as a tape reader, PPR, Handy File, Floppy Cassette, and FA Card.
21.1 SETTING AND DISPLAY UNIT

With the Power Mate i, the setting/display units listed below can be used.

(1) CRT/MDI ........................................ 21.1.1
(2) LCD/MDI ........................................ 21.1.2
(3) Picture display CRT/MDI ....................... 21.1.3
(4) Separate MDI .................................... 21.1.4
(5) Separate MDI for picture display ............. 21.1.5
(6) Separate CRT .................................... 21.1.6
(7) Separate LCD .................................... 21.1.7
(8) Handy operator’s panel ......................... 21.1.8
(9) Detachable LCD/MDI ......................... 21.1.9
(10) Touch panel display unit ................... 21.1.10
(11) Personal computer (connected via HSSB) .. 21.1.11
(12) Personal computer (connected via RS–232C) .. 21.1.12

For the outside dimensions of the setting/display units (1) through (10), see Sections 21.1.1 through 21.1.10.
21.1.1 CRT/MDI

The CRT/MDI is a cost-effective text-based setting/display unit. The display section is a 9-inch monochrome CRT, and the setting section uses a small keyboard. Two types of setting sections are available. One uses alphanumeric keys, while the other uses symbols. 400 (W) × 200 (H) × 244 (D) mm, 5.8 kg in weight

The common functions of the CRT/MDI can be used. This unit is the same as that used with the FANUC Power Mate–MODEL D/H.
The LCD/MDI is a text–based setting/display unit. The LCD/MDI is both thin and lightweight.
The display section is a 7.2–inch monochrome LCD, and the setting section uses a small keyboard.
Two types of setting sections are available. One uses alphanumeric key, while the other uses symbols.
400 (W) × 200 (H) × 124 (D) mm, 3.9 kg in weight

The common functions of the CRT/MDI can be used. This unit is the same as that used with the FANUC Power Mate–MODEL D/H.
The picture display CRT/MDI is a setting/display unit that allows bit map data such as figures and photographs to be stored and displayed on the screen. Text data created with the macro executor can be combined with stored bit map data and displayed on the screen. This means that an easy-to-use data input screen can be created.

The display section is a 9-inch monochrome CRT. The setting section uses a small keyboard.

Two types of setting sections are available. One is English key type, and the other is symbol key type.

Moreover, this unit is available in two versions; one can store 32 screens, while the other can store 64 screens.

The common functions of the CRT/MDI can be used. This unit is the same as that used with the FANUC Power Mate–MODEL D/H.
21.1.4 Separate MDI

The separate MDI is a setting unit that uses a small keyboard, which is used in combination with a separate CRT or LCD unit. A display unit and this setting unit can be oriented vertically. Two types are available. One uses alphanumeric keys, while the other uses symbols. 175 (W) × 200 (H) × 124 (D) mm, 1.3 kg in weight

The common functions of the CRT/MDI can be used. This unit is the same as that used with the FANUC Power Mate–MODEL D/H.
21.1.5 Separate MDI for Picture Display

The separate MDI for picture display is a setting unit that uses a small keyboard, which is used in combination with a separate CRT or LCD unit. This setting unit has a picture display function. (See Section 21.1.3.) A display unit and this setting unit can be oriented vertically. Two types are available. One uses alphanumeric keys, while the other uses symbols. The outside dimensions and weight are the same as those of the CRT/MDI.

The common functions of the CRT/MDI can be used. This unit is the same one as that used with the FANUC Power Mate–MODEL D/H.

21.1.6 Separate CRT

The separate CRT is a 9-inch monochrome CRT display unit, which is used in combination with a separate MDI or separate MDI for picture display. This display unit and a setting unit can be arranged vertically. 260 (W) × 200 (H) × 244 (D) mm, 3.5 kg in weight

This unit is the same as that used with the FANUC Power Mate–MODEL D/H.
21.1.7 Separate LCD

The separate LCD is a 7.2-inch monochrome LCD display unit, which is used in combination with a separate MDI. This display unit and a setting unit can be oriented vertically. 260 (W) × 200 (H) × 74 (D) mm, 1.6 kg in weight.

This unit is the same as that used with the FANUC Power Mate–MODEL D/H.
21.1.8 Handy Operator’s Panel

The handy operator’s panel provides the functions of the machine operator’s panel used for teaching, as well as functions equivalent to those of the CRT/MDI. These functions are used to display and set data such as Power Mate parameters and programs. The functions of a machine operator’s panel are implemented with ladder programs created by the customer. The handy operator’s panel has an emergency stop button and a deadman’s switch. A key sheet dedicated to each machine tool builder can be designed, so that a dedicated machine operator’s panel can be created easily. Two types of FANUC standard key sheets are available. One is for alphanumeric keys, while the other is for symbol keys. 209 (W) × 295 (H) × 80 (D) mm, 1.3 kg in weight

This unit is the same one as that used with the FANUC Power Mate–MODEL D/H.

NOTE

The handy operator’s panel does not support some of the functions provided by the CRT/MDI. The handy operator’s panel cannot be used with the Power Mate i–D (two–path control).
The following two key types are available from FANUC:

**Alphanumeric key type**  
*(A02B–0211–C020#R)*

**Symbol key type**  
*(A02B–0211–C020#S)*
21.1.9 Detachable LCD/MDI

The detachable LCD/MDI is an LCD/MDI unit (Section 21.1.2) housed in a portable dustproof case. On a transfer line, for example, a detachable LCD/MDI can be attached to (and detached from) one Power Mate \( i \) unit, then another, and so on. Two types of setting sections are available. One uses alphanumeric keys, while the other uses symbols.

410 (W) \( \times \) 235 (H) \( \times \) 145 (D) mm, 7.0 kg in weight \( (235 \text{ (D) when the cable loops are included.}) \)

This unit is the same as that used with the FANUC Power Mate–MODEL D/H.
21.1.10 Touch Panel Display Unit

The touch panel display unit can be used as an easily modifiable machine operator’s panel.
The touch panel display unit can flexibly conform to operator panel changes when a line is modified.
Moreover, all CNC screens can be displayed and operated with the touch panel display unit, so that the touch panel display unit can be used for maintenance of the Power Mate i.
For the display section, a 10.4-inch color LCD unit is used. A touch panel is used for the setting section.
The outside dimensions and weight have yet to be finalized.

NOTE
To determine when this unit will be available, contact FANUC.
A connection with a personal computer can be made via the high-speed serial bus (HSSB) to enable high-speed windows-based data transfer. An IBM-PC/AT compatible personal computer with ISA slots can be used. The HSSB interface board is installed in the personal computer. Two types of HSSB interface boards are available: one supporting one channel, and the other supporting two channels. One personal computer can support the connection of up to eight HSSB channels. (However, the number of connectable channels is restricted by the number of free ISA slots.)
A low-cost connection with a personal computer can be made using the RS–232C interface. The two types of software listed below are available for use on the PC.

1) FAPT LADDER–II
   This software enables the display and editing of ladder diagrams, and can also display the PMC status. Ladder creation and online debugging are also possible.

2) DPL/MDI operation package
   This software enables the display and editing of programs and parameters.

The personal computer can be made detachable by using a punch panel and cable manufactured by FANUC.

**NOTE**

A setting/display unit called the DPL/MDI has been used with the conventional FANUC Power Mate–MODEL D/H units. Note, however, that it cannot be used with the Power Mate i. If a setting/display unit is not attached to each Power Mate unit, prepare a notebook–type personal computer with the two types of software, described above, installed. Use the second RS–232C channel of the Power Mate i to connect the personal computer.
# 21.2 EXPLANATION OF THE KEYBOARD

## 21.2.1 For English Key

<table>
<thead>
<tr>
<th>No.</th>
<th>Key</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>(1)</td>
<td>Reset key</td>
<td>Used to reset the CNC to release an alarm or other similar state.</td>
</tr>
<tr>
<td>(2)</td>
<td>Help key</td>
<td>Used to get help with operations such as for the MDI keys, when the operator does not know what to do next.</td>
</tr>
<tr>
<td>(3)</td>
<td>Soft keys</td>
<td>The soft keys are assigned different functions depending on the application. (Soft keys are provided on the CRT or LCD.) The functions currently assigned to the soft keys are displayed on the lowermost line of the screen.</td>
</tr>
<tr>
<td>(4)</td>
<td>Address/numeric keys</td>
<td>Used to enter letters and numbers.</td>
</tr>
<tr>
<td>(5)</td>
<td>Shift key</td>
<td>Some of the address keys have two different letters. When the shift key is pressed first before pressing one of these address keys, the lower-right letter is input. When the shift key is pressed, ^ is displayed in the key input buffer indicating that the lower-right letter will be input.</td>
</tr>
<tr>
<td>No.</td>
<td>Key</td>
<td>Function</td>
</tr>
<tr>
<td>-----</td>
<td>-----</td>
<td>----------</td>
</tr>
<tr>
<td>(6)</td>
<td>Input key</td>
<td>Data input by pressing an address or numeric key is stored in the key input buffer, then displayed. When data input to the key input buffer needs to be written to the offset register, press the <code>&lt;INPUT&gt;</code> key. This key is equivalent to soft key <code>[INPUT]</code>. Either key may be used.</td>
</tr>
</tbody>
</table>
| (7) | Cancel key | Used to delete letters or numbers input to the key input buffer.  
Example)  
When N001X100Z is displayed on the key input buffer, pressing the cancel key deletes the letter Z, and N001X100 is displayed. |
| (8) | Edit keys | Used to edit programs.  
ALTER : Alter  
INSERT : Insert  
DELETE : Delete |
| (9) | Function keys | Used to switch screens for each function. |
| (10) | Cursor keys | Four cursor keys are provided.  
: Moves the cursor to the right or forwards in small units.  
: Moves the cursor to the left or backwards in small units.  
: Moves the cursor downward or forwards in large units.  
: Moves the cursor upward or backwards in large units. |
| (11) | Page-up/down keys | Page-up and page-down keys are provided.  
PAGE : Used to display the next page.  
PAGE : Used to display the previous page. |
21.2.2  For Symbol Key

Symbol keys correspond to alphanumeric keys on a one-to-one basis. For the meanings of the symbol keys, see the explanations of the corresponding keys in Sections 21.2.1 and 21.2.3.

21.2.3  Explanation of the Function Keys

The function keys select what is displayed. Each function is divided into sub-functions, and the sub-functions are selected by soft keys.

There are six function keys: POS, PROG, OFFSET, SYSTEM, MESSAGE, and CUSTOM GRAPH.

- POS: Displays the current position.
- PROG: Displays and edits a program stored in memory.
- OFFSET: Displays an offset value, custom macro variable, and data. Allows data to be input into these items.
- SYSTEM: Displays and sets a parameter and pitch error compensation value, and displays self diagnostic data.
- MESSAGE: Displays an alarm message, and alarm history.
- CUSTOM GRAPH: Not normally used. This key may be used, depending on the machine. In such a case, refer to the relevant manual provided by the machine tool builder.
On the MDI panel, seven (5 + 2) keys are provided. That is, there are five soft keys, and a next-menu key and previous-menu key are provided at both ends. The next menu key and previous menu key are used to select the functions of the soft keys. These soft keys can be assigned with various functions, according to the needs.

The following functions are mainly available via the MDI panel:

- Actual position display
- Contents of program display, program directory display (display of program number, program name, part program storage length left, number of programs left)
- Program editing
- Offset amount display and setting
- Commanded value display, MDI input
- Parameter setting and display
- Alarm message/operator message display
- Custom macro variables display and setting
- Diagnosis
- Others

This manual may refer to a display device with 5 + 2 soft keys as a 7 soft key type.
22.1 DISPLAY

Explanations

- Indication of statuses

The following data are displayed. CRT/MDI or LCD/MDI can display maximum 640 characters (40 × 16 lines).

The status of the control unit is indicated on the screen. Statuses include the state when an alarm is being activated or when the system is in the edit mode. The status line is displayed right above the soft key line.

<table>
<thead>
<tr>
<th>Operation mode (MDI, AUTO, RMT, EDIT, JOG, TJOG, THND, STEP, or ZRN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status of automatic operation (STOP, HOLD, STRT, or ****)</td>
</tr>
<tr>
<td>***** : Reset</td>
</tr>
<tr>
<td>STOP : Automatic operation is in a stopped state.</td>
</tr>
<tr>
<td>HOLD : Automatic operation is in a halt state.</td>
</tr>
<tr>
<td>STRT : Automatic operation has been started.</td>
</tr>
<tr>
<td>Axis movement/dwell (MTN, DWL, or *** )</td>
</tr>
<tr>
<td>FIN wait state (FIN or *** )</td>
</tr>
<tr>
<td>Emergency stop (--EMG--) (displayed above in 3 and 4)</td>
</tr>
<tr>
<td>Alarm status (ALM, BAT , or *** )</td>
</tr>
<tr>
<td>Clock (hh:mm:ss)</td>
</tr>
<tr>
<td>Name of the device or path currently selected</td>
</tr>
</tbody>
</table>

NOTE
The name of a device or path can be specified by the corresponding parameter with a string of up to seven characters. The characters may be numbers, letters, katakana characters, or symbols.

- Key input display

Data input via the address keys or the numerical keys are displayed at the left lower part of the screen.

- Program number, sequence number display

Program number, sequence number is displayed on the right upper part of the screen.

- Alarm display

Alarm number and its contents are displayed briefly.
• Alarm message display  
  Alarm message contents are displayed.

• Present position display  
  Relative position and position in the work coordinates are displayed in 3-times magnified characters.

• Total position display  
  Relative position, position in the work coordinates, position in the machine coordinate, and remaining move distance are displayed in one screen.

• Command value display  
  The following two displays are performed.
  - Previously commanded modal value and command value to be executed (ACTIVE)
  - Command value of the next block

• Setting (parameter set by the operator) display  
  Displays setting value.

• Tool offset amount display  
  Displays offset value. Relative position is also displayed at the same time.

• Program display  
  - Display of program for editing.
  - Display of program currently under execution.
  - Display of program list.
    - A list of program number and program name, of programs stored in the memory is displayed.
    - Used memory size and remaining memory size are also displayed.

• Parameter display

• Self diagnosis result display

• Custom macro variables display

• External operator message, external alarm message display

• Actual speed and actual spindle speed  
  - Actual feedrate per minute (mm/min or inch/min)
  - Actual spindle speed (rpm)

• Program check screen  
  The following are displayed on one screen.
  - Program number on execution
  - Sequence number on execution
  - Program text on execution
  - Current position
  - Modal G codes
  - Modal M codes
  - T code
  - Actual feedrate and spindle speed
  - Status
Displaying the alarm history

A maximum of 25 of the most recent alarms generated in CNC can be recorded. Each alarm record consists of the following items:

- Date and time
- Alarm number
- Alarm message

Any of the records can be deleted from the alarm history. In addition, the operator message history can be displayed.

```
ALARM HISTORY

00100 N00001

97. 02. 14  16:43:48
  010  IMPROPER G-CODE

97. 02. 13  8:22:21
  506   OVER TRAVEL   :+Y

97. 02. 12  20:15:43
  417   SERVO ALARM  :X AXIS DGTL PARAM

AUTO **** *** *** 09:36:48

[ ALARM ][ MSG ][ HISTORY ][ (OPRT) ]
```
22. DISPLAYING AND SETTING DATA

22.2 LANGUAGE SELECTION

The Japanese, English, German, French, Italian, and Spanish are prepared as display languages. Select the language to be displayed by parameters.

22.3 CLOCK FUNCTION

Time is displayed in the hour/minute/second format on each display screen. Some screens allow display of the year, month, and day. The custom macro system variable can be used to read the time. The time will be told through the window at PMC side.

22.4 RUN TIME & PARTS NUMBER DISPLAY

This function displays the integrated power-on time, the integrated cycle operation time, the integrated cutting time and timer on the CRT display screen. The integrated cycle operation time, the integrated cutting time and timer can be altered and preset, using the MDI.

In addition to the above, this function displays the count of the total number of parts machined, the number of parts required and the number of parts on the screen. Each time M02, M30 or a parameter set M code is executed, the count of the total in memory is incremented by 1.

If a program is prepared so as to execute M02, M30 or a parameter set M code each time one part machining is completed, the number of parts machined can be counted automatically.

If the count of the number of parts reaches the number of parts required, a signal is output to the PMC side.

It is possible to change and preset the number of parts required and the number of parts counted, using MDI.

The number of required parts and the number of counted parts can be read and written using custom macro variables.

```
SETTING (TIMER) O0000 N00000
PARTS TOTAL = 0
PARTS REQUIRED = 25
PARTS COUNT = 10
POWER ON = 0H 0M
OPERATING TIME = 0H 0M 0S
CUTTING TIME = 0H 0M 0S
FREE PURPOSE = 0H 0M 0S
CYCLE TIME = 0H 0M 0S
DATE = 1997 09/25
TIME = 16:20:30
MDI **** *** *** 16:20:30
[ OFFSET ][ SETTING ][ ][ ][ (OPRT) ]
```
### 22.5 SOFTWARE OPERATOR’S PANEL

In this function, functions of switches on the machine operator’s panel is done by operation on the MDI panel. Mode selection and jogging override, etc. can be operated by setting operation via the MDI panel with this function, thus allowing committance of corresponding switches on the machine operator’s panel.

This function is valid only when the screen is displayed with operator’s panel. Move cursor with the cursor operation keys, and select various operations, viewing the screen.

The following operations can be done via the MDI panel:

- **Group 1**: Module selection
- **Group 2**: Jogging feed axis selection, Jogging rapid traverse
- **Group 3**: Manual pulse generator feed axis selection, Manual pulse magnification selection
- **Group 4**: Jogging feed rate, Feed rate override, rapid traverse override
- **Group 5**: Optional block skip (Block delete)
  - Single block
  - Machine lock
  - Dry run
- **Group 6**: Protect key
- **Group 7**: Feed hole

There is a parameter per groups 1 to 7 shown above, which decides validity of operation function by MDI panel.
Eight general–purpose switches are provided. Each switch can be named using an arbitrary alphanumeric character string of no more than eight characters.
22.6 DIRECTORY DISPLAY OF FLOPPY CASSETTE

File names in the floppy cassette (FANUC CASSETTE F1) and program file (FANUC PROGRAM FILE Mate can be listed on the display (directory display). Each file name of up to 17 letters can be displayed in directory display.

Files in the floppy cassette are:
Part program, parameter/pitch error compensation data, tool compensation data, and etc.

When part program in part program memory is written into the floppy cassette, program number can be given to it as a file name. When NC parameter is written into the floppy cassette, “PARAMETER” is given them as a fixed name. When tool compensation data is written into the floppy cassette, “OFFSET” is given to it as a fixed name.

<table>
<thead>
<tr>
<th>NO.</th>
<th>FILE NAME</th>
<th>(METER) VOL</th>
</tr>
</thead>
<tbody>
<tr>
<td>0001</td>
<td>PARAMETER</td>
<td>46.1</td>
</tr>
<tr>
<td>0002</td>
<td>ALL.PROGRAM</td>
<td>12.3</td>
</tr>
<tr>
<td>0003</td>
<td>00001</td>
<td>1.9</td>
</tr>
<tr>
<td>0004</td>
<td>00002</td>
<td>1.9</td>
</tr>
<tr>
<td>0005</td>
<td>00003</td>
<td>1.9</td>
</tr>
<tr>
<td>0006</td>
<td>00004</td>
<td>1.9</td>
</tr>
<tr>
<td>0007</td>
<td>00005</td>
<td>1.9</td>
</tr>
<tr>
<td>0008</td>
<td>00010</td>
<td>1.9</td>
</tr>
<tr>
<td>0009</td>
<td>00020</td>
<td>1.9</td>
</tr>
</tbody>
</table>

EDIT **** *** *** 09:36:48
[ F SRH ] [ READ ] [ PUNCH ] [ DELETE ] [ ]
22.7
SCREENS FOR SERVO DATA AND SPINDLE DATA

22.7.1 Servo Setting Screen

On the servo setting screen, parameters required for standard initialization of the servo motor are listed. The parameters can also be set.

```
SERVO SETTING O0000 N00000
X AXIS       Y AXIS
INITIAL SET BIT 00000111 00000001
MOTOR ID NO.  12          12
AMR          00011111 00011111
CMR          2            2
FEEDGEAR N    3            3
(N/M)        10           10
DIRECTION SET 111          111
VELOCITY PULSE NO.     8000     8000
POSITION PULSE NO.     8000     8000
REF COUNTER     8000     8000

MDI **** *** *** *** 09:36:48
[ SV.SET ][ SV.TUN ][      ][      ][ (OPRT) ]
```

22.7.2 Servo Adjustment Screen

On the servo adjustment screen, parameters required for basic adjustment of the servo motor and statuses being monitored are listed for each axis.

```
SERVO SETTING O1000 N00000
X AXIS
(PARAMETR) (MONITOR)
FUNC.BIT 00110100 ALARM 1 00110100
LOOP GAIN 3000 ALARM 2 00110100
TUNING ST. 1 ALARM 3 00000000
SET PERIOD 50 ALARM 4 00000000
INT.GAIN 251 LOOP GAIN 3000
PROP.GAIN -2460 POS ERROR 100
FILTER 2450 CURRENT % 50

>_
AUTO STAT MTN *** *** 09:36:48
[ SV.SET ][ SV.TUN ][      ][      ][ (OPRT) ]
```
22.7.3 Spindle Setting Screen

On the spindle setting screen, parameters required for standard initialization of the serial spindle are listed. The parameters can also be set. This screen is only for the main spindle connected to the first amplifier.

<table>
<thead>
<tr>
<th>SPINDLE SETTING</th>
<th>00000 N00000</th>
</tr>
</thead>
<tbody>
<tr>
<td>GEAR SELECT</td>
<td>:1</td>
</tr>
<tr>
<td>SPINDLE</td>
<td>:1</td>
</tr>
<tr>
<td>(PARAMETER)</td>
<td></td>
</tr>
<tr>
<td>GEAR RATIO</td>
<td>50</td>
</tr>
<tr>
<td>MAX SPINDLE SPEED</td>
<td>3000</td>
</tr>
<tr>
<td>MAX MOTOR SPEED</td>
<td>6000</td>
</tr>
<tr>
<td>MAX C AXIS SPEED</td>
<td>100</td>
</tr>
</tbody>
</table>

>_

MDI **** *** *** 09:36:48
[ SP.SET ][ SP.TUN ][ SP.MON ][    ][ (OPRT) ]

22.7.4 Spindle Adjustment Screen

On the spindle adjustment screen, parameters required for basic adjustment of the serial spindle and statuses being monitored are listed. The screen is only for the main spindle connected to the first amplifier.

<table>
<thead>
<tr>
<th>SPINDLE TUNING</th>
<th>01000 N00000</th>
</tr>
</thead>
<tbody>
<tr>
<td>OPERATION</td>
<td>: SYNCHRONIZATION CONTROL</td>
</tr>
<tr>
<td>GEAR SELECT</td>
<td>: 1</td>
</tr>
<tr>
<td>SPINDLE</td>
<td>: S11</td>
</tr>
<tr>
<td>(PARAMETER)</td>
<td>(MONITOR)</td>
</tr>
<tr>
<td>PROP.GAIN</td>
<td>-2460</td>
</tr>
<tr>
<td>INT.GAIN</td>
<td>241</td>
</tr>
<tr>
<td>LOOP GAIN</td>
<td>3000</td>
</tr>
<tr>
<td>MOTOR VOLT</td>
<td>30</td>
</tr>
<tr>
<td>ZRN GAIN %</td>
<td>100</td>
</tr>
<tr>
<td>REF.SHIFT</td>
<td>2046</td>
</tr>
</tbody>
</table>

>_

AUTO STAT MTN *** *** 09:36:48
[ SP.SET ][ SP.TUN ][ SP.MON ][    ][ (OPRT) ]
The configurations of software and hardware required for maintenance of the CNC are displayed. The system configuration display function provides the following three screens:

- Slot information screen
- Software information screen
- Hardware (module) information screen

### Slot Information

<table>
<thead>
<tr>
<th>Slot No.</th>
<th>Module ID</th>
<th>Software ID</th>
<th>Software Series</th>
<th>Software Edition</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td>10D5 :40</td>
<td>80A0</td>
<td>0002</td>
<td></td>
</tr>
<tr>
<td>01</td>
<td>00CF :66</td>
<td>B435</td>
<td>0001</td>
<td></td>
</tr>
<tr>
<td>03</td>
<td>019D :41</td>
<td>4068</td>
<td>0001</td>
<td></td>
</tr>
</tbody>
</table>

Auto: 18:46:43

#### Software Information

<table>
<thead>
<tr>
<th>Software Type</th>
<th>Software Series</th>
<th>Software Edition</th>
</tr>
</thead>
<tbody>
<tr>
<td>SYSTEM</td>
<td>88F0</td>
<td>0001</td>
</tr>
<tr>
<td>SERVO</td>
<td>9090</td>
<td>0001</td>
</tr>
<tr>
<td>PMC(SYS)</td>
<td>407B</td>
<td>0001</td>
</tr>
<tr>
<td>PMC(LAD)</td>
<td>FS16</td>
<td>0001</td>
</tr>
<tr>
<td>MACRO LIB</td>
<td>BZG1</td>
<td>0001</td>
</tr>
<tr>
<td>BOOT</td>
<td>881I</td>
<td>0001</td>
</tr>
</tbody>
</table>

Auto: 19:14:23

1. Software type
2. Software series
3. Software edition
4. Contents of ROM (system ROM only)
Hardware (module) information

The slot number, board name, modules mounted on the board are displayed for each slot.

1 Slot number (The slot number corresponds to the number displayed on the slot information screen.)
2 Name of the PC board inserted in the slot
3 Hardware (modules) mounted on the PC board
4 Types of hardware (modules), mounted/not mounted
22.9 HELP FUNCTION

When an alarm occurs, or when the operator is not certain what to do next, pressing the HELP key on the MDI panel displays detailed alarm information or instructions for operation. One of the following three screens can be displayed:

- On the alarm detail screen, detailed information on the alarm currently activated is displayed. Using this information, the operator can identify the cause of the alarm and what action to take. Any alarm information can be displayed on this screen.

- On the operation instruction screen, when the operator is not sure of what to do next during CNC operation (i.e., program editing and data input/output) necessary instructions are displayed.

- Parameter numbers are listed on the parameter list screen. When the number of the parameter to be set or referenced is unknown, bring up this screen.

☐ An alarm detail screen for when an alarm (P/S 010) is activated.

HELP (ALARM DETAIL) O1234 N00001

NUMBER: 094
M'SAGE: P TYPE NOT ALLOWED (COORD CHG)
FUNCTION: RESTART PROGRAM
ALARM:
WHEN COORDINATE SYSTEM SETTING IS CONDUCTED AFTER HOLDING AUTOMATIC OPERATION, P-TYPE (WHEN TOOL IS DAMAGED) PROGRAM RESTART CANNOT BE EXECUTED.

S 0 T0000
AUTO **** *** *** ALM 09:36:48
[ ALAM ][ OPR ][ PARA ][] (OPRT)

☐ Parameter list screen

HELP (PARAMETER TABLE) O1234 N00001

1/4
*SETTING (NO. 0000 – )
*READER/PUNCHER INTERFACE (NO. 0100 – )
*AXIS CONTROL/SETTING UNIT (NO. 1000 – )
*COORDINATE SYSTEM (NO. 1200 – )
*STROKE LIMIT (NO. 1300 – )
*FEED RATE (NO. 1400 – )
*ACCEL/DECELERATION CTRL (NO. 1600 – )
*SERVO RELATED (NO. 1800 – )
*DI/DO (NO. 3000 – )

S 0 T0000
AUTO **** *** *** 09:36:48
[ ALAM ][ OPR ] PARA [] (OPRT)


☐ Operation instruction screen

<table>
<thead>
<tr>
<th>HELP (OPERATION METHOD)</th>
<th>O1234 N00001</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;&lt;1. PROGRAM EDIT&gt;&gt;</td>
<td>1/4</td>
</tr>
<tr>
<td>*DELETE ALL PROGRAMS</td>
<td></td>
</tr>
<tr>
<td>MODE : EDIT</td>
<td></td>
</tr>
<tr>
<td>SCREEN: PROGRAM</td>
<td></td>
</tr>
<tr>
<td>OPR : (0-9999) – &lt;DELETE&gt;</td>
<td></td>
</tr>
</tbody>
</table>

*DELETE ONE PROGRAM

| MODE : EDIT | |
| SCREEN: PROGRAM | |
| OPR : (O+PROGRAM NUMBER) – <DELETE> | |

>__ S 0 T0000

AUTO **** *** *** 09:36:48

[ ALAM ] [ OPR ] [ PARA ] [ ] [ (OPRT) ]
22.10 DATA PROTECTION KEY

A data protection key can be installed on the machine side for protection of various NC data. The following three input signals are offered, according to type of data to be protected.

- **KEY 1**
  Allows input of tool compensation amount.

- **KEY 2**
  Allows setting data input and macro variable input.

- **KEY 3**
  Allows part program input and editing.

- **KEY 4**
  Allows input of PMC data (counter, data table).

22.11 DISPLAYING OPERATION HISTORY

This function displays a history of the key and signal operations, performed by the operator, upon the occurrence of a failure or alarm. The history can also be displayed for previously generated alarms. The following history data is recorded:

- MDI key/soft key operation sequences
  Example: A to Z, <POS>, <PAGE↑>, [SF1]

- On/off status transitions of selected input and output signals
  Example: G0000.7↑, SBK↑

- NC alarm information
  Example: P/S0010

- Time (date, time) stamp
  Example: 97/09/25
  09:27:55

The history data can be output to an input/output device, connected via the RS–232–C interface. Previously output history data can be input from an input/output device.
22.12 PERIODIC MAINTENANCE SCREEN

The periodic maintenance screen shows the current statuses of those consumables that require periodic replacement (backup battery, LCD backlight, touch pad, etc.). An item whose service life has expired is indicated by the machine run time or the like.

<table>
<thead>
<tr>
<th>ITEM NAME</th>
<th>REMAIN</th>
</tr>
</thead>
<tbody>
<tr>
<td>BATTERY FOR CONTROLLER</td>
<td>0H</td>
</tr>
<tr>
<td>BATTERY FOR PULSECODER</td>
<td>5000H</td>
</tr>
<tr>
<td>LCD BACK LIGHT</td>
<td>10000H</td>
</tr>
<tr>
<td>COOLANT</td>
<td>720H</td>
</tr>
</tbody>
</table>

>_<
EDIT **** *** *** 19:27:05
[ ] [MAINTE ] [ ] [ ] [(OPRT) ]

22.13 MAINTENANCE INFORMATION SCREEN

The history of the maintenance carried out by FANUC service personnel and machine tool builder can be recorded via the screen. The screen has the following features:

- Alphabetical characters can be input from MDI. (Half-size kana can be input only when Japanese display is selected.)
- The recording screen can be scrolled, line by line.
- Edited maintenance information can be read and punched.
- Data can be stored into flash ROM.
- Full-size (shift JIS) codes can be displayed. (Input codes are read only.)
23.1 FOREGROUND EDITING

The following part program storage and editing is possible

- **Program tape registration to the memory**
  - Single program registration
  - Multi program tape registration

- **Program input via MDI**

- **Program deletion**
  - Single program deletion
  - All programs deletion
  - Multi programs deletion by specification the range

- **Program punching**
  - Single program punching
  - All programs punching
  - Multi programs punching by specification the range

- **Program editing**
  - Change
    - Word change
    - Change of 1-word to multi-words
  - Insertion
    - Word insertion
    - Multi words, and multi blocks insertion
  - Deletion
    - Word deletion
    - Deletion to EOB
    - Deletion to the specified word

- **Part program collation**
  Collation of program stored in the memory and program on the tape can be done.

- **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.

23.2 BACKGROUND EDITING

Part program storage and editing can be done during machining. The same functions as foreground editing can be performed.
However, it is not possible to delete all programs at one time.
The following editing is possible.

- **Conversion**
  - Address conversion
    An address in the program can be converted to another address. For example address X in the program can be converted to address Y.
  - Word conversion
    A word in the program can be converted to another word. For example, a programmed M03 can be converted to M04.

- **Program copy**
  A part or all of a program can be copied to make a new program.

- **Program move**
  A part or all of a program can be moved to make a new program.

- **Program merge**
  A new program can be created by merging two programs.

Number of registered programs can be selected from the following: 63, 125, 200, or 400.

The following part program storage length can be selected: 10, 20, 40, 80, or 160 m.

Program can be prepared by storing machine position obtained by manual operation in the memory as program position. Data other than the coordinate value (M codes, G codes, feed rates, etc.) are registered in the memory by the same operation as part program storage and editing.

Part program registration and punch can be commanded externally.

- **Program registration**
  A part program can be registered in memory through the connected input device for background editing using the external read start signal.

- **Program punch**
  A part program can be punched through the connected output device for background editing using the external punch start signal.
24 DIAGNOSIS FUNCTIONS
24.1
SELF DIAGNOSIS
FUNCTIONS

The NC checks the following itself.

- Abnormality of detection system
- Abnormality of position control unit
- Abnormality of servo system
- Overheat
- Abnormality of CPU
- Abnormality of ROM
- Abnormality of RAM
- Abnormality in data transfer between MDI
- Abnormality of part program storage memory
- Abnormality in tape reader read function
- Abnormality in data transfer between PMC

Input/output signals from PMC to CNC, or vice versa, and inner status of the NC can be displayed.
The NC has the following input/output data. These data are input/output via various input/output devices as CRT/MDI, tape reader, etc.

- **Input data**
  The NC has the following input data.
  - Part program
  - Tool compensation amount
  - Setting data
  - Custom macro common variable
  - Pitch error compensation data
  - Parameters

- **Output data**
  The NC has the following output data.
  - Part program
  - Tool compensation amount
  - Setting data
  - Custom macro common variable
  - Pitch error compensation data
  - Parameters
25. DATA INPUT/OUTPUT

25.1 RS–232–C INTERFACES

The following can be input/output via the RS–232–C interface.

- Part program registration/output
- Tool offset amount input/output
- Custom macro common variable input/output
- Pitch error compensation data input/output
- Parameter punch input/output

Usually, when parameters are entered or output through an external I/O unit, the parameter screen is used; when programs are entered or output through an external I/O unit, the program screen is used. Thus, a different screen is usually used for each type of data. However, when the ALL IO screen is used, this screen alone can handle the input/output of all programs, parameters, offset data, and macro variables.

The Power Mate i has two RS–232C channels. The first channel is used for data input/output using the FANUC Handy File and for applications such as a macro executor. The second channel is used for maintenance with a notebook-type personal computer (FAPT LADDER–II, DPL/MDI operation package).

NOTE

The second RS–232C channel does not have control lines such as RS, CS, ER, DR, and CD. This means that only those units that do not use control lines can be connected to this channel.

25.2 INPUT/OUTPUT DEVICES

The following Input/Output devices are prepared, which are connectable to the RS–232–C interface.

25.2.1 FANUC Handy File

The FANUC Handy File is a compact multi functional input/output floppy disk unit for use with various types of FA equipment. Programs can be transferred or edited through operations performed directly on the Handy File or through remote operation from connected equipment.

Compared with media such as paper tape, a 3.5" floppy disk is both compact and durable, and eliminates noise during input/output. Programs with a total capacity of up to 1.44 MB (equivalent to about 3600 m paper tape) can be saved on a single floppy disk.
Files on a memory card can be referenced, and different types of data such as part programs, parameters, and offset data on a memory card can be input and output in text file format.

The major functions are listed below.

- Displaying a directory of stored files
  The files stored on a memory card can be displayed on the directory screen.

- Searching for a file
  A search is made for a file on a memory card and, if found, it is displayed on the directory screen.

- Reading a file
  Text–format files can be read from a memory card.

- Writing a file
  Data such as programs can be stored to a memory card in text file format.

- Deleting a file
  A file can be selected and deleted from a memory card.
25.3.1 Batch Save and Restore Operations based on Memory Cards

Outline

The contents of the memory of the Power Mate i can be saved to a memory card, and the contents of a memory card can be loaded back into the memory of the Power Mate i as a batch. When a display unit such as a CRT unit is not connected, these operations can be performed by using the seven-segment LED unit, rotary switch, and push button on the front of the Power Mate i.

Batch memory save operation

SRAM data such as parameters and NC programs, ladder programs, and C executor and macro executor data and programs can be saved in file format to a memory card as a batch. (The contents of system files cannot be saved as a batch.) An alarm is issued if the write protect switch of the memory card is not released. (Saved data can also be loaded back by a boot operation.)

Batch memory restore operation

SRAM data, ladder programs, C executor and macro executor data and programs saved to a memory card as a batch can be loaded back into the memory of the Power Mate i as a batch. An alarm is issued if the write protect switch of the memory card is not applied.
25.4 DATA INPUT/OUTPUT FUNCTION BASED ON THE I/O LINK AND DATA INPUT/OUTPUT FUNCTION B BASED ON THE I/O LINK

Power Mate programs, parameters, macro variables, and diagnostic (PMC) data can be entered to or output from Series 16/18/21 through the FANUC I/O Link.

With FANUC I/O Link, slaves in groups 0 to 15 can be connected, enabling data input/output to and from a maximum of 16 Power Mates. The ordinary data input/output function based on I/O Link can only be executed in the foreground. When data input/output function B based on I/O Link is used, the external I/O device control function is associated with I/O Link so that an input/output group number and program number can be specified from the PMC. The external I/O device control function operates in the background. Therefore, when no other background operation is being performed, data can be input/output, regardless of the NC mode and the currently selected screen.

The programs, parameters, macro variables, and diagnostic (PMC) data of a slave Power Mate are stored in tape format within the part program storage length; these data items are stored as master program data in a master program memory area.

Data input/output can be performed between the master and a slave of a selected group. When the ordinary data input/output function based on I/O Link is used, a group is selected by means of parameter setting. When data input/output function B based on I/O Link is used, a group is selected by issuing the DI signal. Data input/output cannot be performed between the master and more than one group at a time.
When the servo amplifier β series with I/O Link is used as an additional (slave) axis of the CNC, the power mate CNC manager enables the display and setting of data from the CNC. Up to eight slave units can be connected.

The power motion manager supports the following functions:
1) Current position display (absolute/machine coordinate)
2) Parameter display and setting
3) Diagnosis
4) System configuration screen
5) Alarm

The sample screen shows the data for four units.

The analog input function digitizes an analog signal ranging from –10 DC to +10 V applied externally, and outputs the resulting signed 12-bit data as the F signal of the PMC. The resolution is 4.9 mV (uninsulated type). By specifying a parameter, an offset can be applied to an input voltage.
26 SAFFETY FUNCTIONS
26.1
EMERGENCY STOP

26.1.1 General
Using the emergency stop signal effectively enables the design of safe machine tools.
The emergency stop signal is provided to bring a machine tool to an emergency stop. It must be input to the Power Mate i, servo amplifier, and spindle amplifier. An emergency stop signal is usually generated by closing the B contact of a pushbutton switch.

26.1.2 Power Mate i
When the emergency stop signal (*ESP) contact is closed, the Power Mate i enters the emergency stop released state, such that the servo and spindle motors can be controlled and operated. When the emergency stop signal (*ESP) contact opens, the Power Mate i is reset and enters the emergency stop state, and the servo and spindle motors are decelerated to a stop.

The Power Mate i must be supplied with an emergency stop signal from either the built–in I/O module or the I/O module supporting the FANUC I/O Link.
26.1.3 Servo/Spindle

Shutting off the servo amplifier power causes a dynamic brake to be applied to the servo motor. Even when a dynamic brake is applied, however, a servo motor attached to a vertical axis can move under the force of gravity. To overcome this problem, use a servo motor with a brake.

While the spindle motor is running, shutting off the motor–driving power to the spindle amplifier allows the spindle motor to continue running under its own inertia, which is quite dangerous. When the emergency stop signal (*ESP) contact opens, it is necessary to confirm that the spindle motor has been decelerated to a stop, before the spindle motor power is shut off.

The FANUC servo amplifier α/β series products are designed to satisfy the above requirements. The emergency stop signal should be input to the power supply module (called the PSM) for α series. The PSM outputs a motor power MCC control signal, which can be used to switch the power applied to the power supply module on and off.

For the servo amplifier β series, input an emergency stop signal to the servo amplifier, and control the turn–on/off of the motor power using an external circuit.

The Power Mate i is designed to detect overtravel by using a software limit function. Normally, no hardware limit switch is required to detect overtravel. If the machine goes beyond a software limit because of a servo feedback failure, however, it is necessary to provide a stroke end limit switch, connected so that the emergency stop signal can be used to stop the machine.

Fig. 26.1.3 shows an example showing how to use the emergency stop signal with this Power Mate i and α series servo amplifier.
NOTE

1. To use a spindle motor and amplifier produced by a manufacturer other than FANUC, refer to the corresponding documentation as well as this manual. Design the emergency stop sequence such that, if the emergency stop signal contact opens while the spindle motor is rotating, the spindle motor is decelerated until it stops.

2. The servo amplifier β series has no motor power MCC control signal. To turn on and off the MCC, use an emergency stop signal from an external circuit.
26.2
OVERTRAVEL
FUNCTIONS

26.2.1
Overtravel

When the movable section has gone beyond the stroke end, a signal is output, the axis decelerates to a stop, and overtravel alarm is displayed. All directions on all axes has overtravel signals.

26.2.2
Stored Stroke Check 1

The movable section of the machine is parameter set in machine coordinates value. If the machine moves beyond the preset range, it decelerates to a stop and alarm is displayed. (This function is valid after manual reference point return at power on.)

This function can be used instead of hardware overtravel limit switch. When both is equipped with, both are valid.

Unlike overtravel detection, stored stroke check 1 checks whether the distance between the current position and that at which the tool will be stopped after deceleration exceeds the limit.

The shaded part is the inhibition area.

26.2.3
Externally Setting the Stroke Limit

When a new tool is mounted, position the tip of the tool on the two corners of the limit area, and specify the machine coordinates of the corners in the parameters for stroke limit 1. The machine coordinates are stored in the CNC as the limit positions. Then input signals for setting the stroke limit. Stroke limit setting signals are provided for each axis and each direction. Checking of the stroke limit can also be selected by turning on or off the limit release signal common to all axes.
26.3 INTERLOCK

26.3.1 Interlock per Axis
Axis feed specified to each axis can be stopped separately. If interlock is specified to any of the moving axis during cutting feed, all axes of the machine movement will decelerate to a stop. When interlock signal is reset, the moving starts.

26.3.2 All Axes Interlock
Feed of all axes can be inhibited. When all axes interlock is commanded during move, it decelerates and stops. When all axes interlock signal is reset, the moving restarts.

26.3.3 Interlock for Each Axis Direction
Feeding of a specific axis in a specific direction can be inhibited independently of other axes. If the interlock signal is input to any of the axes during a cutting feed operation, all axes decelerate and come to a stop. When the interlock signal for each axis direction is released, the axes start moving again.

26.4 ABNORMAL LOAD DETECTION
When a cutting tool collides with the machine body or is damaged during cutting, the load torque applied to the servo motors is larger than during normal feeding or cutting. The abnormal load detection function calculates the load torque and transfers the value from the CNC to the PMC. If the load torque is larger than the value set in a parameter, the function stop the motor or reverses the motor rotation to retract the tool by the distance set in a parameter. In this way, damage to the machine is prevented.
27 STATUS OUTPUT
<table>
<thead>
<tr>
<th>Signal Code</th>
<th>Signal Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>27.1</td>
<td>NC READY SIGNAL</td>
</tr>
<tr>
<td>27.2</td>
<td>SERVO READY SIGNAL</td>
</tr>
<tr>
<td>27.3</td>
<td>REWINDING SIGNAL</td>
</tr>
<tr>
<td>27.4</td>
<td>ALARM SIGNAL</td>
</tr>
<tr>
<td>27.5</td>
<td>DISTRIBUTION END SIGNAL</td>
</tr>
<tr>
<td>27.6</td>
<td>AUTOMATIC OPERATION SIGNAL</td>
</tr>
<tr>
<td>27.7</td>
<td>AUTOMATIC OPERATION START SIGNAL</td>
</tr>
<tr>
<td>27.8</td>
<td>FEED HOLD SIGNAL</td>
</tr>
<tr>
<td>27.9</td>
<td>RESET SIGNAL</td>
</tr>
<tr>
<td>27.10</td>
<td>IN–POSITION SIGNAL</td>
</tr>
<tr>
<td>27.11</td>
<td>MOVE SIGNAL</td>
</tr>
</tbody>
</table>

**27.1 NC READY SIGNAL**
This signal is sent to the PMC when NC power is on and control becomes possible. Sending of this signal will be stopped when NC power is turned off.

**27.2 SERVO READY SIGNAL**
This signal is sent to the PMC when the servo system becomes operatable. Axes necessary to be braked must be braked when this signal is not sent.

**27.3 REWINDING SIGNAL**
This signal shows that tape reader or main program in memory is rewinding.

**27.4 ALARM SIGNAL**
This signal is transmitted when the NC comes under an alarm status.

**27.5 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.

**27.6 AUTOMATIC OPERATION SIGNAL**
This signal is sent out when it is under automatic operation.

**27.7 AUTOMATIC OPERATION START SIGNAL**
This signal is sent out when automatic operation is being activated.

**27.8 FEED HOLD SIGNAL**
This signal is sent out when automatic operation is held by feed hold.

**27.9 RESET SIGNAL**
This signal is sent out to show that the NC has been reset.

**27.10 IN–POSITION SIGNAL**
This signal shows that an axis is under in–position status. This signal is output for all axes.

**27.11 MOVE SIGNAL**
This signal shows that an axis is moving. This signal is sent out for every axis. This move signal can be combined with the interlock signal to automatically clamp and unclamp the machine, or control on/off of the lubricating oil.
27.12 AXIS MOVE DIRECTION SIGNAL

This signal is output to show move direction of each axis. This signal is output for each axis.

27.13 RAPID TRAVERSING SIGNAL

This signal shows that the move command is done under rapid traverse.

27.14 TAPPING SIGNAL (ONLY FOR Power Mate i–D)

This signal is output to show that the machine is under tapping cycle (G74, G84) is under operation.

27.15 CONSTANT SURFACE SPEED CONTROL SIGNAL (ONLY FOR Power Mate i–D)

This signal shows that the machine is under constant surface speed control mode (G96).

27.16 INCH INPUT SIGNAL

This signal shows that input is done under inch input mode (G20).

27.17 DI STATUS OUTPUT SIGNAL

To inform the exterior of the states of software operator’s panel, which are set via MDI, and machine operator’s panel, following DI state output signals are sent.

- Mode-select check signal
- Mirror-image check signal

27.18 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-axis and the operating range (machine coordinates) within which the position switch signal is output.

Up to ten position switch signals can be output.
With the external data input function, data can be sent to the CNC externally, for example, from the machine to perform certain operations. The external data input is as follows.

- External program number search
- External alarm message
- External operator message
28.1 EXTERNAL PROGRAM NUMBER SEARCH

A program number from 1 - 9999 can be given from outside to the NC to call the corresponding program from the NC memory. In machines with automatic loading function of various workpiece, this function can be used to automatically select and execute program suitable to the workpiece.

28.2 EXTERNAL ALARM MESSAGE

By sending alarm number from outside, the NC is brought to an alarm status; an alarm message is sent to the NC, and the message is displayed on the screen of the NC. Reset of alarm status is also done with external data.

Up to 4 alarm numbers and messages can be sent at a single time. Alarms 0 to 999 can be sent. To distinguish these alarms from other alarms, the CNC displays them by adding 1000 to each alarm number. The messages of up to 32 characters can be sent together with an alarm.

28.3 EXTERNAL OPERATOR’S MESSAGE

Message to the operator is given from outside the NC, and the message is displayed.

The message is sent after the message number (0 to 999). Only one message with message number can be sent at a single time. Maximum 255 characters can be used for a single message.

The message numbers 0 to 99 are displayed along with the message. To distinguish these alarms from other alarms, the CNC displays them by adding 2000 to each alarm number. When a message from 100 to 999 is displayed, the message number is not displayed; only its text is displayed. An external data will clear the operator messages.
When the PMC inputs the code signal corresponding to a key on the MDI panel to the CNC, the code signal can be input in the same way as with actual operation of the key on the MDI panel. For example, this function is usable in the following case: After allowing to travel the tool at an arbitrary machining position by using the playback function, when to store its positions as the program command, X, Y, Z, <SHIFT>, etc. must be input via key operations. However, these operations can be realized simply by depressing a switch on the operator’s panel at the machine side. When the switch is pressed, the PMC inputs code signals corresponding to keys X, Y, Z, and <SHIFT> to the CNC. This produces the same results as with actual key operations.
30 PERSONAL COMPUTER FUNCTION

The open CNC allows the machine tool builder to incorporate a high–level man–machine interface, such as conversational automatic programming and conversational operation that makes maximum use of the machine tool builder’s know–how.

With the high–speed serial bus, the Power Mate i can be connected to a commercially available IBM compatible personal computer or intelligent terminal to allow massive amounts of data to be sent at high speed. A personal computer can also be connected via the RS–232C interface. When compared with the high–speed serial bus, smaller amounts of data are sent at lower speed via RS–232C interface, but RS–232C is cheaper to implement.
The high-speed serial bus is a serial interface used to transfer data at high speed between the CNC control unit and a personal computer installed on the operator panel side.

By installing a dedicated interface board in a commercially available IBM PC-compatible personal computer, the CNC control unit can be connected to the personal computer via the high-speed serial bus. The high-speed serial bus has the following features:

- Large amounts of data can be transferred between the personal computer and CNC control unit at high speed.
- A highly reliable optical fiber cable is used for connection.
- The machine tool builder can select an appropriate personal computer according to the specifications of the machine system.

### Hardware for open CNC (system with commercially available personal computer connected to CNC via high-speed serial bus)

<table>
<thead>
<tr>
<th>Item</th>
<th>Specification</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interface board on CNC</td>
<td>For option slot</td>
<td></td>
</tr>
<tr>
<td>Interface board on personal computer</td>
<td>ISA specification</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Power supply: +5 V only</td>
<td></td>
</tr>
<tr>
<td>Connection cable</td>
<td>Optical fiber cable</td>
<td>Maximum length: 50 m</td>
</tr>
<tr>
<td>Personal computer requirements</td>
<td>CPU: 486 or better</td>
<td>The installation environment shall satisfy the conditions described in the manual supplied with the personal computer.</td>
</tr>
<tr>
<td></td>
<td>At least one ISA slot</td>
<td></td>
</tr>
</tbody>
</table>
APPENDIX
## RANGE OF COMMAND VALUE

### Linear axis

- In case of millimeter input, feed screw is millimeter

<table>
<thead>
<tr>
<th></th>
<th>Increment system</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>IS–A</td>
</tr>
<tr>
<td>Least input increment</td>
<td>0.01 mm</td>
</tr>
<tr>
<td>Least command increment</td>
<td>0.01 mm</td>
</tr>
<tr>
<td>Max. programmable dimension</td>
<td>±999999.99 mm</td>
</tr>
<tr>
<td>Max. rapid traverse</td>
<td>240000 mm/min</td>
</tr>
<tr>
<td>Feedrate range <strong>NOTE</strong></td>
<td>1 to 240000 mm/min</td>
</tr>
<tr>
<td>Incremental feed</td>
<td>0.01, 0.1, 1, 10 mm/step</td>
</tr>
<tr>
<td>Tool compensation</td>
<td>0 to ±9999.99 mm</td>
</tr>
<tr>
<td>Dwell time</td>
<td>0 to 999999.999 sec</td>
</tr>
</tbody>
</table>

- In case of inch input, feed screw is millimeter

<table>
<thead>
<tr>
<th></th>
<th>Increment system</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>IS–A</td>
</tr>
<tr>
<td>Least input increment</td>
<td>0.001 inch</td>
</tr>
<tr>
<td>Least command increment</td>
<td>0.01 mm</td>
</tr>
<tr>
<td>Max. programmable dimension</td>
<td>±999999.99 inch</td>
</tr>
<tr>
<td>Max. rapid traverse</td>
<td>240000 mm/min</td>
</tr>
<tr>
<td>Feedrate range <strong>NOTE</strong></td>
<td>0.01 to 9600 inch/min</td>
</tr>
<tr>
<td>Incremental feed</td>
<td>0.001, 0.01, 0.1, 1 inch/step</td>
</tr>
<tr>
<td>Tool compensation</td>
<td>0 to ±999.999 inch</td>
</tr>
<tr>
<td>Dwell time</td>
<td>0 to 999999.9999 sec</td>
</tr>
</tbody>
</table>
### In case of inch input, feed screw is inch

<table>
<thead>
<tr>
<th>Increment system</th>
<th>IS–A</th>
<th>IS–B</th>
<th>IS–C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Least input increment</td>
<td>0.001 inch</td>
<td>0.0001 inch</td>
<td>0.00001 inch</td>
</tr>
<tr>
<td>Least command increment</td>
<td>0.001 inch</td>
<td>0.0001 inch</td>
<td>0.00001 inch</td>
</tr>
<tr>
<td>Max. programmable dimension</td>
<td>±99999.999 inch</td>
<td>±9999.9999 inch</td>
<td>±9999.9999 inch</td>
</tr>
<tr>
<td>Max. rapid traverse NOTE</td>
<td>9600 inch/min</td>
<td>9600 inch/min</td>
<td>4000 inch/min</td>
</tr>
<tr>
<td>Feedrate range NOTE</td>
<td>0.01 to 9600 inch/min</td>
<td>0.01 to 9600 inch/min</td>
<td>0.01 to 4000 inch/min</td>
</tr>
<tr>
<td>Incremental feed</td>
<td>0.001, 0.01, 0.1, 1 inch/step</td>
<td>0.0001, 0.001, 0.01, 0.1 inch/step</td>
<td>0.00001, 0.0001, 0.001, 0.01 inch/step</td>
</tr>
<tr>
<td>Tool compensation</td>
<td>0 to ±99.999 inch</td>
<td>0 to ±99.9999 inch</td>
<td>0 to ±99.99999 inch</td>
</tr>
<tr>
<td>Dwell time</td>
<td>0 to 999999.99 sec</td>
<td>0 to 999999.999 sec</td>
<td>0 to 9999.9999 sec</td>
</tr>
</tbody>
</table>

### In case of millimeter input, feed screw is inch

<table>
<thead>
<tr>
<th>Increment system</th>
<th>IS–A</th>
<th>IS–B</th>
<th>IS–C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Least input increment</td>
<td>0.01 mm</td>
<td>0.001 mm</td>
<td>0.0001 mm</td>
</tr>
<tr>
<td>Least command increment</td>
<td>0.001 inch</td>
<td>0.0001 inch</td>
<td>0.00001 inch</td>
</tr>
<tr>
<td>Max. programmable dimension</td>
<td>±999999.999 mm</td>
<td>±99999.999 mm</td>
<td>±9999.9999 mm</td>
</tr>
<tr>
<td>Max. rapid traverse NOTE</td>
<td>9600 inch/min</td>
<td>9600 inch/min</td>
<td>4000 inch/min</td>
</tr>
<tr>
<td>Feedrate range NOTE</td>
<td>1 to 240000 mm/min</td>
<td>1 to 240000 mm/ min</td>
<td>1 to 100000 mm/ min</td>
</tr>
<tr>
<td>Incremental feed</td>
<td>0.01, 0.1, 1, 10 mm/step</td>
<td>0.001, 0.01, 0.1, 1 mm/step</td>
<td>0.0001, 0.001, 0.01, 0.1 mm/ step</td>
</tr>
<tr>
<td>Tool compensation</td>
<td>0 to ±99.999 mm</td>
<td>0 to ±99.9999 mm</td>
<td>0 to ±99.99999 mm</td>
</tr>
<tr>
<td>Dwell time</td>
<td>0 to 999999.99 sec</td>
<td>0 to 999999.999 sec</td>
<td>0 to 9999.9999 sec</td>
</tr>
</tbody>
</table>

### Rotation axis

<table>
<thead>
<tr>
<th>Increment system</th>
<th>IS–B</th>
<th>IS–C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Least input increment</td>
<td>0.001 deg</td>
<td>0.0001 deg</td>
</tr>
<tr>
<td>Least command increment</td>
<td>0.001 deg</td>
<td>0.0001 deg</td>
</tr>
<tr>
<td>Max. programmable dimension</td>
<td>±999999.999 deg</td>
<td>±9999.99999 deg</td>
</tr>
<tr>
<td>Max. rapid traverse NOTE</td>
<td>240000 deg/min</td>
<td>100000 deg/min</td>
</tr>
<tr>
<td>Feedrate range NOTE</td>
<td>1 to 240000 deg/min</td>
<td>1 to 100000 deg/min</td>
</tr>
<tr>
<td>Incremental feed</td>
<td>0.001, 0.01, 0.1, 1 deg/ step</td>
<td>0.0001, 0.001, 0.01, 0.1 deg/ step</td>
</tr>
</tbody>
</table>

**NOTE**

1. The feedrate range shown above are limitations depending on CNC interpolation capacity. As a whole system, limitations depending on servo system must also be considered.
2. IS–A is available for the Power Mate i–H only.
3. IS–C is available for the Power Mate i–D only.
**LIST OF FUNCTIONS AND TAPE FORMAT**

Some functions cannot be added as options depending on the model.
In the tables below, \(_{i:}\) presents a combination of arbitrary axis addresses using X,Y,Z,A,B and C (such as X_Y_Z_A_).

- \(x\) = 1st basic axis (X usually)
- \(y\) = 2nd basic axis (Y usually)
- \(z\) = 3rd basic axis (Z usually)

<table>
<thead>
<tr>
<th>Functions</th>
<th>Illustration</th>
<th>Tape format</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning (G00)</td>
<td><img src="image1" alt="Illustration" /></td>
<td>G00 (_{i:});</td>
</tr>
<tr>
<td>Linear interpolation (G01)</td>
<td><img src="image2" alt="Illustration" /></td>
<td>G01 (_{i:F:});</td>
</tr>
<tr>
<td>Circular interpolation (G02, G03)</td>
<td><img src="image3" alt="Illustration" /></td>
<td>G17 (<em>{G02 G03}) X_Y_R_I_J_F</em>;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>G18 (<em>{G02 G03}) X_Z_R_I_K_F</em>;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>G19 (<em>{G02 G03}) Y_Z_R_J_K_F</em>;</td>
</tr>
<tr>
<td>Dwell (G04) (In case of X–Y plane)</td>
<td><img src="image4" alt="Illustration" /></td>
<td>G04 (<em>{X_P</em>});</td>
</tr>
<tr>
<td>Change of offset value by program (G10)</td>
<td></td>
<td>G10 L11 P_R_;</td>
</tr>
<tr>
<td>Tool length offset A (G43, G44, G49)</td>
<td><img src="image5" alt="Illustration" /></td>
<td>(<em>{G43 G44}) Z_H</em>;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(<em>{G43 G44}) H</em>;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>H : Tool offset G49 : Cancel</td>
</tr>
<tr>
<td>Plane section (G17, G18, G19)</td>
<td></td>
<td>G17;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>G18;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>G19;</td>
</tr>
<tr>
<td>Functions</td>
<td>Illustration</td>
<td>Tape format</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| Inch/millimeter conversion (G20, G21) | | G20 ; Inch input  
G21 ; Millimeter input |
| Reference position return check (G27) | ![Reference position](image) | G27 IP _ ; |
| Reference position return (G28) 2nd, reference position return (G30) | ![Reference position](image) | G28 IP _ ;  
G30 IP _ ; |
| Return from reference position to start point (G29) | ![Reference position](image) | G29 IP _ ; |
| Skip function (G31) | ![Reference position](image) | G31 IP _ F _ ; |
| Custom macro (G65, G66, G67) | ![Macro](image) | One-shot call  
G65 P _ L_  
<Argument assignment> ;  
P : Program No.  
L : Number of repetition  
Modal call  
G66 P _ L_  
<Argument assignment> ;  
G67 ; Cancel |
| Canned cycles (G73, G74, G76, G78–G89) (Power Mate i–D/D2) | Refer to II–13. FUNCTIONS TO SIMPLIFY PROGRAMMING | G80 ; Cancel  
G73  
G74  
G76  
G78  
G79  
G81  
G89  
X _Z_P_Q_R_F_ ; |
| Absolute/incremental programming (G90/G91) | | G90 _ ; Absolute command  
G91 _ ; Incremental command  
G90 _G91 _ ; Combined use |
| Change of workpiece coordinate system (G92) | ![Coordinate system](image) | G92 IP _ ; |
| Feed per minute, Feed per revolution (G94, G95) | mm/min inch/min  
mm/rev inch/rev | G94F _ ;  
G95F _ ; |
| Constant surface speed control (G96, G97) | | G96S _ ;  
G97S _ ; |
## TAPE CODE LIST

<table>
<thead>
<tr>
<th>Character</th>
<th>ISO code</th>
<th>EIA code</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>8 7 6 5 4 3 2 1</td>
<td>8 7 6 5 4 3 2 1</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td></td>
<td></td>
<td>Number 0</td>
</tr>
<tr>
<td>1</td>
<td></td>
<td></td>
<td>Number 1</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td></td>
<td>Number 2</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td>Number 3</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td></td>
<td>Number 4</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td></td>
<td>Number 5</td>
</tr>
<tr>
<td>6</td>
<td></td>
<td></td>
<td>Number 6</td>
</tr>
<tr>
<td>7</td>
<td></td>
<td></td>
<td>Number 7</td>
</tr>
<tr>
<td>8</td>
<td></td>
<td></td>
<td>Number 8</td>
</tr>
<tr>
<td>9</td>
<td></td>
<td></td>
<td>Number 9</td>
</tr>
<tr>
<td>A</td>
<td></td>
<td></td>
<td>Address A</td>
</tr>
<tr>
<td>B</td>
<td></td>
<td></td>
<td>Address B</td>
</tr>
<tr>
<td>C</td>
<td></td>
<td></td>
<td>Address C</td>
</tr>
<tr>
<td>D</td>
<td></td>
<td></td>
<td>Address D</td>
</tr>
<tr>
<td>E</td>
<td></td>
<td></td>
<td>Address E</td>
</tr>
<tr>
<td>F</td>
<td></td>
<td></td>
<td>Address F</td>
</tr>
<tr>
<td>G</td>
<td></td>
<td></td>
<td>Address G</td>
</tr>
<tr>
<td>H</td>
<td></td>
<td></td>
<td>Address H</td>
</tr>
<tr>
<td>I</td>
<td></td>
<td></td>
<td>Address I</td>
</tr>
<tr>
<td>J</td>
<td></td>
<td></td>
<td>Address J</td>
</tr>
<tr>
<td>K</td>
<td></td>
<td></td>
<td>Address K</td>
</tr>
<tr>
<td>L</td>
<td></td>
<td></td>
<td>Address L</td>
</tr>
<tr>
<td>M</td>
<td></td>
<td></td>
<td>Address M</td>
</tr>
<tr>
<td>N</td>
<td></td>
<td></td>
<td>Address N</td>
</tr>
<tr>
<td>O</td>
<td></td>
<td></td>
<td>Address O</td>
</tr>
<tr>
<td>P</td>
<td></td>
<td></td>
<td>Address P</td>
</tr>
<tr>
<td>Q</td>
<td></td>
<td></td>
<td>Address Q</td>
</tr>
<tr>
<td>R</td>
<td></td>
<td></td>
<td>Address R</td>
</tr>
<tr>
<td>S</td>
<td></td>
<td></td>
<td>Address S</td>
</tr>
<tr>
<td>T</td>
<td></td>
<td></td>
<td>Address T</td>
</tr>
<tr>
<td>U</td>
<td></td>
<td></td>
<td>Address U</td>
</tr>
<tr>
<td>V</td>
<td></td>
<td></td>
<td>Address V</td>
</tr>
<tr>
<td>W</td>
<td></td>
<td></td>
<td>Address W</td>
</tr>
<tr>
<td>X</td>
<td></td>
<td></td>
<td>Address X</td>
</tr>
<tr>
<td>Y</td>
<td></td>
<td></td>
<td>Address Y</td>
</tr>
<tr>
<td>Z</td>
<td></td>
<td></td>
<td>Address Z</td>
</tr>
<tr>
<td>Character</td>
<td>ISO code</td>
<td>EIA code</td>
<td>Meaning</td>
</tr>
<tr>
<td>-----------</td>
<td>----------</td>
<td>----------</td>
<td>---------</td>
</tr>
<tr>
<td>DEL</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Del 8 7 6 5 4 3 2 1</td>
<td>Delete (deleting a mispunch)</td>
</tr>
<tr>
<td>NUL</td>
<td></td>
<td></td>
<td>No punch. With EIA code, this code cannot be used in a significant information section.</td>
</tr>
<tr>
<td>BS</td>
<td>8 7 6 5 4 3 2 1</td>
<td>BS 8 7 6 5 4 3 2 1</td>
<td>Backspace</td>
</tr>
<tr>
<td>HT</td>
<td></td>
<td></td>
<td>Tabulator</td>
</tr>
<tr>
<td>LF or NL</td>
<td>8 7 6 5 4 3 2 1</td>
<td>CR or EOB 8 7 6 5 4 3 2 1</td>
<td>End of block</td>
</tr>
<tr>
<td>CR</td>
<td></td>
<td></td>
<td>Carriage return</td>
</tr>
<tr>
<td>SP</td>
<td></td>
<td></td>
<td>Space</td>
</tr>
<tr>
<td>%</td>
<td>8 7 6 5 4 3 2 1</td>
<td>ER 8 7 6 5 4 3 2 1</td>
<td>Absolute rewind stop</td>
</tr>
<tr>
<td>(</td>
<td>8 7 6 5 4 3 2 1</td>
<td>(2–4–5) 8 7 6 5 4 3 2 1</td>
<td>Control out (start of comment)</td>
</tr>
<tr>
<td>)</td>
<td>8 7 6 5 4 3 2 1</td>
<td>(2–4–7) 8 7 6 5 4 3 2 1</td>
<td>Control in (end of comment)</td>
</tr>
<tr>
<td>+</td>
<td>8 7 6 5 4 3 2 1</td>
<td>+ 8 7 6 5 4 3 2 1</td>
<td>Plus sign</td>
</tr>
<tr>
<td>-</td>
<td>8 7 6 5 4 3 2 1</td>
<td>– 8 7 6 5 4 3 2 1</td>
<td>Minus sign</td>
</tr>
<tr>
<td>:</td>
<td>8 7 6 5 4 3 2 1</td>
<td>: 8 7 6 5 4 3 2 1</td>
<td>Colon (address O)</td>
</tr>
<tr>
<td>/</td>
<td>8 7 6 5 4 3 2 1</td>
<td>/ 8 7 6 5 4 3 2 1</td>
<td>Optional block skip</td>
</tr>
<tr>
<td>.</td>
<td>8 7 6 5 4 3 2 1</td>
<td>. 8 7 6 5 4 3 2 1</td>
<td>Period (decimal point)</td>
</tr>
<tr>
<td>#</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6012) 8 7 6 5 4 3 2 1</td>
<td>Sharp</td>
</tr>
<tr>
<td>$</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6012) 8 7 6 5 4 3 2 1</td>
<td>Dollar sign</td>
</tr>
<tr>
<td>&amp;</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6012) 8 7 6 5 4 3 2 1</td>
<td>Ampersand</td>
</tr>
<tr>
<td>'</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6010) 8 7 6 5 4 3 2 1</td>
<td>Apostrophe</td>
</tr>
<tr>
<td>*</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6010) 8 7 6 5 4 3 2 1</td>
<td>Asterisk</td>
</tr>
<tr>
<td>,</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6011) 8 7 6 5 4 3 2 1</td>
<td>Comma</td>
</tr>
<tr>
<td>;</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6011) 8 7 6 5 4 3 2 1</td>
<td>Semicolon</td>
</tr>
<tr>
<td>&lt;</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6011) 8 7 6 5 4 3 2 1</td>
<td>Left angle bracket</td>
</tr>
<tr>
<td>=</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6011) 8 7 6 5 4 3 2 1</td>
<td>Equal sign</td>
</tr>
<tr>
<td>&gt;</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6011) 8 7 6 5 4 3 2 1</td>
<td>Right angle bracket</td>
</tr>
<tr>
<td>?</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6011) 8 7 6 5 4 3 2 1</td>
<td>Question mark</td>
</tr>
<tr>
<td>@</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6011) 8 7 6 5 4 3 2 1</td>
<td>Commercial at mark</td>
</tr>
<tr>
<td>&quot;</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6011) 8 7 6 5 4 3 2 1</td>
<td>Quotation mark</td>
</tr>
<tr>
<td>[</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6013) 8 7 6 5 4 3 2 1</td>
<td>Left square bracket</td>
</tr>
<tr>
<td>]</td>
<td>8 7 6 5 4 3 2 1</td>
<td>Parameter (No.6014) 8 7 6 5 4 3 2 1</td>
<td>Right square bracket</td>
</tr>
</tbody>
</table>
### NOTE

1. The symbols used in the remark column have the following meanings.
   - **(Space)**: The character will be registered in memory and has a specific meaning. It is used incorrectly in a statement other than a comment, an alarm occurs.
   - **×**: The character will not be registered in memory and will be ignored.
   - **Δ**: The character will be registered in memory, but will be ignored during program execution.
   - **○**: The character will be registered in memory. If it is used in a statement other than a comment, an alarm occurs.
   - **□**: If it is used in a statement other than a comment, the character will not be registered in memory. If it is used in a comment, it will be registered in memory.

2. Codes not in this table are ignored if their parity is correct.

3. Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.

4. A character with all eight holes punched is ignored and does not generate TH alarm in EIA code.
### EXTERNAL DIMENSIONS OF EACH UNIT

<table>
<thead>
<tr>
<th>Classification</th>
<th>Name</th>
<th>Number of Figure</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Control unit</strong></td>
<td>Control unit</td>
<td>Fig. 1</td>
</tr>
<tr>
<td><strong>I/O</strong></td>
<td>I/O Card D, E</td>
<td>Fig. 2</td>
</tr>
<tr>
<td></td>
<td>FSSB I/O Module</td>
<td>Fig. 3</td>
</tr>
<tr>
<td><strong>Unit for display and data input</strong></td>
<td>CRT/MDI, CRT/MDI with picture display</td>
<td>Fig. 4</td>
</tr>
<tr>
<td></td>
<td>LCD/MDI</td>
<td>Fig. 5</td>
</tr>
<tr>
<td></td>
<td>Separate CRT</td>
<td>Fig. 6</td>
</tr>
<tr>
<td></td>
<td>Separate MDI, separate MDI with picture display</td>
<td>Fig. 7</td>
</tr>
<tr>
<td></td>
<td>Separate PDP</td>
<td>Fig. 8</td>
</tr>
<tr>
<td></td>
<td>Separate LCD</td>
<td>Fig. 9</td>
</tr>
<tr>
<td></td>
<td>Detachable LCD/MDI</td>
<td>Fig. 10</td>
</tr>
<tr>
<td></td>
<td>Handy operator’s panel</td>
<td>Fig. 11</td>
</tr>
<tr>
<td><strong>External device</strong></td>
<td>α Position coder</td>
<td>Fig. 12</td>
</tr>
<tr>
<td></td>
<td>Manual pulse generator</td>
<td>Fig. 13</td>
</tr>
<tr>
<td></td>
<td>Manual pulse generator (Pendant type)</td>
<td>Fig. 14</td>
</tr>
<tr>
<td></td>
<td>Separate detector interface unit</td>
<td>Fig. 15</td>
</tr>
<tr>
<td></td>
<td>Battery case for separate absolute pulse coder</td>
<td>Fig. 16</td>
</tr>
<tr>
<td></td>
<td>Battery case for external control unit</td>
<td>Fig. 17</td>
</tr>
<tr>
<td></td>
<td>Punch panel (for 1 channel)</td>
<td>Fig. 18</td>
</tr>
<tr>
<td></td>
<td>Punch panel (for 2 channels)</td>
<td>Fig. 19</td>
</tr>
<tr>
<td></td>
<td>FANUC Handy File</td>
<td>Fig. 20</td>
</tr>
<tr>
<td></td>
<td>I/O Link connection unit</td>
<td>Fig. 21</td>
</tr>
<tr>
<td></td>
<td>HSSB interface board</td>
<td>Fig. 22</td>
</tr>
</tbody>
</table>
Fig. 1 Control unit

Specification: A02B-0259-B501

Weight: 1.8kg
Fig. 2  I/O Card D, E

A16B–2202–0732 (DI/DO=96/64)
Fig. 3  FSSB I/O Module

Specification : A02B–0236–C211, C212
**Fig. 4** CRT/MDI, CRT/MDI with picture display

Specification:
- A02B–0166–C201#R (English key)
- A02B–0166–C201#S (Symbolic key)
- A02B–0166–C221#R (English key, with picture display for 32 screen)
- A02B–0166–C221#S (Symbolic key, with picture display for 32 screen)
- A02B–0166–C222#R (English key, with picture display for 64 screen)
- A02B–0166–C222#S (Symbolic key, with picture display for 64 screen)

Weight: 5.8kg
Fig. 5 LCD/MDI

Specification: A02B–0166–C261#R (English key)
A02B–0166–C261#S (Symbolic key)

Weight: 3.9kg
Fig. 6 Separate CRT

Specification : A02B–0120–C111

Panel cut drawing

Stad (M4) for grounding

Weight : 3.5kg
Fig. 7  Separate MDI, separate MDI with picture display

Specification:  
A02B–0166–C210#R (English key)  
A02B–0166–C210#S (Symbolic key)  
A02B–0166–C231#R (English key, with picture display for 32 screen)  
A02B–0166–C231#S (Symbolic key, with picture display for 32 screen)  
A02B–0166–C232#R (English key, with picture display for 64 screen)  
A02B–0166–C232#S (Symbolic key, with picture display for 64 screen)  

Weight: 1.3kg
Fig. 8 Separate PDP

Specification: A02B–0200–C100

Weight: 2.1kg
Fig. 9  Separate LCD

Specification: A02B-0166-C251

Weight: 1.6 kg
Fig. 10 Detachable LCD/MDI

Specification:  A02B–0166–C271#R (English key)
              A02B–0166–C271#S (Symbolic key)

Weight: 7.0 kg
Fig. 11 Handy operator’s panel

Specification: A02B–0211–C020#R (English key)
A02B–0211–C020#S (Symbolic key)

Weight: 1.3 kg

Tolerance: ±1 mm
Fig. 12 α Position coder

Specification: A860–0309–T302 (Max. 10000 rpm)
**Fig. 13** Manual pulse generator

Specification: A860–0202–T001

- Weight: 0.35kg
Fig. 14  Manual pulse generator (Pendant type)

Specification : A860–0202–T004 to T015, T020

A860–0202–T004 to T009

A860–0202–T010 to T015, T020
Fig. 15  Separate detector interface unit

Specification : A02B–0236–C203, –C204
Fig. 16 Battery case for separate absolute pulse coder

Specification: A06B–6050–K060
Fig. 17 Battery case for external control unit

Specification : A02B–0236–C281

The battery unit is fitted with a 14–m battery cable.
**Fig. 18** Punch panel (for 1 channel)

Specification: A02B–0259–C191 (Cable length: 1 m)
A02B–0259–C192 (Cable length: 2 m)
A02B–0259–C193 (Cable length: 5 m)

Ground pin for the frame ground (M4, length: 200 mm)
A ground pin for the frame ground must be provided near the punch panel.
Fig. 19  Punch panel (for 2 channels)

Specification:  
- A02B–0259–C051 (Cable length: 1m)  
- A02B–0259–C052 (Cable length: 2m)  
- A02B–0259–C053 (Cable length: 3m)

Ground pin for the frame ground (M4, length: 200 mm)  
A ground pin for the frame ground must be provided near the punch panel.
Fig. 20  FANUC Handy File

Specification: A13B–0159–B001 (English key)
A13B–0159–B002 (Japanese key)

Weight: 1.4kg
Unit: mm
**Fig. 21** I/O Link connection unit

Specification:
- A20B–2000–0410 (electric – optical)
- A20B–2000–0411 (electric – electric)
- A20B–2000–0412 (optical – optical)

Unit: mm

- Printed-circuit board
- Installation dimensions: 160 x 90
- Outside dimensions: 180 x 160 x 30
- Components depth: Up to 50
- Cable: 4-M4
- Weight: 600g
Fig. 22  HSSB interface board

Specification: A20B–8001–0583 (HSSB 1ch)
A20B–8001–0582 (HSSB 2ch)

Weight: 0.2kg
<< Numbers >>
1–block plural M command, 58
2nd and 3rd reference position return (G30), 41

<< A >>
Abnormal load detection, 166
Absolute and incremental programming (G90, G91), 49
Accuracy compensation function, 82
Activation of automatic operation, 111
Additional optional block skip, 65
Alarm signal, 168
All axes interlock, 166
All–axes machine lock, 115
Analog input function, 160
Automatic acceleration/deceleration, 32
Automatic coordinate system setting, 46
Automatic operation, 108
Automatic operation signal, 168
Automatic operation start signal, 168
Automatic operation stop, 112
Automatic reference position return (G28, G29), 40
Auxiliary function lock, 115
Axes control, 98
Axis control with PMC, 103
Axis move direction signal, 169
Axis names, 18

<< B >>
Background editing, 151
Backlash compensation, 83
Basic addresses and command value range, 64
Batch save and restore operations based on memory cards, 158
Bell–shaped acceleration/deceleration after cutting feed interpolation, 35
Buffer register, 111
Butt–type reference position setting, 42

<< C >>
C language executer function, 97
Canned cycle cancel (G80), 73
Canned cycles (G73, G74, G76, G80 – G89, G98, G99) (only for Power Mate–D), 67
Changing of tool offset amount (programmable data input) (G10), 81
Circular interpolation (G02, G03), 25
Clock function, 138

Constant surface speed control, 54
Constant surface speed control signal, 169
Control axis detach, 100
Control–in/control–out, 65
Controlled axes, 16, 17
Coordinate systems, 44
Coordinate value and dimension, 48
CRT/MDI, 118
Custom macro, 87, 88
Cutting feed rate, 29
Cutting feed rate clamp, 29
Cycle start, 111

<< D >>
Data input/output, 155
Data input/output function based on the I/O Link and data input/output function B based on the I/O Link, 159
Data input/output using a memory card, 157
Data protection key, 148
Decimal point input/pocket calculator type decimal point input, 50
Detachable LCD/MDI, 126
DI status output signal, 169
Diagnosis functions, 153
Directory display of floppy cassette, 141
Display, 135
Displaying and setting data, 134
Displaying operation history, 148
Distribution end signal, 168
DNC operation (only for Power Mate i–D), 109
Dry run, 115
Dwell (G04), 37

<< E >>
Emergency stop, 162
Exact stop (G09), 37
Execution of automatic operation, 111
Expanded part program editing, 152
Explanation of the function keys, 132
Explanation of the keyboard, 130
Explanation of the soft keys, 133
External alarm message, 171
External control of I/O device, 152
External data input, 170
External dimensions of each unit, 184
External operator’s message, 171
External program number search, 171
External workpiece number search, 110
Externally setting the stroke limit, 165

**F**
- FANUC Handy File, 156
- Feed functions, 27
- Feed hold, 112
- Feed hold signal, 168
- Feed stop, 102
- Follow up function, 99
- For English key, 130
- For symbol key, 132
- Foreground editing, 151
- Functions to simplify programming, 66

**H**
- Handle feed in the same mode as for jogging, 107
- Handle interruption, 112
- Handy operator’s panel, 124
- Help function, 146
- High-speed M/S/T interface, 59
- High-speed serial bus (HSSB), 174

**I**
- In-position signal, 168
- Inch input signal, 169
- Inch/metric conversion (G20, G21), 50
- Increment system, 19
- Incremental feed, 106
- Input unit (10 times), 19
- Input/output devices, 156
- Interlock, 166
- Interlock for each axis direction, 166
- Interlock per axis, 166
- Interpolation functions, 22
- Interruption type custom macro, 94

**J**
- Jog override, 31

**K**
- Key input from PMC (external key input), 172

**L**
- Label skip, 65
- Language selection, 138
- LCD/MDI, 119
- Left-handed rigid tapping cycle (G74), 77
- Linear acceleration/deceleration after cutting feed interpolation, 34
- Linear acceleration/deceleration before cutting feed interpolation, 36
- Linear axis and rotation axis, 51
- Linear interpolation (G01), 24
- Linear scale with absolute addressing reference marks, 43
- List of functions and tape format, 179
- List of specifications, 5

**M**
- Machine coordinate system, 45
- Macro executer function, 96
- Main program, 62
- Maintenance information screen, 149
- Manual absolute on/off, 107
- Manual feed, 106
- Manual handle feed (1st), 106
- Manual handle feed (2nd, 3rd), 107
- Manual interruption during automatic operation, 112
- Manual operation, 105
- Manual reference position return, 39
- Maximum stroke, 19
- MDI operation, 109
- Measurement functions, 84
- Mechanical handle feed, 99
- Memory operation, 109
- Mirror image, 99
- Miscellaneous functions, 57, 58
- Move signal, 168
- Multi-step skip function (G31 P1 – G31 P4), 86

**N**
- NC ready signal, 168
- Number of registered programs, 152
- Number of tool offsets, 81

**O**
- Operation mode, 109
- Optional block skip, 65
- Override, 31
- Override cancel, 31
Overtravel, 165
Overtravel functions, 165

- **P** -
  Part program storage and editing, 150
  Part program storage length, 152
  Pattern data input, 95
  Peck rigid tapping cycle (G84 or G74), 78
  Per minute feed (G94), 29
  Per revolution feed (G95), 30
  Periodic maintenance screen, 149
  Personal computer (connected via HSSB), 128
  Personal computer (connected via RS–232C), 129
  Personal computer function, 173
  Picture display CRT/MDI, 120
  Plane selection (G17, G18, G19), 47
  Play back, 152
  Polar coordinate interpolation (G12.1, G13.1), 26
  Position switch function, 169
  Positioning (G00), 23
  Power Mate i, 162
  Power mate CNC manager, 160
  Preparatory functions, 20
  Program configuration, 61
  Program end (M02, M30), 112
  Program name, 62
  Program number, 62
  Program number search, 110
  Program stop (M00, M01), 112
  Program test functions, 114
  Programmable parameter entry (G10, G11), 83

- **R** -
  Range of command value, 177
  Rapid traverse, 28
  Rapid traverse bell–shaped acceleration/deceleration, 33
  Rapid traverse override, 31
  Rapid traversing signal, 169
  Reference position, 38
  Reference position external setting, 43
  Reference position return check (G27), 41
  Reset, 112
  Reset signal, 168
  Retraction for tapping, 113
  Return signal, 42
  Rewind, 110
  Rewinding signal, 168
  Rigid tap (only for Power Mate–D), 74
  Rigid tapping (G84), 76
  Rigid tapping return, 113
  Rotation axis roll–over function, 51
  RS–232–C interfaces, 156
  Run time & parts number display, 138

- **S** -
  S code output, 53
  Safety functions, 161
  Screens for servo data and spindle data, 142
  Selection of execution programs, 110
  Self diagnosis functions, 154
  Separate CRT, 122
  Separate LCD, 123
  Separate MDI, 121
  Separate MDI for picture display, 122
  Sequence number, 63
  Sequence number search, 110
  Servo adjustment screen, 142
  Servo off, 99
  Servo ready signal, 168
  Servo setting screen, 142
  Servo/spindle, 163
  Setting a workpiece coordinate system (Using G92), 46
  Setting and display unit, 116, 117
  Setting the reference position without dogs, 39
  Simple synchronous control, 100
  Single block, 115
  Skip function (G31), 85
  Software operator’s panel, 139
  Spindle adjustment screen, 143
  Spindle functions (only for Power Mate–D), 52
  Spindle orientation, 54
  Spindle output control by the PMC, 53
  Spindle output switching, 54
  Spindle override, 54
  Spindle setting screen, 143
  Spindle speed analog output (S analog output), 53
  Spindle speed serial output (S serial output), 53
  Status output, 167
  Stored pitch error compensation (only for Power Mate–D), 83
  Stored stroke check 1, 165
  Sub program, 63
  System configuration display function, 144

- **T** -
  T code output, 56
<table>
<thead>
<tr>
<th>Function/Feature</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tandem control</td>
<td>104</td>
</tr>
<tr>
<td>Tangential speed constant control</td>
<td>29</td>
</tr>
<tr>
<td>Tape code list</td>
<td>181</td>
</tr>
<tr>
<td>Tape codes</td>
<td>63</td>
</tr>
<tr>
<td>Tape format</td>
<td>65</td>
</tr>
<tr>
<td>Tape horizontal (TH) parity check and tape vertical (TV) parity check</td>
<td>65</td>
</tr>
<tr>
<td>Tapping signal (only for Power Mate i-D)</td>
<td>169</td>
</tr>
<tr>
<td>Tool compensation function</td>
<td>79</td>
</tr>
<tr>
<td>Tool compensation memory</td>
<td>81</td>
</tr>
<tr>
<td>Tool functions</td>
<td>55</td>
</tr>
<tr>
<td>Tool length compensation (G43, G44, G49)</td>
<td>80</td>
</tr>
<tr>
<td>Tool length measurement</td>
<td>86</td>
</tr>
<tr>
<td>Torque limit function</td>
<td>101</td>
</tr>
<tr>
<td>Touch panel display unit</td>
<td>127</td>
</tr>
<tr>
<td>Workpiece coordinate system</td>
<td>46</td>
</tr>
</tbody>
</table>
# Revision Record

FANUC Power Mate i–MODEL D/H DESCRIPTIONS (B–63172EN)

<table>
<thead>
<tr>
<th>Edition</th>
<th>Date</th>
<th>Contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>01</td>
<td>Sep., '98</td>
<td></td>
</tr>
</tbody>
</table>
· No part of this manual may be reproduced in any form.

· All specifications and designs are subject to change without notice.