MITSUBISHI ELECTRIC

Changes for the Better

MITSUBISHI CNC

Instruction Manual

C70

IB-1500267(ENG)-G

Introduction

This manual is referred to when using the C70.
This manual explains how to operate the screens of the C70. Read this manual thoroughly before using the CNC unit. To safely use this CNC unit, thoroughly study the "Precautions for Safety" on the next page before use.

Details described in this manual

⚠ CAUTION

⚠ For items described in "Restrictions" or "Usable State", the instruction manual issued by the machine manufacturer takes precedence over this manual.
⚠ An effort has been made to note as many special handling methods in this user's manual. Items not described in this manual must be interpreted as "Not Possible".
⚠ This manual has been written on the assumption that all option functions are added. Refer to the specifications issued by the machine manufacturer before starting use.
⚠ Refer to the manuals issued by the machine manufacturer for each machine tool explanation.
⚠ Some screens and functions may differ or may not be usable depending on the NC version.

< Important Usage Notes >

In this NC unit, the machining programs, parameters and tool compensation data are saved in the memory (memory elements). This NC unit's memory is backed up by lithium batteries, and under normal conditions will last 6 years from the date of manufacture. However, data contents could be lost under the conditions described below.
To prevent data loss, output important programs, parameters, etc., to the external memory devices and save them.

Data in the memory can be lost under these kinds of conditions.

(1) Incorrect operation

Data can be lost if the operator inadvertently changes data while editing a program or setting parameters.
(This is not really a data loss, but it is a loss from the standpoint that the original data is gone.)

Data can be lost if the operator inadvertently deletes data or initializes NC unit.

(2) Battery life expires

When the battery life expires and there is not enough voltage to store the data in the memory, data can be lost by turning the power OFF.

(3) Faults

Data can be lost when faults occur and the control unit must be replaced.

Precautions for Safety

Always read the specifications issued by the machine manufacturer, this manual, related manuals and attached documents before installation, operation, programming, maintenance or inspection to ensure correct use.
Understand this numerical controller, safety items and cautions before using the unit.
This manual ranks the safety precautions into "**DANGER**", "**WARNING**" and "**CAUTION**".

⚠ **DANGER**	When the user may be subject to imminent fatalities or major injuries if handling is mistaken.
⚠ **WARNING**	When the user may be subject to fatalities or major injuries if handling is mistaken.
⚠ **CAUTION**	When the user may be subject to injuries or when physical damage may occur if handling is mistaken.

Note that even items ranked as "⚠ **CAUTION**", may lead to major results depending on the situation. In any case, important information that must always be observed is described.

The meanings of the pictorial signs are given below.

The following sings indicate prohibition and compulsory.

⃠	This sign indicates prohibited behavior (must not do). For example,  indicates “Keep fire away”.

●	This sign indicated a thing that is pompously (must do). For example,  indicates “it must be grounded”.

The meaning of each pictorial sing is as follows.

 CAUTION	 CAUTION rotated object	 CAUTION HOT	 Danger Electric shock risk	 Danger explosive
 Prohibited	 Disassembly is prohibited	 KEEP FIRE AWAY	 General instruction	 Earth ground

⚠ DANGER

1. Items related to maintenance

- Do not touch the terminals while power is on. Doing so could cause electric shock.
- Correctly connect the battery. Also, do not charge, disassemble, heat, place in fire, short circuit, or solder the battery.
  Mishandling of a battery may cause overheating, cracks or ignition which could result in injury and fire.
- Switch off all phases of the externally supplied power used in the system when cleaning the module or retightening the terminal or module mounting screws.
  Not doing so could result in electric shock. Loose tightening of terminal screws can cause a short circuit or malfunction. Over tightening of screws can cause damages to the screws and/or the module, resulting in fallout, short circuits, or malfunction.
- The capacitor is mounted to the modules. Do not incinerate the modules so that the incineration of capacitor may cause burst. For disposal of the modules, request for specialized industrial waste disposal services who has incineration facility.

2.Items related to inspection

- Never open the front case or terminal covers while the power is ON or the unit is running, as this may lead to electric shocks.
- Never run the unit with the front case or terminal cover removed. The high voltage terminal and charged sections will be exposed and may lead to electric shocks.
- Never remove the front case or terminal cover at times other than wiring work or periodic inspections even if the power is OFF. The insides of the controller and servo drive unit are charged and may lead to electric shocks.
- When performing wiring work or inspections, turn the power OFF, wait at least ten minutes, and then check the voltage with a tester, etc. Failing to do so may lead to electric shocks.
- Never operate the switches with wet hands, as this may lead to electric shocks.
- Do not damage, apply excessive stress, place heavy things on or sandwich the cables, as this may lead to electric shocks or fire.
- Do not touch the controller, servo drive unit or servomotor terminal blocks while the power is ON, as this may lead to electric shocks or fire.
- Do not touch the built-in power supply, built-in grounding or signal wires of the controller and servo drive unit, as this may lead to electric shocks.

⚠ WARNING

1. Items related to program development

⚠ Do not put any startup switch for C70 on GOT's touch key. If a communication fault (including cable disconnection) occurs between GOT and CNC C70, the communication will be suspended and the GOT will become inoperative. In this case, even when you release your hands from the startup switch, CNC will fail to recognize the cutoff of startup signal, which may cause serious accidents.

2. Items related to operation

⚠ If the operation start position is set in a block which is in the middle of the program and the program is started, the program before the set block is not executed. Please confirm that G and F modal and coordinate values are appropriate. If there are coordinate system shift commands or M, S, T and B commands before the block set as the start position, carry out the required commands using the MDI, etc. If the program is run from the set block without carrying out these operations, there is a danger of interference with the machine or of machine operation at an unexpected speed, which may result in breakage of tools or machine tool or may cause damage to the operators.

⚠ Under the constant surface speed control (during G96 modal), if the axis targeted for the constant surface speed control moves toward the spindle center, the spindle rotation speed will increase and may exceed the allowable speed of the workpiece or chuck, etc. In this case, the workpiece, etc. may jump out during machining, which may result in breakage of tools or machine tool or may cause damage to the operators.

⚠ CAUTION

1. Items related to product and manual

- ⚠ For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine manufacturer takes precedence over this manual.
- ⚠ An effort has been made to describe special handling of this machine, but items that are not described must be interpreted as "Not Possible".
- ⚠ This manual is written on the assumption that all option functions are added. Refer to the specifications issued by the machine manufacturer before starting use.
- ⚠ Refer to the manuals issued by the machine manufacturer for each machine tool explanation.
- ⚠ Some screens and functions may differ or may not be usable depending on the NC version.

2. Items related to installation and assembly

- ⏚ Always ground the signal cable to ensure stable operation of the system. Ground the NC unit, power distribution panel and machine to a one-point ground to establish the same potential.

3. Items related to preparations before use.

- ❗ Always set the stored stroke limit. Setting no limits could result in collision with the machine end.
- ❗ Always turn the power OFF before connecting/disconnecting the input/output device cables. The NC and input/output device could be damaged if the cable is connected in the power ON state.

4. Items related to screen operation

- ⚠ If the tool compensation amount is changed during automatic operation (including single block stop), the amount will be validated from the next block or several blocks following.
- ⚠ If the workpiece coordinate offset amount is changed during single block stop, the changes will be valid from the next block.
- ⚠ Pay close attention to the sequence operation when carrying out forced data setting (forced output) in the PLC I/F (diagnosis) screen.
- ⚠ If the operation start position is set from a block in the program and the program is started, the program before the set block is not executed. If there are coordinate system shift commands or M, S, T, and B commands before the block set as the starting position, carry out the required commands using the MDI, etc. There is a danger of interference with the machine if the operation is started from the set starting position block without carrying out these operations.
- ❗ To prevent the influence of data loss and data transformation over the line, always carry out data comparison after transferring a machining program.
- ⃠ Do not change the setup parameters without prior consent from the machine manufacturer.

⚠ CAUTION

5. Items related to programming

- ⚠ If there is no value after the G command, the operation will be the "G00" operation when the program is run due to key chattering, etc., during editing.
- ⚠ " ; " "EOB" and " % " "EOR" are symbols used for explanation.
  The actual codes are: For ISO: "CR, LF", or "LF" and "%".
  Programs created on the Edit screen are stored in the NC memory in a "CR, LF" format, but programs created with external devices such as the FLD or RS-232C may be stored in an "LF" format.
  The actual codes for EIA are: "EOB (End of Block)" and "EOR (End of Record)".
- ⚠ When creating the machining program, select the appropriate machining conditions so as not to exceed the machine and NC performance, capacity and limits. The examples do not consider the machining conditions.
- ⃠ Do not change the fixed cycle program without prior consent from the machine manufacturer.

6. Items related to operation

- ⚠ Do not enter the movable range of the machine during automatic operation. Make sure not to place hands, legs or face near the spindle during rotation.
- ⚠ Always carry out dry run operation before actual machining, and confirm the machining program, tool offset amount and workpiece coordinate system offset amount.

7. Items related to faults and errors

- ❗ If the BATTERY LOW warning is output, save the machining programs, tool data and parameters to an input/output device, and then replace the battery. If the BATTERY alarm occurs, the machining programs, tool data and parameters may be damaged. After replacing the battery, reload each data item.
- ❗ If the axis overruns or makes an abnormal noise, press the EMERGENCY STOP button immediately, and stop the axis.
- ❗ When setting the parameter not to check the overheat, the control unit and the communication terminal may not be controlled because of overheat.
  In such case, axis runaway may cause a machine breakage, an accident resulting in injury or death, or device breakage.
  To prevent the serious results, ordinarily set the parameters so that the overheat check is valid.

8. Items related to maintenance

- ⚠ Do not apply voltages on the connector other than those indicated in Connection and Maintenance manual. Doing so may lead to destruction or damage.
- ⚠ Incorrect connections may damage the devices, so connect the cables to the specified connectors.
- ⃠ Do not connect or disconnect the connection cables between each unit while the power is ON.
- ⃠ Do not connect or disconnect any PCB while the power is ON.
- ⚠ Do not replace the battery while the power is ON.
- ⚠ Do not short-circuit, charge, overheat, incinerate or disassemble the battery.
- ⚠ Dispose of the spent battery according to local laws.

⚠ CAUTION

⚠ Read the manual carefully and pay careful attention to safety for the on-line operation (especially program change, forced stop or operation change) performed by connecting peripheral devices to the controller during operation. Erroneous operation may cause machine breakage or accident.

⊘ Never try to disassemble or modify module. It may cause product failure, operation failure, injury or fire.

⚠ Use any radio communication device such as a cellular phone or a PHS phone more than 25cm (9.84 inch) away in all directions of C70. Failure to do so may cause a malfunction.

⚠ Completely turn off the externally supplied power used in the system before installation or removing the module. Not doing so could result in electric shock, damage to the module or operation failure.

⚠ Do not install/remove the module on to/from base unit or terminal block more than 50 times, after the first use of the product. Failure to do so may cause the module to malfunction due to poor contact of connector.

⊘ Do not drop or impact the battery installed to the module. Doing so may damage the battery, causing battery liquid to leak in the battery. Do not use the dropped or impacted battery, but dispose of it.

⚠ Before touching the module, always touch grounded metal, etc. to discharge static electricity from human body. Failure to do so may cause the module to fail or malfunction.

⊘ Do not directly touch the module's conductive parts and electronic components. Touching them could cause an operation failure or give damage to the module.

9. Items related to inspection

⚠ Be sure to ground the controller, servo amplifier and servomotor. Do not ground commonly with other devices. (Ground resistance: 100 Ω or less)

⚠ The wiring work and inspections must be done by a qualified technician.

⚠ Wire the units after mounting the controller, servo amplifier and servomotor. Failing to do so may lead to electric shocks or damage.

⚠ Perform the daily and periodic inspections according to the instruction manual.

⚠ Perform maintenance and inspection after backing up the program and parameters for the controller and servo amplifier.

⊘ Do not place fingers or hands in the clearance when opening or closing any opening.

⚠ Periodically replace consumable parts such as batteries according to the instruction manual.

⊘ Do not touch the lead sections such as ICs or the connector contacts.

⊘ Do not place the controller or servo amplifier on metal that may cause a power leakage or wood, plastic or vinyl that may cause static electricity buildup.

⊘ Do not perform a megger test (insulation resistance measurement) during inspection.

⚠ When replacing the Motion controller or servo amplifier, always set the new unit settings correctly.

⚠ After maintenance and inspections are completed, confirm that the position detection of the absolute position detector function is correct.

⊘ Do not short circuit, charge, overheat, incinerate or disassemble the batteries.
The electrolytic capacitor will generate gas during a fault, so do not place your face near the controller or servo amplifier.

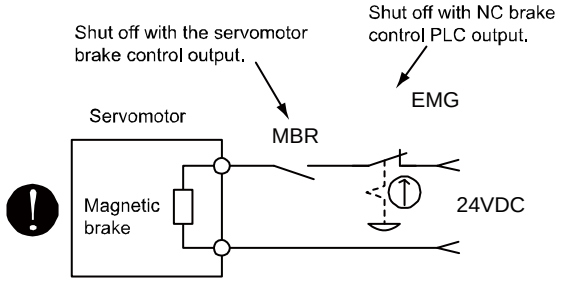
⚠ CAUTION

⚠ The electrolytic capacitor and fan will deteriorate. Periodically change these to prevent secondary damage from faults. Replacements can be made by our sales representative, or at the nearest service center.

⚠ If an error occurs in the self diagnosis of the controller or servo amplifier, confirm the check details according to the instruction manual, and restore the operation.

⚠ If a dangerous state is predicted in case of a power failure or product failure, in order to prevent that state, use a servomotor with electromagnetic brakes for maintenance or mount a brake mechanism externally.

⚠ Use a double circuit construction so that the electromagnetic brake operation circuit can be operated by emergency stop signals set externally.



⚠ If an error occurs, remove the cause, secure the safety and then resume operation after alarm release.

⃠ The unit may suddenly restart after a power failure is restored, so do not go near the machine.
(Design the machine so that personal safety can be ensured even if the machine restarts suddenly.)

⚠ Confirm and adjust the program and each parameter before operation. Unpredictable movements may occur depending on the machine.

⃠ Extreme adjustments and changes may lead to unstable operation, so never make them.

⃠ Do not apply a voltage other than that specified in the instruction manual on any terminal. Doing so may lead to destruction or damage.

⃠ Do not mistake the terminal connections, as this may lead to destruction or damage.

⃠ Do not mistake the polarity (+ / -), as this may lead to destruction or damage.

10. Items related to servo/spindle parameters

⚠ Do not make remarkable adjustments or changes of the parameters as the operation may became unstable.

Disposal

(Note) This symbol mark is for EU countries only.
This symbol mark is according to the directive 2006/66/EC Article 20 Information for end-users and Annex II.

Your MITSUBISHI ELECTRIC product is designed and manufactured with high quality materials and components which can be recycled and/or reused.
This symbol means that batteries and accumulators, at their end-of-life, should be disposed of separately from your household waste.
If a chemical symbol is printed beneath the symbol shown above, this chemical symbol means that the battery or accumulator contains a heavy metal at a certain concentration. This will be indicated as follows:
Hg: mercury (0,0005%), Cd: cadmium (0,002%), Pb: lead (0,004%)
In the European Union there are separate collection systems for used batteries and accumulators.
Please, dispose of batteries and accumulators correctly at your local community waste collection/ recycling centre.

Please, help us to conserve the environment we live in!

CONTENTS

I. SCREEN OPERATIONS

1. Setting and Display Unit 1
1.1 Graphic Operation Terminal (GOT) 1
1.2 Screen Operation of GOT 2

2. CNC Monitor Screen 4
2.1 Setting and Display Unit Operation 6
2.1.1 Display Area of NC Screens 6
2.1.2 Screen Transition Diagram (L system) 8
2.1.3 Screen Transition Diagram (M system) 9
2.1.4 Screen Selection Procedure 10
2.1.5 Data Setting Method 14
2.2 Monitor 18
2.2.1 Position 19
2.2.1.1 Position Display Counter Zero and Origin Zero 20
2.2.1.2 Manual Numerical Value Command (S, T, M) 21
2.2.1.3 Displaying Automatic Operation Program 24
2.2.2 Coordinate 25
2.2.3 Command 27
2.2.3.1 Execution Program Monitor 27
2.2.3.2 Execution Modal Monitor 28
2.2.3.3 Total Integrating Time Display 30
2.2.4 Program Search 32
2.2.4.1 Memory Search 33
2.2.5 Common Variable 34
2.2.5.1 Common Variable Display 35
2.2.5.2 Common Variable Setting 36
2.2.5.3 Common Variable Data Deleting 36
2.2.6 Local Variable 37
2.2.6.1 Local Variable Data Display 38
2.3(I) Tool Offset (L system) 39
2.3.1 Wear Data 40
2.3.1.1 Setting Tool Offset Data 41
2.3.1.2 Erasing the Tool Offset Data 41
2.3.1.3 Tool Wear and Tool Length Data Setting Mode (incremental/absolute) 42
2.3.2 Tool Length Data 43
2.3.2.1 Manual Tool Length Measurement I 44
2.3.2.2 Manual Numerical Command Operation on the TOOL DATA Screen (M, T) 50
2.3.2.3 Tool Presetter 51
2.3.3 Tool Nose Data 56
2.3.4 Tool Life Management I ("#1096 T_L type" is 1) 57
2.3.4.1 Tool Life Management Method 58
2.3.4.2 Conditions for Counting (incrementing) 58
2.3.4.3 Setting Tool Life Management Data 59
2.3.4.4 Erasing Tool Life Management Data in Display Screen Units 59
2.3.4.5 Cautions 59
2.3.5 Tool Life Management II ("#1096 T_Ltype" is 2) 60
2.3.5.1 Group Registration 61
2.3.5.2 Tool Life Incrementation Methods 64
2.3.5.3 Parameters 65
2.3.6 Tool Registration 66
2.3.6.1 Outline of Functions 66
2.3.6.2 Tool Registration in the Magazine Pot 67

2.3.6.3 Tool Registration in the Spindle, Standby and Indexing Areas 68
2.3.6.4 Deleting Tool Registration Data 68
2.3.6.5 Manual Numerical Command Operation (M, T) on the TOOL REGISTRATION Screen 69
2.3.7 Workpiece Coordinate 69
2.3(II) Tool Offset (M system) 70
2.3.1 Tool Offset 71
2.3.1.1 Tool Offset Data Setting 72
2.3.1.2 Tool Offset Data Clear 72
2.3.1.3 Tool Offset Data Setting Modes (Absolute and Incremental) 73
2.3.1.4 Manual Tool Length Measurement 75
2.3.1.5 Manual Numeric Command Operation on the TOOL OFFSET Screen (M, T) 78
2.3.2 Tool Registration 79
2.3.2.1 Function Outline 79
2.3.2.2 Tool Registration in Magazine Pot 80
2.3.2.3 Tool Registration in HEAD, NEXT, and INDEX 81
2.3.2.4 Tool Registration Data Clear 81
2.3.2.5 Manual Numeric Command Operation on the TOOL REGISTRATION Screen (M, T) 82
2.3.3 Tool Life 83
2.3.3.1 Function Outline 83
2.3.3.2 TOOL LIFE Screen Data Display 84
2.3.3.3 TOOL LIFE Data Display and Setting (TOOL LIFE Data Screen Page 2) 88
2.3.3.4 Clear of All TOOL LIFE Data (HEAD, NEXT, GROUP LIST Screen Page 1) . 89
2.3.4 Workpiece Coordinate 90
2.3.4.1 Setting Workpiece Coordinate System Offset Data 91
2.3.4.2 Setting External Workpiece Coordinate System Offset Data 91
2.3.4.3 Displaying Machine Position Data 91
2.4 Parameters 92
2.4.1 Machining Parameters 94
2.4.1.1 Process Parameters 94
2.4.1.2 Control Parameters 94
2.4.1.3 Axis Parameters 95
2.4.1.4 Barrier Data 95
2.4.2 Setup Parameters 96
2.5 Program 97
2.5.1 Function Outline 98
2.5.2 Edit type 99
2.5.3 Menu Function 100
2.5.3.1 MDI Screen Menu Function 101
2.5.3.2 EDIT Screen Menu Function 104
2.5.4 Program Edit Operation 108
2.5.4.1 Common Operation Between Screen Editing and Word Editing 108
2.5.4.2 Screen Editing 111
2.5.4.3 Word Editing 117
2.5.5 MDI Screen Extension Operation 138
2.5.5.1 MDI Data Registration in Memory 138
2.5.6 Edit Screen Extension Operation 139
2.5.6.1 Edit Data Call 139
2.5.6.2 New Program Registration and Preparation 143
2.6 Data In/Out 144
2.6.1 Program Erase 145
2.6.2 Program File 149
2.6.3 Program Copy 150
2.6.3.1 Machining Program Copy 150
2.6.3.2 Machining Program Condense 151

2.6.3.3 Machining Program Merge 152
2.6.3.4 Changing the Machining Program Number 153
2.7 Diagnosis 154
2.7.1 Alarm Message 155
2.7.1.1 Tracing of Alarm and Stop Codes 156
2.7.2 Servo Monitor 157
2.7.2.1 Servo Monitor 157
2.7.2.2 Servo Diagnosis 160
2.7.2.3 Power Supply Diagnosis 161
2.7.2.4 Synchronous Error 163
2.7.3 Spindle Monitor 164
2.7.3.1 Spindle Monitor 164
2.7.3.2 Spindle Diagnosis 172
2.7.4 PLC Interface Diagnosis of CNC CPU 173
2.7.4.1 PLC-I/F Setting and Display 173
2.7.4.2 PLC Device Data Display 176
2.7.4.3 PLC Interface Signal Forcible Definition (Single-shot Type) 177
2.7.4.4 PLC Interface Signal Forcible Definition (Modal Type) 178
2.7.4.5 Diagnosis Executed When an Emergency Stop Status Occurs 179
2.7.5 Absolute Position Monitor 180
2.7.5.1 ABS Servo Monitor 180
2.7.5.2 Absolute Position Initialization 181
2.7.6 PLC Axis Monitor 183
2.7.7 Operation History 184
2.7.7.1 Diagnosis of Operation History and the Occurrence Time of History (History Data) 185
2.7.7.2 Correspondence of Operation Keys and Key history 186
2.7.7.3 Input/Output Signal History Covering Range 188
2.7.7.4 Suspending the Operation History Function 188
2.7.7.5 Saving the Operation History Data 189
2.7.7.6 Switching the History Display 189
2.7.7.7 Clearing the Operation History and the Occurrence Time of History (History Data) 190
2.7.7.8 Setting and Displaying the Time and Date of Occurrence of Operation History Data 190
2.7.7.9 Outputting the Storage Data for Operation History 190
2.7.7.10 Precautions 192
2.7.8 System Configuration 193
2.7.8.1 S/W Module Tree (NC System) 193
2.7.8.2 S/W Module Tree(2) (Drive Unit) 193
2.7.8.3 H/W Monitor 194
2.7.8.4 OPTION 194
2.7.9 NC Data Sampling 195
2.7.9.1 Display Items 196
2.7.9.2 Parameters 197
2.7.9.3 Address Designation 203
2.7.9.4 Data Output Format 205
2.7.9.5 Outputting the Data 206
2.7.9.6 Flow of the Operation 207

II. MACHINE OPERATIONS

1. Operation State 2
1.1 Operation State Transition Diagram 2
1.2 Power OFF 3
1.3 Not Ready 3
1.4 Ready 4
1.4.1 Reset 4
1.4.2 Automatic Operation Start 4
1.4.3 Automatic Operation Pause 4
1.4.4 Automatic Operation Stop 4

2. Indicator Lamps 5
2.1 NC Unit Ready 5
2.2 Automatic Operation Busy 5
2.3 Automatic Operation Start Busy 5
2.4 Automatic Operation Pause Busy 5
2.5 Return to Reference Point 5
2.6 NC Alarm 5
2.7 M00 5
2.8 M02/M30 5

3. Reset Switch and Emergency Stop Button 6
3.1 Reset Switch 6
3.2 Emergency Stop Button 6

4. Operation Mode 7
4.1 Mode Selection Switch 7
4.2 Jog Feed Mode 8
4.3 Rapid Traverse Feed Mode 9
4.4 Return to Reference Position Mode 10
4.5 Incremental Feed Mode 12
4.6 Handle Feed Mode 13
4.7 Memory Mode 14
4.8 MDI Operation Mode 15

5. Operation Panel Switches in Operation Mode 16
5.1 Rapid Traverse Override 16
5.2 Cutting Feed Override 16
5.3 Manual Feedrate 16
5.4 Handle/Incremental Feed Magnification Factor 17
5.5 Handle Feed Axis Selection 17
5.6 Manual Pulse Generator 17
5.7 Cycle Start and Feed Hold 18
5.8 Feed Axis Selection 18

6. Operation Panel Switch Functions 19
6.1 All Axes Machine Lock 19
6.2 Chamfering (L system) 19
6.3 Miscellaneous Function Lock 19
6.4 Single Block 19
6.5 Dry Run 20
6.6 Manual Override 20
6.7 Override Cancel 20
6.8 Optional Stop 20
6.9 Optional Block Skip 21
6.10 Manual Absolute 22
6.11 Error Detect 23

6.12 Follow-up Function 23
6.13 Axis Removal 23
6.14 Manual/Automatic Synchronous Feed 23
6.15 Handle Interruption 24
6.15.1 Outline 24
6.15.2 Interruptible Conditions 24
6.15.3 Interruption Effective Axis 25
6.15.4 Axis Movement Speed Resulting from Interruption 25
6.15.5 Path Resulting After Handle Interruption 26
6.15.6 Handle Interruption in Tool Radius Compensation 28
6.15.7 Interrupt Amount Reset 30
6.15.8 Operation Sequence 31

III. MAINTENANCE

1 Maintenance Works 2
1.1 Instruction of Inspection Works 2
2 Daily Inspection 4
3 Display on 7-segment LED 5
3.1 Detailed display of alarm/stop codes 6
3.2 Notes 7
3.3 Examples of LED Display 10
4 Periodic Inspection 17
5 Hardware Replacement Methods 18
5.1 Module Installation 18
5.1.1 Instructions for Handling 18
5.1.2 Installation and removal of module 19
5.2 Battery for CNC CPU 21
5.2.1 Battery Life for CNC CPU 21
5.2.2 CNC CPU Battery Replacement Procedure 22
5.3 Battery inside PLC CPU 23
5.3.1 Battery Life 23
5.3.2 Replacement Procedure for battery inside PLC CPU 24
6. CNC Data Input/Output 25
6.1 Screen Transition to the "CNC Data In/Out" Screen 25
6.2 Setting Communication 26
6.3 "CNC Data In/Out" Screen Display 29
6.4 Operation Windows 32
6.4.1 Function Selection Window 32
6.4.2 Device Selection Window 32
6.4.3 CNC Data Selection Window 33
6.4.4 Copy Confirmation Window 34
6.4.5 Overwrite Confirmation Window 35
6.4.6 Delete Confirmation Window 36
6.4.7 Key Window 37
6.5 Selecting the Function 39
6.6 Selecting the Device 40
6.7 Selecting the Directory 41
6.8 Selecting the Directory and a File 42
6.9 Copying a File 43
6.9.1 Copying any File other than the SRAM.BIN file 43
6.10 Deleting Files 45
6.11 Refreshing the List 48
6.12 Stop the USB Drive 49
6.13 Creating a Directory 50
6.14 Changing a File Name when Outputting the File 51
6.15 Summary of Screen Transition from the "CNC Data In/Out" Screen 53
6.16 Various Status 54
6.16.1 Data Protection 54
6.16.2 CNC Data 55
6.16.3 Messages 57
6.16.4 Parameters 61
6.16.5 Signals 61
6.17 Restrictions 61
7 Data Backup and Restoration 62

7.1 GOT Data Backup and Reinstallation 62
7.1.1 Backup procedures 62
7.1.2 Reinstallation Procedures 64
7.2 PLC/CNC CPU Data Backup and Restoration 65
7.2.1 Data Backup 66
7.2.1.1 Backup procedures 66
7.2.1.2 Backup Files Structure 69
7.2.2 Data Restoration 70
7.2.2.1 Restoration procedures 71

IV. APPENDIXES

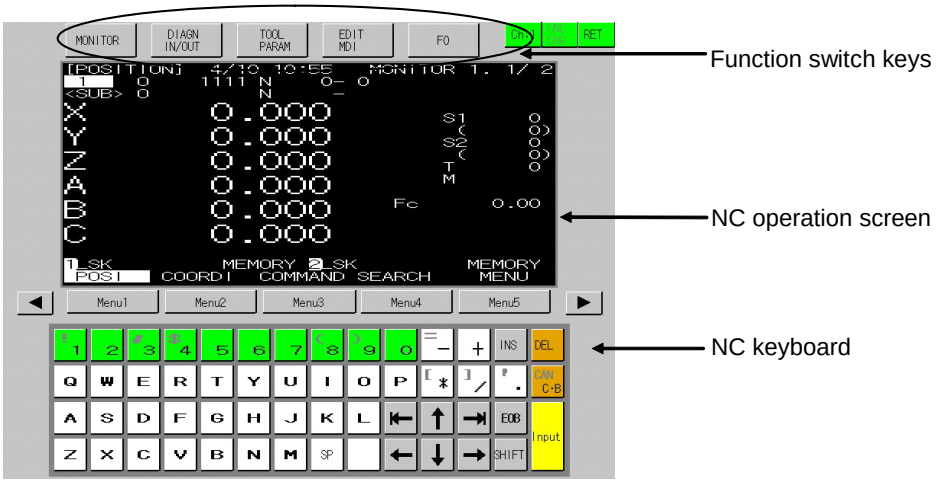
Appendix 1. Registering and Editing Fixed Cycle Programs 1
Appendix 1.1 Parameter for Fixed Cycle Operation 1
Appendix 1.2 Inputting a Fixed Cycle Program 1
Appendix 1.3 Outputting a Fixed Cycle Program 1
Appendix 1.4 Deleting a Fixed Cycle Program 1
Appendix 1.5 Standard Fixed Cycle Subprogram 2
Appendix 2. List of Function Codes 14
Appendix 3. List of Command Value Ranges 15
Appendix 4. Data Protection 16
Appendix 4.1 Data Protection Key 16
Appendix 4.2 Edit Lock B, C 18
Appendix 5. Table of Conversion Codes for Error Code Output 19
Appendix 5.1 Code Conversion Specifications 19
Appendix 5.2 Code Table 20
Appendix 5.3 Restrictions 22
Appendix 6. Operation Messages on Setting Display Unit 23
Appendix 6.1 Operation Errors 23
Appendix 6.2 Operator Messages 30
Appendix 6.3 Dialog Error Messages on Input/Output Screen 34
Appendix 7 Explanation of Alarms 36
Appendix 7.1 Operation Errors (M) 36
Appendix 7.2 Stop Codes (T) 53
Appendix 7.3 Servo/Spindle Alarms (S) 60
Appendix 7.3.1 Servo Errors (S01/S03/S04) 60
Appendix 7.3.2 Initial Parameter Errors (S02) 75
Appendix 7.3.3 Parameter Errors (S51) 77
Appendix 7.3.4 Servo Warnings (S52) 78
Appendix 7.4 MCP Alarms (Y) 81
Appendix 7.5 System Alarms (Z) 99
Appendix 7.6 Absolute Position Detection System Alarms (Z7*) 108
Appendix 7.7 Emergency Stop Alarms (EMG) 112
Appendix 7.8 Auxiliary Axis Operation Errors (M) 116
Appendix 7.9 Built-in PLC Alarms (U) 118
Appendix 7.10 Multi CPU Errors (A) 119
Appendix 7.11 Network Errors (L) 150
Appendix 7.12 Program Errors (P) 160
Appendix 8 User Parameters 180

I. SCREEN OPERATIONS

1. Setting and Display Unit

1.1 Graphic Operation Terminal (GOT)

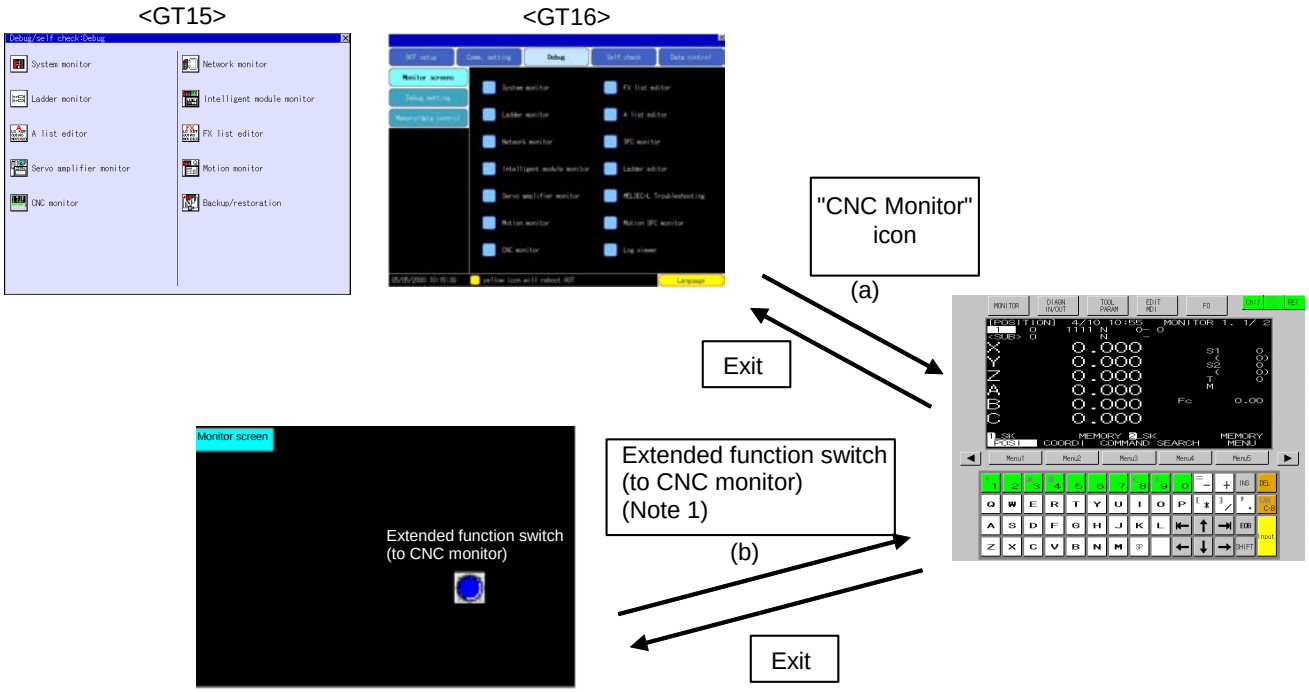
An NC operation screen and an NC keyboard shown below are displayed on GOT.



This screen is dedicated for NC operation and its size is fixed.

1.2 Screen Operation of GOT

Using GOT enables the NC screen as follows.



(Note) Refer to “2.1.2 Screen Transition Diagram (L system)” (Refer to 2.1.3 for M system.) for details.

NC screen appears by either method.

(a) Select “CNC monitor” key on Utility screen.
(b) Select the extended function switch (to CNC Monitor) (Note 1) on Monitor screen.

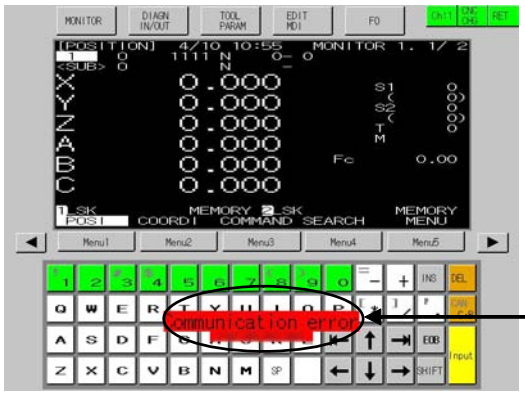
(Note 1) This switch differs depending on the machine. Refer to the instruction manual issued by the machine tool builder.

When NC monitor function cannot work, NC data will not be displayed.
Touch the screen to return to the Utility screen.

List of error messages
(If more than one error are occurring at the same time, the error with higher priority is displayed.)

Priority	Error message	Remedy
High	E71 communication driver is not installed. (NC monitor)	Install E71 communication driver.
Middle	Monitor data not found. (NC monitor)	Download the NC screen data (special module screen).
Low	The IP address of NC to monitor is not set up. (NC monitor)	Set the IP address by the drawing S/W (Ethernet setting), and download the monitor data.

When a communication error occurs during monitoring, the error disappears at recovery of communication.



Communication error

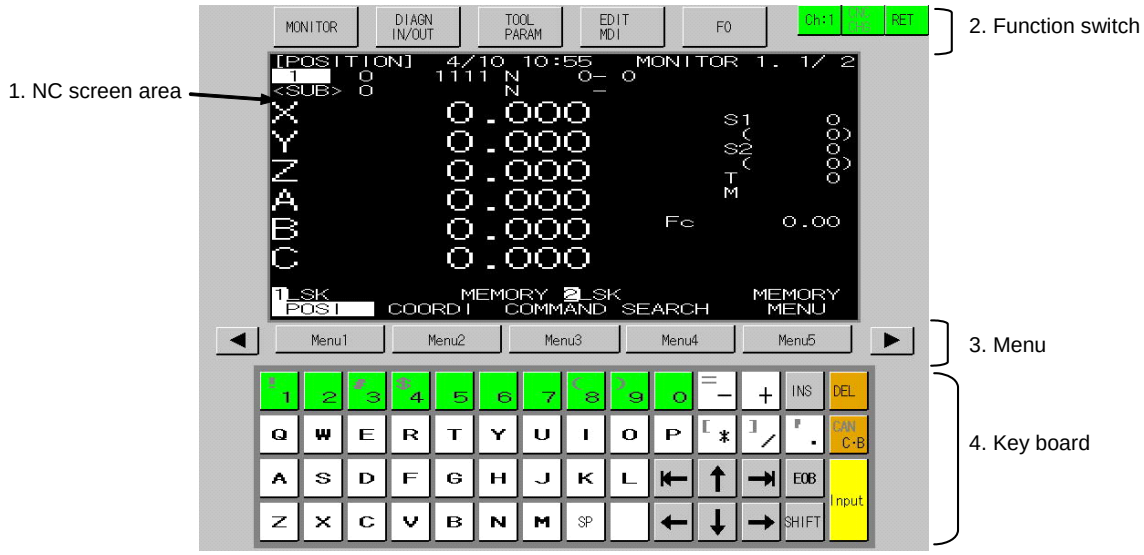
In the message display area, the alarm or warning message that has the highest priority among the alarms currently occurs under the current part system.

Message	Details of message	Remedy
Communication error	An error has occurred in the communication with the NC.	• Check the connection between the NC and GOT. (Cable connection, noise, etc.) • Check the NC and GOT network settings.
KEY OPERATION INVALID	The NC-dedicated display unit or display unit other than touch panel is connected, and key inputs from the menu section or keyboard section are invalid. Inputs from the display unit other than touch panel are valid in this case.	The key inputs become valid by following operations. • Disconnect the NC-dedicated display unit. • Press the OPERATE menu.

When some error occurs with the communication with NC, a dialogue box will appear.
Refer to the appendix written later for details of the other alarms or messages.

2. CNC Monitor Screen

In this screen, the various information which are needed to setup and maintain the machine and NC system are displayed and set.



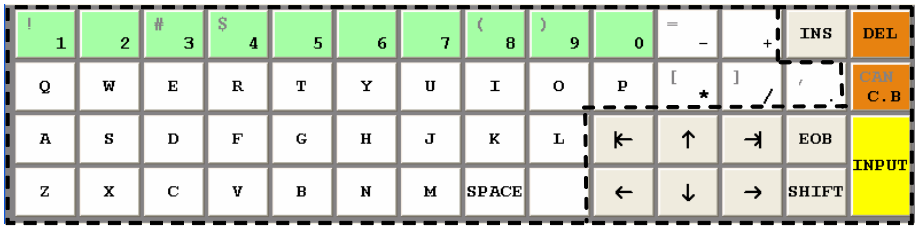
(1) Display items

Display items	Details
1. NC screen area	The functions such as position monitoring, alarm diagnosis, tool offset/parameters, and program edit.
2. Function switch	The following keys used to select the display function.
MONITOR	Selects position monitoring function. Refer to “2.2 Monitor”.
DIAGN IN/OUT	Selects alarm diagnosis function or data input/output function. Refer to “2.7 Diagnosis” and “2.6 Data In/Out.
TOOL PARAM	Selects tool offset function and parameter setting. Refer to "2.3(I) Tool Offset(L system)", "2.3(II) Tool Offset(M system)" and “2.4 Parameters”.
EDIT MDI	Selects program edit function. Refer to “2.5 Program”.
3. Menu	
MENU 1 to 5	Changes a screen according to menu. If the NC-dedicated display is valid, the "KEY OPERATION INVALID" message and OPERATE menu will appear. Select this menu when operating the keys on the GOT.
BACK	Displays the previous page if there are multiple pages in a screen.
NEXT	Displays the next page if there are multiple pages in a screen.
4. Keyboard	This is used to set a data in NC screens. Refer to “(2) Composition of displayed keyboard“ for details.

(2) Composition of displayed keyboard

Composition of the displayed keyboard is shown below.

Data setting keys



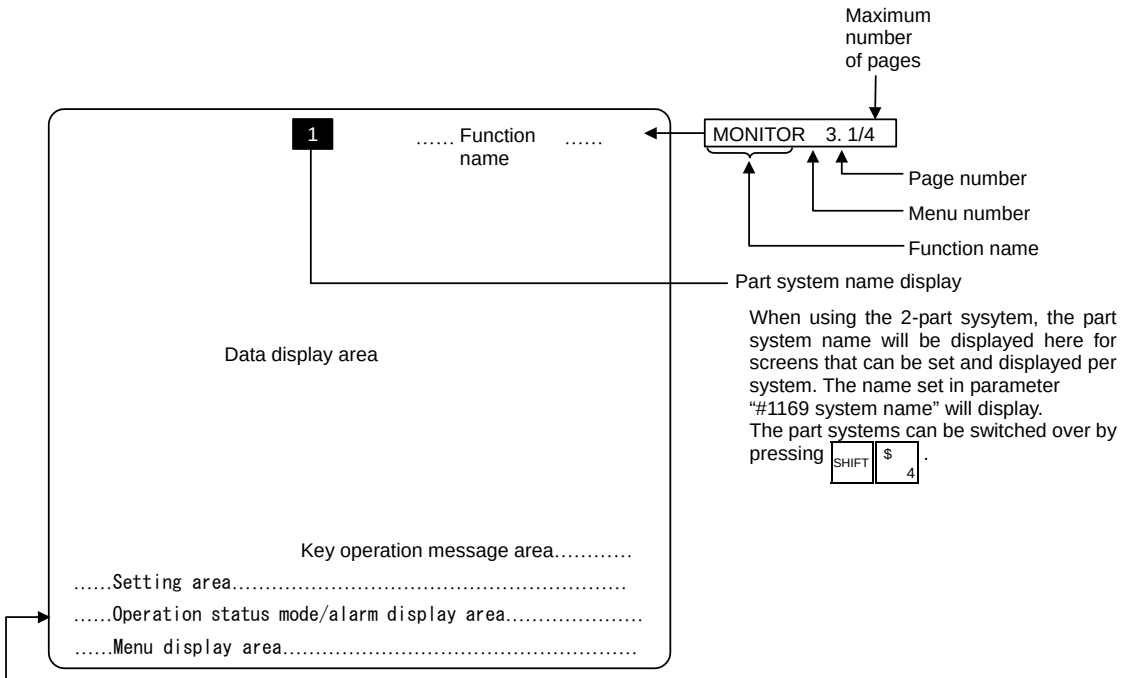
Special keys

Key type	Keys	Function
Data setting keys	A ~ Z ! 1 2 # 3 $ 4 5 6 7 (8) 9 0 SP = - + [*] / , .	These keys are used to set alphabetic characters (only capital letters), numbers, space, and symbols, etc. The characters in gray can be set after touching SHIFT key.
Special keys	↑ ↓ ← →	Moves a cursor up/down/left/right. (Repeat function is available)
	→| |←	Moves a cursor one block forward/backward. (Repeat function is available)
	DEL	Deletes a character at the cursor.
	INS	Changes an insertion mode.
	CAN C.B	Deletes one block at the cursor while a machining program is edited. Touching CAN C.B key after SHIFT key deletes all blocks displayed in the screen.
	EOB	Adds EOB (;) code in the editing machining program.
	INPUT	Fixes the setting.
	SHIFT	Changes the key function.

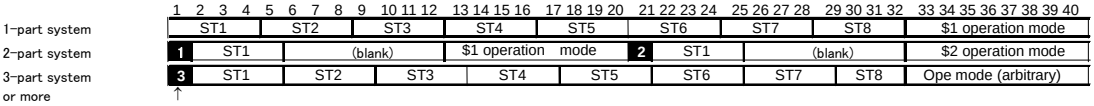
2.1 Setting and Display Unit Operation

2.1.1 Display Area of NC Screens

NC screen display area is divided into the following four areas:
(1) Data display area
(2) Operation status mode and alarm display area
(3) Menu display area
(4) Setting area and key operation message area



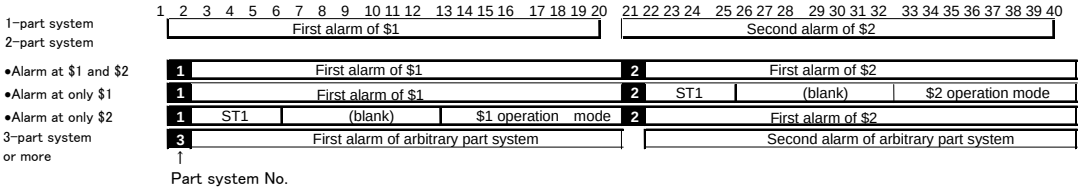
Operation status mode display and menu display (during normal operation) (Note 1)



This is displayed when 6 or more menus exist.
The selected menu is reverse-displayed.

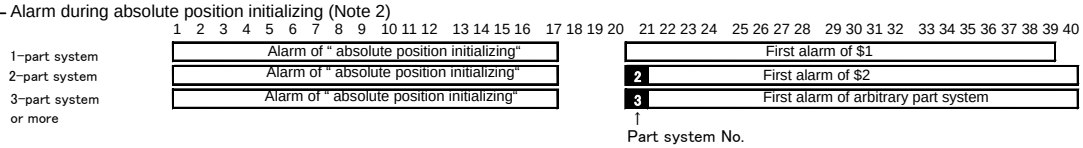
ST1: Operation status (EMG, RST, LSK, etc.)
ST2: Metric/Inch command (mm/in.)
ST3: Absolute/Incremental command mode (ABS/INC)
ST4: During tool radius compensation/Cancel (G40 to G42)
ST5: Workpiece coordinate system (G54 to G59, P01 to P48)
ST6: Executing subprogram. (SB1 to SB8)
ST7: Executing fixed cycle. (fix)
ST8: (Not used.)

Alarm message display (during alarm occurrence)



Alarm is highlighted and message (warning) is normally displayed.

Alarm during absolute position initializing (Note 2)



(Note 1) When the other display units are also connected, the message as "KEY OPERATION INVAID" and OPERATE menu are displayed.
To invalidate the key operation on the panel computer, press the OPERATE menu.

(Note 2) While the absolute position initializing is being executed in a part system, all the part systems, including the systems which are not under the execution, show "absolute position initializing" in their each operation status mode/alarm display area.

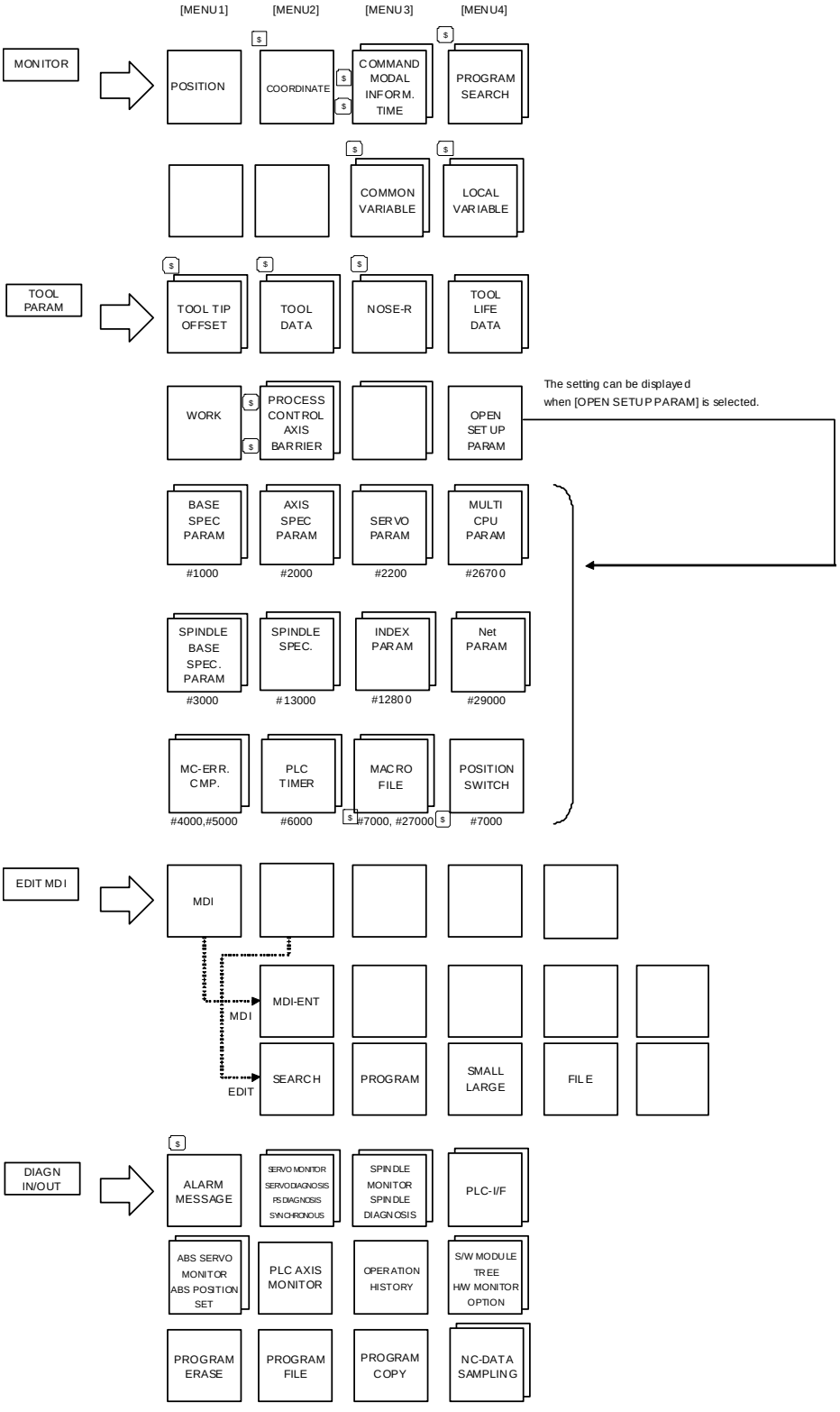
(1) Operation status mode

Position	Display symbol	Explanation
ST1	EMG	During emergency stop
	RST	During reset
	LSK	When paper tape reader is in label skip state
	HLD	During feed hold stop
	STP	During single block stop
	␣␣␣	Normal operation state other than the above
ST2	mm	Metric command
	in.	Inch command
ST3	ABS	Absolute command mode G90
	INC	Incremental command mode G91
ST4	G40	Tool radius compensation cancel state
	G41	During tool radius compensation (left)
	G42	During tool radius compensation (right)
ST5	G54	Selection of the workpiece coordinate is indicated.
	G59	
ST6	␣␣␣	This indicates that subprogram is not executed.
	SB1 SB2:	Machining program execution is controlled according to subprogram data. Each value indicates the subprogram depth.
ST7	fix	Fixed cycle is being executed.
	␣␣␣	Fixed cycle is not executed.
ST8		

(Note 1) ␣␣␣ denotes blank display.

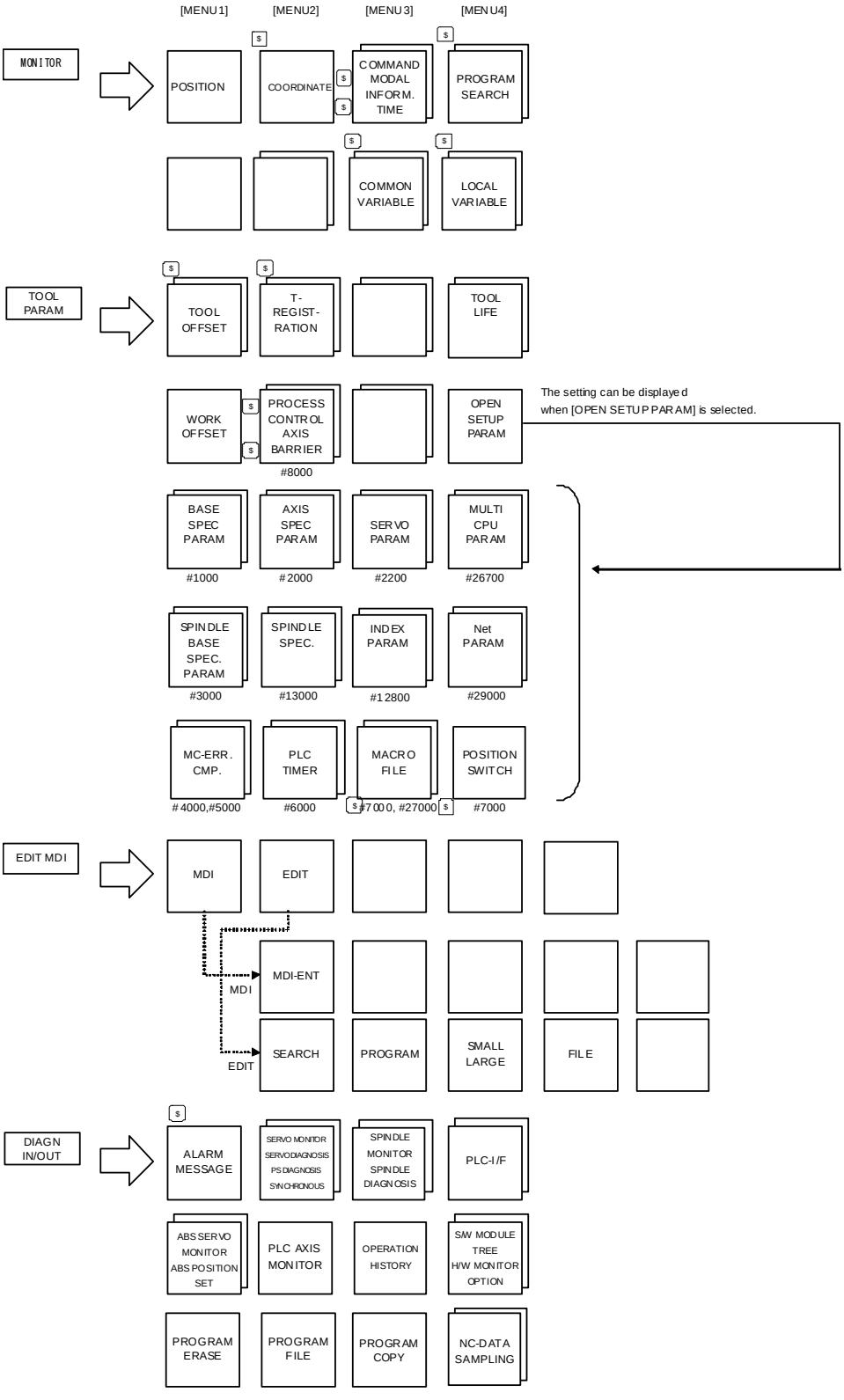
2.1.2 Screen Transition Diagram (L system)

If the multi-part system is provided, the displayed part system for screens indicated with the [$] mark will change when the [SHIFT] and [4] keys are pressed.
All data may not fit on one page according to the number of part systems or number of axes.
Use the [BACK] and [NEXT] keys to change the page in this case.



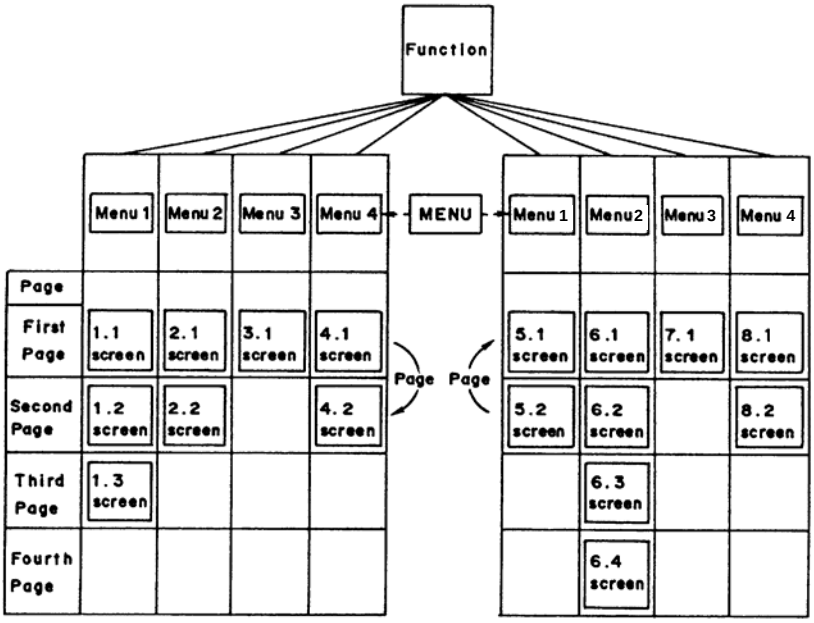
2.1.3 Screen Transition Diagram (M system)

If the multi-part system is provided, the displayed part system for screens indicated with the [$] mark will change when the [SHIFT] and [4] keys are pressed.
All data may not fit on one page according to the number of part systems or number of axes.
Use the [BACK] and [NEXT] keys to change the page in this case.



2.1.4 Screen Selection Procedure

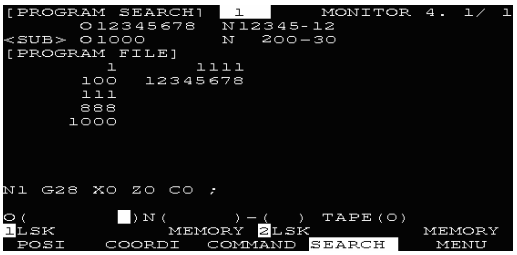
Select a screen according to the following procedure:
(1) Select a function screen by using the appropriate function key.
(2) Select a menu screen in the function by using the appropriate menu key.
(3) Select a page in the menu screen by using the page key.



(1) Select a function screen

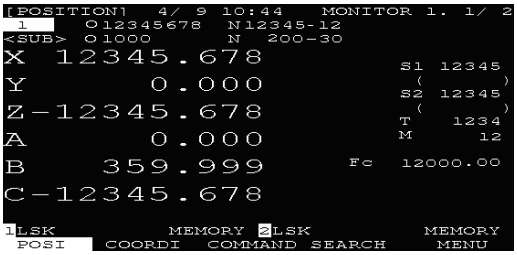
[MONITOR] [DIAGN IN/OUT] [TOOL PARAM] [EDIT MDI]

Press the function selection key corresponding to the function screen to be displayed.
(Example) Press the [MONITOR] key.



1) The previously displayed menu screen is displayed in the data display area.
2) The first display screen after power is turned on is the screen on the first menu.

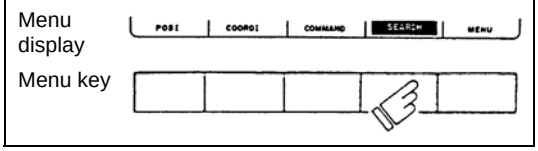
If the same function selection key is pressed again, a return is made to the first page screen of the first menu.
(Example) Press the [MONITOR] key again.

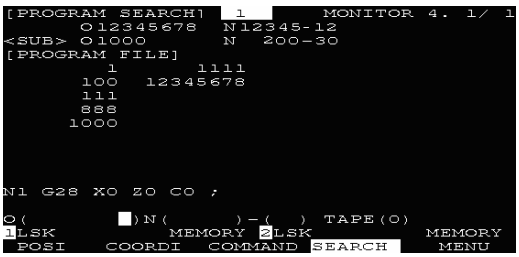


(2) Select a menu screen in the function

Up to five menus are displayed at a time. When a menu key below the menu display is pressed, the menu screen corresponding to the menu key is displayed.

Press the menu key corresponding to the menu display.

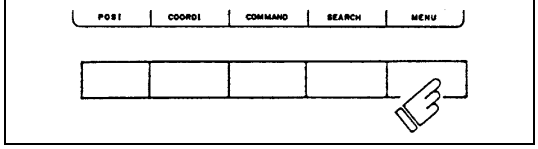


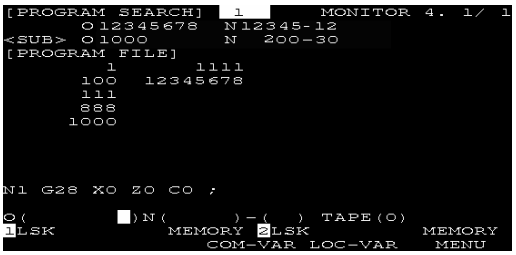


1) The selected menu screen is displayed in the data display area.
2) The selected menu is highlighted in the menu display area.

When the rightmost menu in the menu display area is MENU, it indicates that other menus than the displayed menus exist. Make menu change by pressing the menu key below MENU, then select the menu screen to be displayed.

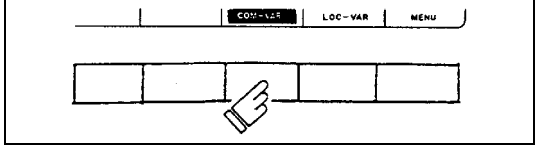
(1) Press the menu key.





1) Only the menu display area is changed and the remaining menu group is displayed.

(2) Press the menu key corresponding to the menu display.

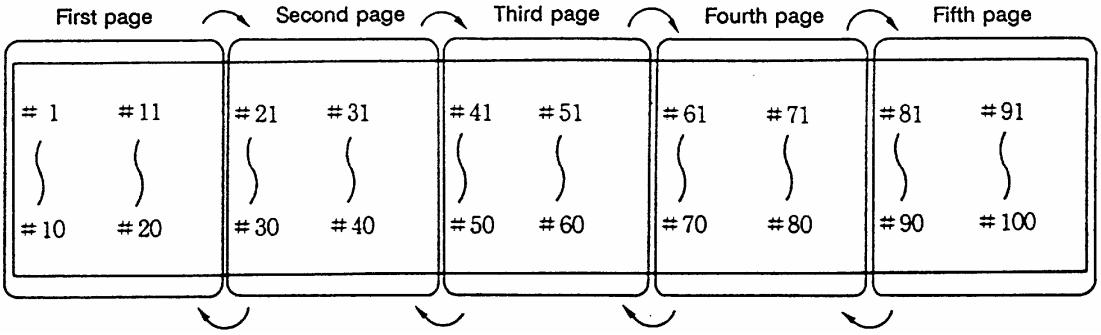


(3) Select a page in the menu screen

When the menu screen contains a number of pages, feed pages by using the page key.

The rightmost page key (NEXT) displays the "next page". The leftmost page key (BACK) displays the "previous page".

Using the rightmost key NEXT , feed page.

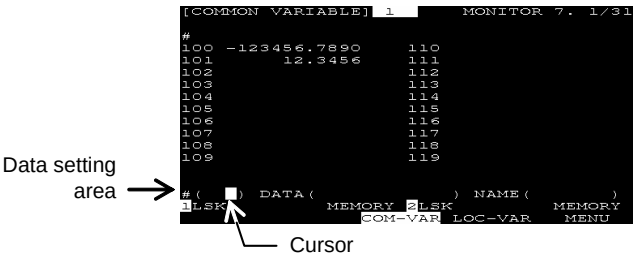


Using the leftmost key BACK , feed page.

2.1.5 Data Setting Method

(1) Outline of data setting

The data setting method consists of the following steps.
When a screen is selected, the cursor is displayed in the right end within the first parentheses in the setting area.



(1) Enter the data number.

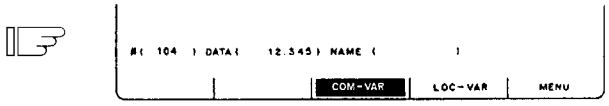
Enter the number of the data to be set by using the numeric keys.
(Example) To set data in #104, press
[1] [0] [4] .

(2) Move the cursor.

Press the [→] key.

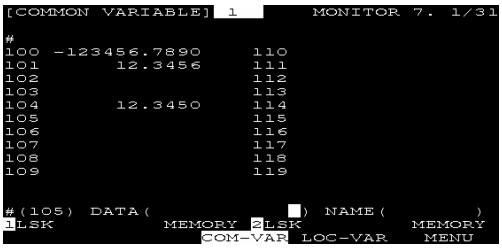
(3) Press data keys.

Seeing the data display area contents, enter new data by using the keys.
(Example) To change to 12.345, press
[1] [2] [.] [3] [4] [5] .



(4) Press the input key.

Check the setup contents displayed in the setting area and set the data in memory by pressing the [INPUT] key.



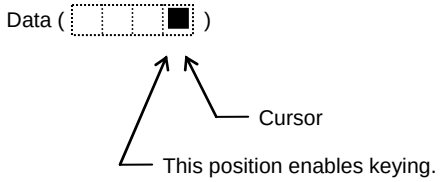
1) Data setting processing is performed according to the setting area contents, and the result is displayed in the data display area.
2) The data number in the setting area is incremented by one, and the cursor is displayed in the right end within the second parentheses.

3) To consecutively set data, repeat (3) and (4).
4) The data No. can be changed by the following methods:
   - Press [INPUT] key. (the No. increases one by one)
   - Press [↓] key. (the No. increases one by one)
   - Press [↑] key. (the No. decreases one by one)
   - Move the cursor to the data No. setting area using [|←] or [→|] key, and change it directly.

(Note 1) Data in the setting area is only displayed on the screen and is not set in memory until the [INPUT] key is pressed. If the screen is changed before the [INPUT] key is pressed, the data in the setting area becomes invalid.

(2) Cursor control and operation examples

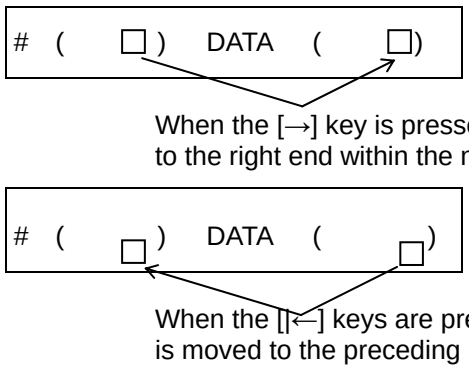
1) Data write into the display screen (by keying) is made at the position indicated by the cursor. When the cursor is not displayed, keying is not effective.



2) When any key is pressed, already displayed data is moved one column to the left and the data corresponding to the key pressed at the cursor position is displayed.

DATA (1[2]) When [#/3] is pressed, DATA (12[3])

3) If a number of parentheses exist in the data setting area, pressing the [→] key when the cursor is in the right end within parentheses causes the cursor to move to the right end within the next pair.



4) When the [DEL] key is pressed, the data at the cursor position is deleted. To cancel one character entered by using any data key, etc., use the [DEL] key.

If you press [3] [3] by mistake,	☞	# (12) DATA (123[3])
If you once press the [DEL] key,	☞	# (12) DATA (12[3])
If you again press the [DEL] key,	☞	# (12) DATA (1[2])

Each time the [DEL] key is pressed, one character of data at the cursor position is deleted and the data to the left of the deleted character is moved one column to the right.

5) Data in parentheses where the cursor exists is erased by pressing the [C.B CAN] key.

Display is made in the setting area as shown in the right.	☞	# (10) DATA (12.34 5)
If you press the [C.B CAN] key,	☞	# (10) DATA (☐)

6) Data in all parentheses in the setting area is erased by pressing [SHIFT] [C.B CAN] .

Display is made in the setting area as shown in the right.	☞	# (10) DATA (12.34 5)
If you press [SHIFT][C.B CAN] ,	☞	# ( ) DATA (☐)

7) The cursor in parentheses is moved one column to the left or right by pressing the [←] or [→] key desired character of data entered by using the data keys can be corrected.

Display is made in the setting area as shown in the right.	☞	# (10) DATA (12.34 5)
If you make successive four strokes of the [←] key,	☞	# (10) DATA (1 2 .345)
If you press [3] ,	☞	# (10) DATA (13 . 345)

2 is corrected to 3 and the cursor is moved one column to the right.

If you press the [→] key,	☞	# (10) DATA (13. 3 45)

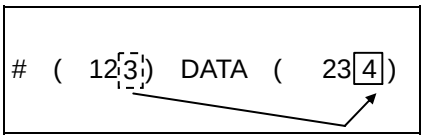
The cursor is only moved one column to the right.

If you press [0] [0] [0] ,	☞	# (10) DATA (13.00 0)

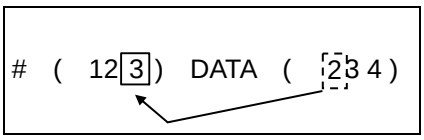
The character at the cursor position is rewritten and the cursor is also moved one column to the right. Data is corrected in sequence.

(Note 1) If [→] is pressed when the cursor exists in the right end within one parentheses, the cursor is moved to the right end within the following parenthesis part; if [←] is pressed when the cursor exists in the left end within one parentheses, the cursor is moved to the right end within the preceding parentheses.

8) When the [→|] keys are pressed, the cursor is moved to the right end within the following parentheses.



If you press the [→|] key, the cursor is moved to the right end within the following parentheses.



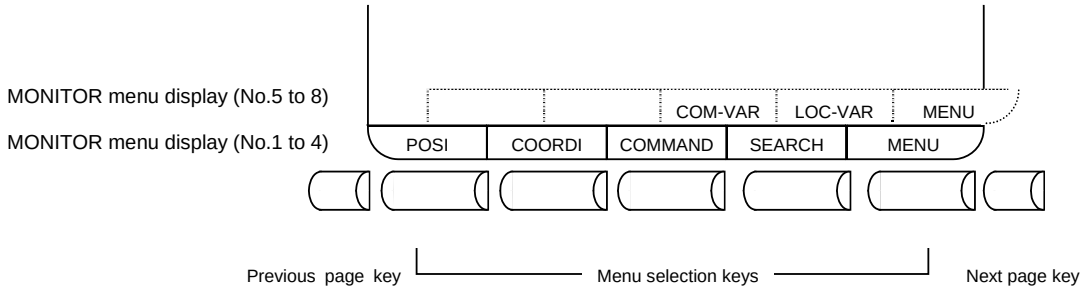
If you press the [|←] key, the cursor is moved to the right end within the preceding parentheses.

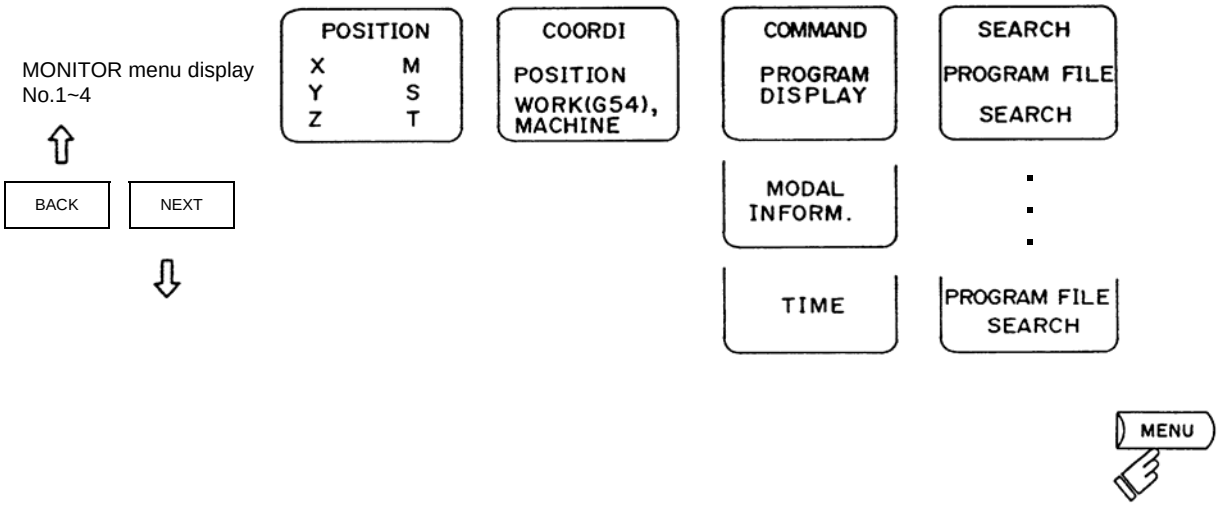
(3) Miscellaneous information

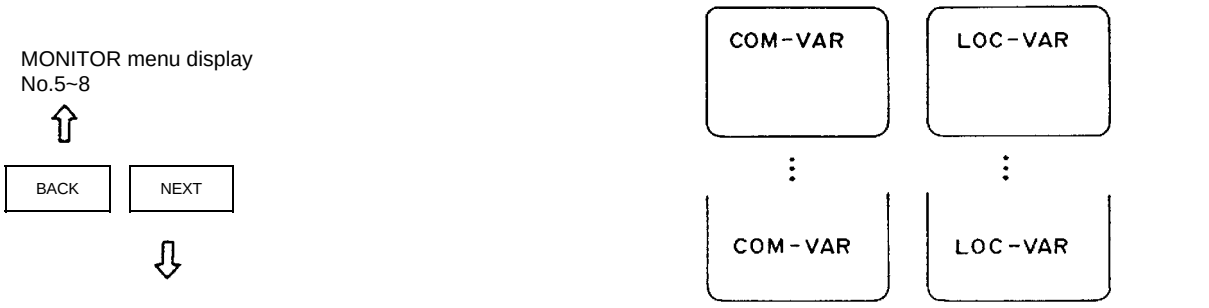
1) Data can also be set by other special methods. See the appropriate items. (For example, manual numeric command setting is performed by the reverse display setting method.)
2) If an invalid key is pressed when data is set within parentheses, a "setting error" will occur at input time and the data will not be accepted. Again set correct data from the beginning.

2.2 Monitor

When the function selection key [MONITOR] is pressed, the following menu appears:







2.2.1 Position

When the menu [POSI] is pressed, the POSITION screen is displayed.

(6-axis specifications)

The following can be performed on the POSITION screen:

(1) Origin set
The current value (POSITION) data of each axis can be set to 0.

(2) Manual numeric command
Miscellaneous function output of M, S, T, etc., can be set through the screen.

Display item	Explanation
4/9 10:44	The date and the time are displayed.
O12345678 N12345-12	The currently executing program number, sequence number, and block number are displayed.
<SUB> O1000 N200-30	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.
X-12345.678 Y 0.000 Z-12345.678 A 0.000 B 359.999 C-12345.678	The current position during execution and its abbreviation (if the position is specific or is placed in specific state) are displayed. #1~#4 (first to fourth reference point positions)] [(servo off state) > < (axis removed state) MR (mirror image)
S1 12345 ( ) S2 12345 ( )	The spindle rotation command value is displayed. The actual spindle speed is shown in ( ). When multiple spindles are used, the 3rd and following spindles are displayed on the following pages.
T 1234 M 12 Fc 12000.00	The tool command value is displayed. The last four digits of the miscellaneous function command value are displayed. During interpolation feed, the speed in the current vector direction in moving is displayed. During independent axis feed, the speed of the axis with the highest speed is displayed.

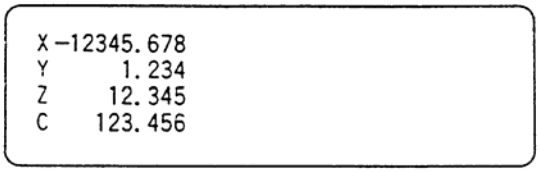
2.2.1.1 Position Display Counter Zero and Origin Zero

Counter Zero
The POSITION display only is set to zero and the absolute value data remains unchanged.

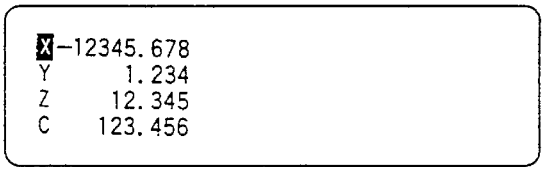
Origin Zero
This sets both POSITION display and absolute value data to zero.
It is equivalent to G92 X0 Y0 Z0 ;.

(Note) Origin zero is valid only when "#1123 origin" is set

In the following operations, the [INPUT] key leads to the counter zero function and the [C.B CAN] key leads to the origin zero (set zero) function.

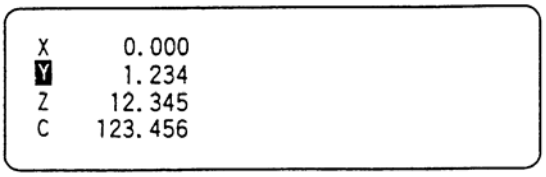


Press the address key [X] .

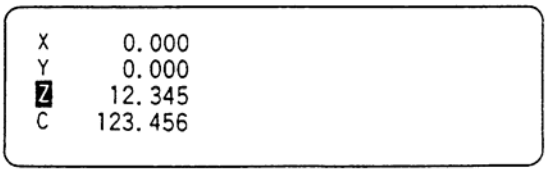


1) The address indication corresponding to the key is highlighted.

Press the [INPUT] key (counter zero) or [C.B CAN] key (origin zero).



1) The axis position data is set to zero and the next axis name is highlighted.
2) By repeatedly pressing the [INPUT] or [C.B CAN] key, the position data of other axes can be cleared to zero.
3) Upon completion of zero clear of final axis, the display is no longer reversed.
4) If you press an axis address key midway, the address of specified axis is highlighted.
5) When you press a key other than axis address key, the display is no longer reversed.



2.2.1.2 Manual Numerical Value Command (S, T, M)

You can easily execute spindle function S, tool function T and miscellaneous function M by operation on the screen. Namely, you can key in S, T and M commands as if they were commanded by a program.

(1) Conditions that allow manual numerical value command

M, S or T command sequence is not under way. Even during automatic start or pause, for example, the manual numerical value command is available if above conditions are met.

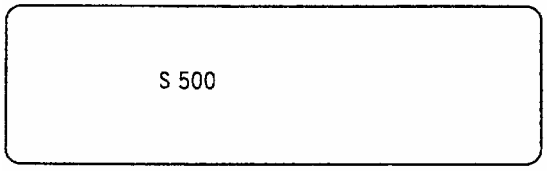
(2) Operating procedures of manual numerical value commands

1) Select the position display [POSI] menu screen.
2) Press the address key corresponding to the command. This causes the corresponding commanded value display section to be highlighted and makes the system ready for input of manual numerical value command. The spindle function key is [S] **(Note 1)**, tool function key is [T] and miscellaneous function key is [M].
3) Key-in the numerical value to be input.
4) Press the [INPUT] key.

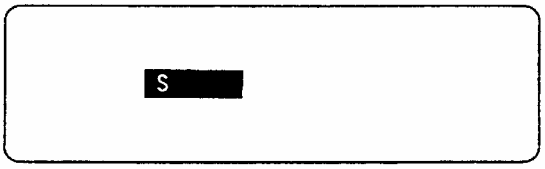
(Note 1) [S] is displayed when there is only one spindle, and [S1], [S2] etc are displayed when there are more than one spindle. Manual numerical value command cannot be performed for the 2nd miscellaneous functions.

(Example) The procedures to execute S1200 by manual numerical value command are given below. First select POSITION display on MONITOR screen.

On screen, last executed command value is displayed. ☞



Press the address key [S] . ☞

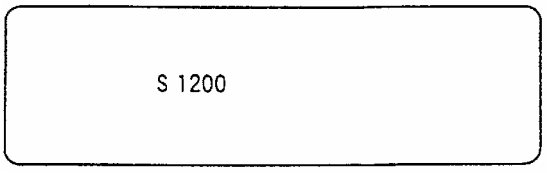


1) The address corresponding to the pressed key and numerical value setting range are highlighted.

Set the numerical value by number keys.
[1] [2] [0] [0] ☞

1) The set numbers are displayed successively as highlighted.

Press the [INPUT] key. ☞



1) The S command is executed.
2) The reversed display on screen returns to normal.

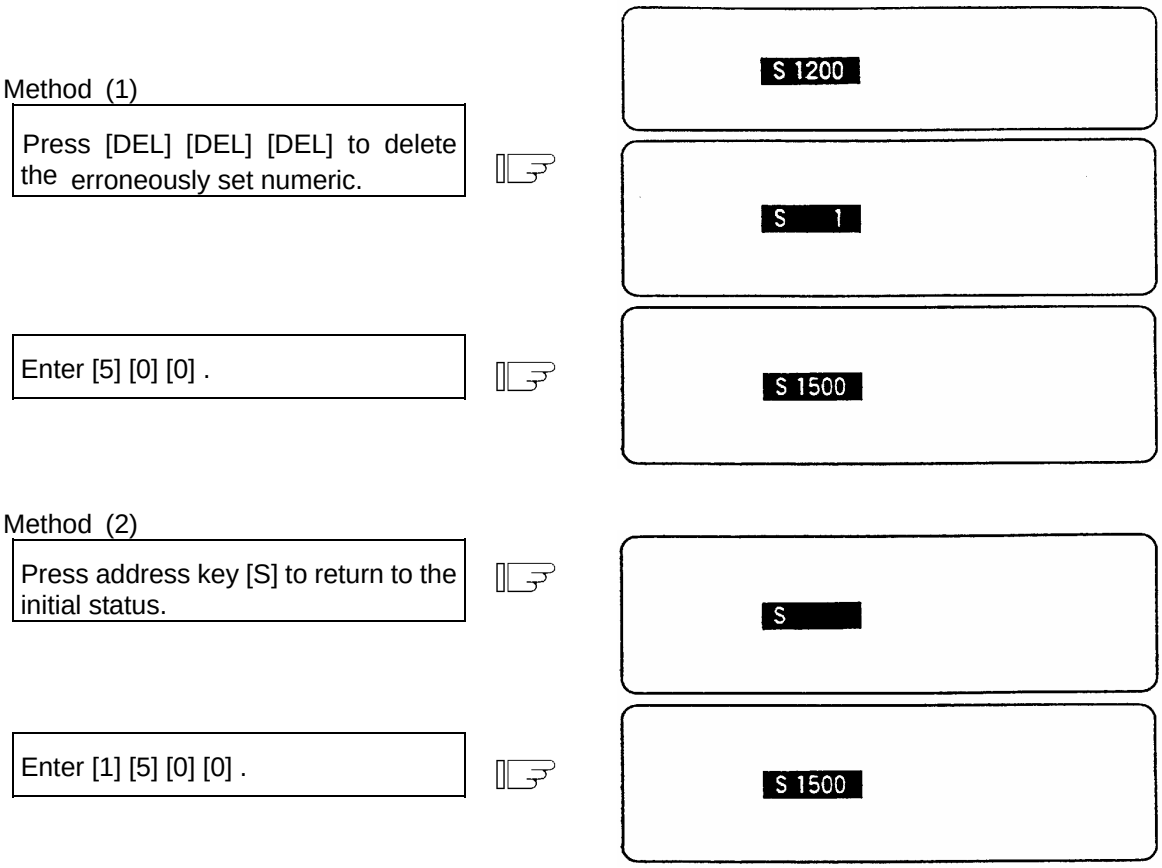
(3) Action to be taken when an erroneous numeric is set and the correct one is desired to be set

There are two methods:

Method (1) While pressing the [DEL] key, delete the set digits one by one. Then, retry to enter the correct digits.

Method (2) Retry the entry, beginning with pressing the address key corresponding to the command.

(Example) In this condition, the numeric is desired to be replaced by S1500.



(4) Setting/output range of manual numeric command

The setting and output range for the manual numeric command are indicated.

	BCD	Signed binary
M	0~9999	
S		±99999
T	0~9999	

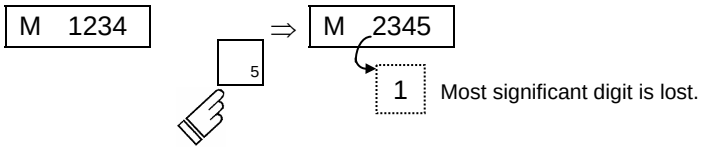
(Note 1) If the type is BCD output and a negative number is set, the positive value converted from it will be output.

(Example) Manual numeric command

		Output
M −100	⇒	M 100

(Note 2) If the number of digits specified in the command exceeds the setting range, the most significant digit will be lost.

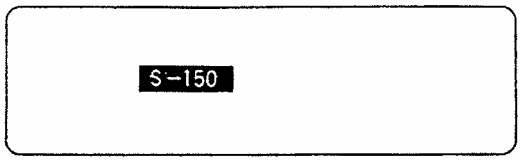
(Example)



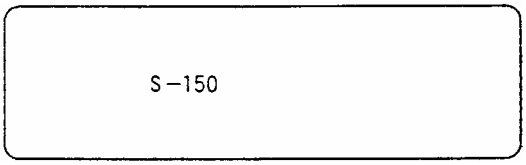
(5) Other notes on operation

(1) When a minus command is set:
Before setting the numeric, press the [-] (minus) key.
(Example) If S-150 is specified:

Press address key [S] .
Then, press key [-] [1] [5] [0] in order.



Press [INPUT] key.



(2) When manual numeric command operation stops halfway:
If the operation is desired to be stopped before input after pressing the address key, press any non-numeric key.
(a) If a manual numeric command address-key such as MST is pressed, the previous operation will stop. In this case, the next manual numeric command sequentially begins.
(b) If an axis address key (X, Y, or Z, etc.) is pressed, the manual numeric command will stop. In this case, the origin zero or counter zero mode is then entered.
(c) If the [SHIFT] [C.B CAN] keys are pressed, the manual numeric command will stop. In this case, the POSITION screen is blanked.
(d) If one of the following keys is pressed, the operation will not stop:
- Position display function key [MONITOR]
- [-] key pressed before a numeric is set (will be processed as a minus command.)
- [DEL] key when a numeric has been set (The set data will be deleted.)

(3) The macro interruption codes (M96, M97) and subprogram call codes (M98, M99) will not be processed even if they are issued.
(4) No peripheral-speed command is available.
In the constant peripheral speed mode, no command is processed, if specified.
(5) The set data will be canceled if screen change is executed during manual numeric command operation.
(6) If operations in which manual numerical commands are carried out (M, S, T keys) are attempted when the manual numerical command protect function is valid, the error message "E05 SETTING NOT POSSIBLE" will occur.

2.2.1.3 Displaying Automatic Operation Program

(1) Displaying the operation program during automatic operation
During memory or MDI operation, up to four blocks of the specified program are displayed. The block being executed or the completed block will display at the top line.

(2) Displaying the operation program after SEARCH
In operation program display immediately after SEARCH, the block for which the search command is issued is the command program to be executed next.

(3) Displaying the operation program at branch to or at return from subprogram
When a branch command (M98) block is executed, the subprogram is immediately displayed. When a return command (M99) block is executed, the main program is immediately displayed.

(4) Difference between one block of machining program and one execution block

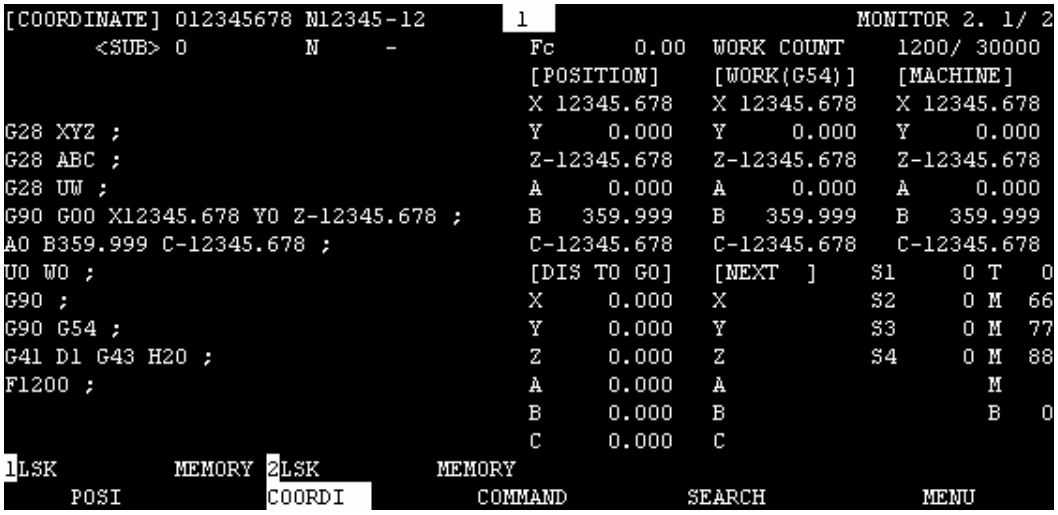
1) A command block containing an EOB only or a command program consisting of only a comment statement is not regarded as an execution block but is processed together with the following command program.

2) A command block such as a variable command that contains neither move nor MSTB command is not regarded as an execution block but is processed together with the next block that contains a move or MSTB command.

(Note) When a parameter "MACRO SINGLE" is ON, a variable command block is regarded as an execution block.

2.2.2 Coordinate

When the menu [COORDI] is pressed, the COORDINATE screen is displayed.



Display item	Explanation
O12345678 N12345-12	The currently executing program number, sequence number, and block number are displayed.
<SUB>	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.
G28 XYZ ; G28 ABC ; G28 UW ; G90 G00 X12345.678 Y0 Z-12345.678 ; :	The current machining program being executed is displayed. This is the same as POSITION screen display.
WORK COUNT: □ / □ Work count Max. work count	Work count : Indicates count data of the number of workpieces that has been machined. Max. work count : The max. workpiece value set in "#8003 WRK LIMIT" is displayed. Display range : 0 to 999999
[POSITION] X 12345.000 Y 0.000 Z-12345.678 A 0.000 B 0.000 C 12345.678	The current position of the axis being executed and the abbreviation of the axis' status are displayed. The display is as same as the POSITION screen display.

Display item	Explanation
[WORK (G54)] X 12345.000 Y 0.000 Z-12345.678 A 0.000 B 0.000 C 12345.678	G54 to G59 workpiece coordinate system modal numbers and the coordinates in the workpiece coordinate system are displayed.
[MACHINE] X 12345.000 Y 0.000 Z-12345.678 A 0.000 B 0.000 C 12345.678	The coordinates of each axis in the basic machine coordinate system, in which the zero point is determined depending on the machine, are displayed.
[DIS TO GO] X 12345.000 Y 0.000 Z-12345.678 A 0.000 B 0.000 C 12345.678	The remaining distance of the move command being executed (incremental distance from the current position to the end point of the block) is displayed during automatic operation start busy or pause busy.
[NEXT] X Y Z A B C	The coordinates of the end point in the block executed next are displayed.
S1:0 :	The program command S modal value (maximum 7 sets) currently being executed are displayed.
T 0	The program command T modal value currently being executed is displayed.
M 66 :	The program command M modal values (maximum 4 sets) currently being executed are displayed.
B	The program command 2nd miscellaneous function modal value currently being executed is displayed.

2.2.3 Command

When the menu [COMMAND] is pressed, the COMMAND screen is displayed.
This screen consists of three pages. It displays the execution program monitor, execution modal monitor, and cumulative time data. Page switching is by pressing the [NEXT] or [BACK] key.

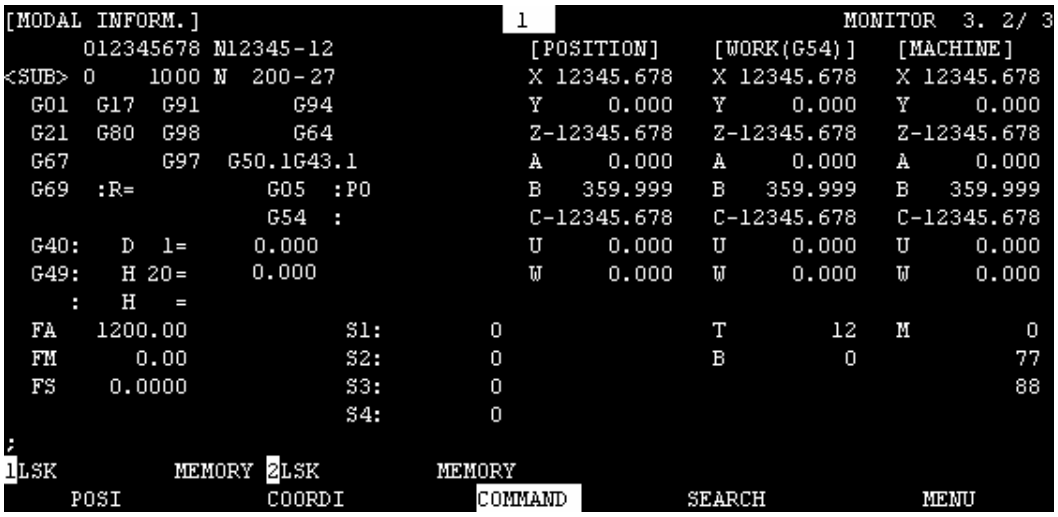
2.2.3.1 Execution Program Monitor

This screen displays the active machining program's execution blocks for monitoring.

Display item	Explanation
O12345678 N0-0	The currently executing program number, sequence number, and block number are displayed.
<SUB> O1000 N200-30	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.
G28 XYZ; G28 ABC ; G90 G00 X12345.678 Y0 Z-12345.678 ; : :	The current program being executed is displayed. The cursor is moved to the top of the current block being executed. When program execution reaches N90, the cursor is also moved to the top of N90. When the N100 block is executed, the N100 block is displayed starting at the top of the screen and the cursor is also moved to the top.
[WORK] X 12345.678 Y 0.000 Z-12345.678 A 0.000 B 359.999 C-12345.678 U 0.000 W 0.000	The coordinates in the workpiece coordinate system being currently executed are displayed.

2.2.3.2 Execution Modal Monitor

By switching the screen from the execution program's monitor screen (in the previous section) by using the [NEXT] key, the execution modal's monitor screen is displayed. This screen mainly displays the modal values of the active machining program for monitoring.



Display item	Explanation
O12345678 N12345-12	The currently executing program number, sequence number, and block number are displayed.
<SUB> O1000 N200-27	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.
[POSITION] X 12345.000 Y 0.000 Z-12345.678 A 0.000 B 359.999 C-12345.678 U 0.000 W 0.000	The current position of the axis being executed and the abbreviation of the axis' status are displayed. The display is as same as the POSITION screen display.
[WORK(G54)] X 12345.678 Y 0.000 Z-12345.678 A 0.000 B 359.999 C-12345.678 U 0.000 W 0.000	The coordinates in the workpiece coordinate system being currently executed are displayed. (Same as POSITION screen display.)

Display item	Explanation
[MACHINE] X 12345.678 Y 0.000 Z-12345.678 A 0.000 B 359.999 C-12345.678 U 0.000 W 0.000	The coordinates of each axis in the basic machine coordinate system, in which the zero point is determined depending on the machine, are displayed.
[MODAL INFORM.] G01.....G94 ʃ ʃ G67.....G50.1G43.1 G69: R=	The modal state of the current G command being executed is displayed. **(Note)** Fixed cycle operation: When a fixed cycle command is executed, the G command in the fixed cycle control subprogram does not reflect the G modal of the calling source program.
G41: D = G43: H = H =	The tool radius compensation modal and offset number and wear amount are displayed. The tool length offset and offset number and wear amount are displayed.
FA 1200.00	The modal value of the current program command F being executed is displayed. (Unit: mm/min)
FM 0.00	The manual feed rate is displayed. (Unit: mm/min)
FS 0.0000	The manual feed rate is displayed. (Unit: mm/rev)
S1: 0 S2: 0 :	The modal value of the current program command S being executed is displayed. (Up to two spindles)
T 12	The modal value of the current program command T being executed is displayed.
M 0 77 88	A maximum of four modal values of the current program command M being executed are displayed.
B 0	The second miscellaneous function modal value of the current program command being executed is displayed.
G28 X0. Y0. Z0. ...	The current program block being executed is displayed.

2.2.3.3 Total Integrating Time Display

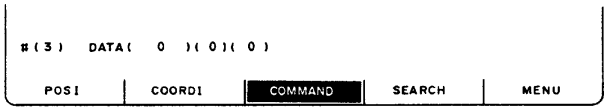
By switching the screen from the execution program's monitor screen by using the [NEXT] key, the TIME screen is displayed.

Display item	Explanation
O12345678 N12345-12	The currently executing program number, sequence number, and block number are displayed.
<SUB> O1000 N200-27	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.
#1 DATE 0/ 4/ 9 2 TIME 11:20:39 3 POWER ON 23:19: 2 4 AUTO OP 6: 7: 15 5 AUTO STL 2: 48: 46 6 EXT TIME 1 0: 0: 0 7 EXT TIME 2 0: 0: 0	The date and time are set. year/month/day hour : min : s The total integrating time in each operation state is displayed.

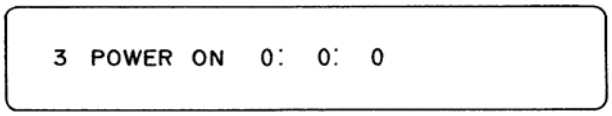
(1) TIME setting

Set the number, hour, minute, and second corresponding to the TIME to be set.

Set 3 in # ( ).
Set 0 in DATA ( ) ( ) ( ).



Press the [INPUT] key.

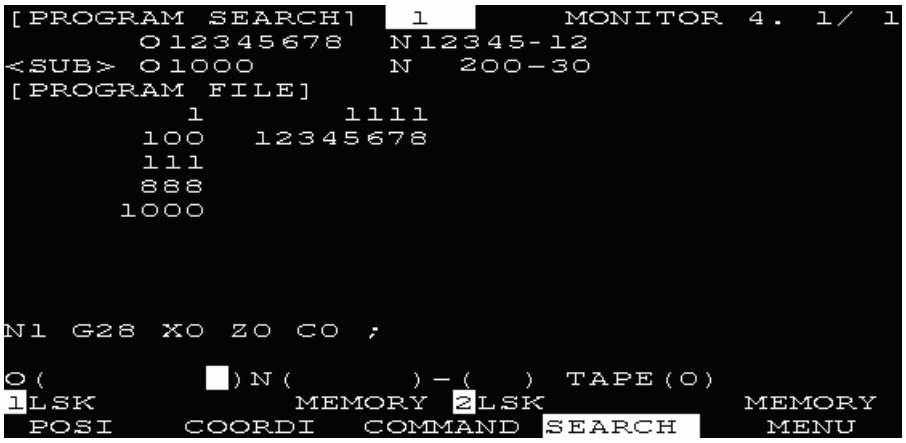


DATE : Set date (set "YEAR" in the Christian Era).
TIME : Set time in 24-hour mode.
POWER ON : Total integrating time of the time from control unit power ON to OFF.
AUTO OP : Total integrating time of the machining time from AUTO STL button pressed in the memory mode to M02/M30 or reset button pressed.
AUTO STL : Total integrating time during automatic starting from AUTO STL button pressed in the memory mode or MDI to feed hold stop, block stop, or reset button pressed.
EXT TIME 1 : Dependent on PLC sequence.
EXT TIME 2 : Dependent on PLC sequence.

(Note) Integration time (#3 POWER ON to #7 EXT TIME 2): When display reaches the maximum value (9999:59:59), integration is stopped and the maximum value remains displayed.

2.2.4 Program Search

When the menu [SERCH] is pressed, the PROGRAM SEARCH screen is displayed.
The PROGRAM SEARCH screen enables you to call the program number, sequence number, and block number for automatic operation from the machining programs registered in memory.



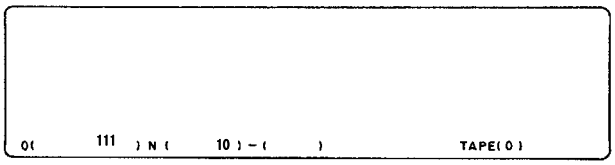
Display item	Explanation
O12345678 N12345-12	The currently executing program number, sequence number, and block number are displayed.
<SUB> O 1000 N 200-30	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.
[PROGRAM FILE] 1 1111 100 12345678 111 888 1000	The numbers of the machining programs registered in memory are listed. The numbers ranging from 1 to 99999999 are displayed in the ascending order. If the number of the registered programs exceeds one page of display, PROGRAM FILE is displayed extending across pages.

After MDI operation is executed, programs cannot be searched for unless reset is executed.
To restart from the middle of the program, search for the restart block, and then carry out MDI operation to restore the modal state.

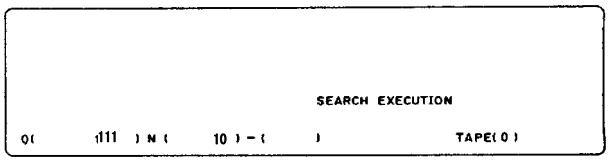
2.2.4.1 Memory Search

Any work program is called from the machining programs registered in memory before work.
Set the program number to be called, the sequence number, and block number. Set the tape search setting area to 0. The initial state when power is turned on is memory search.

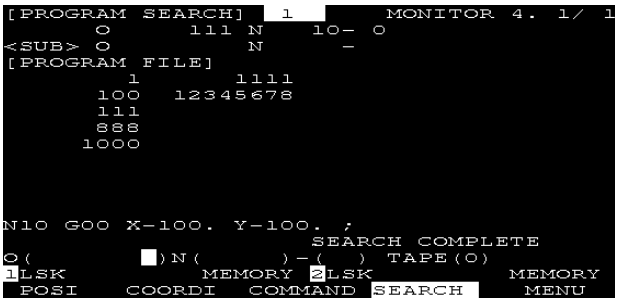
Set the program number to be called.
Set the sequence number and block number as required.
(Example)
To call O111 N10 block,
O (111) N (10) - ( )
TAPE (0)



Press the [INPUT] key.



1) A search is started.
2) When the specified program number, sequence number, and block number are found, SEARCH COMPLETE message is displayed.
   The found numbers are displayed in O and N. Data in the found block is displayed in the work program display area.

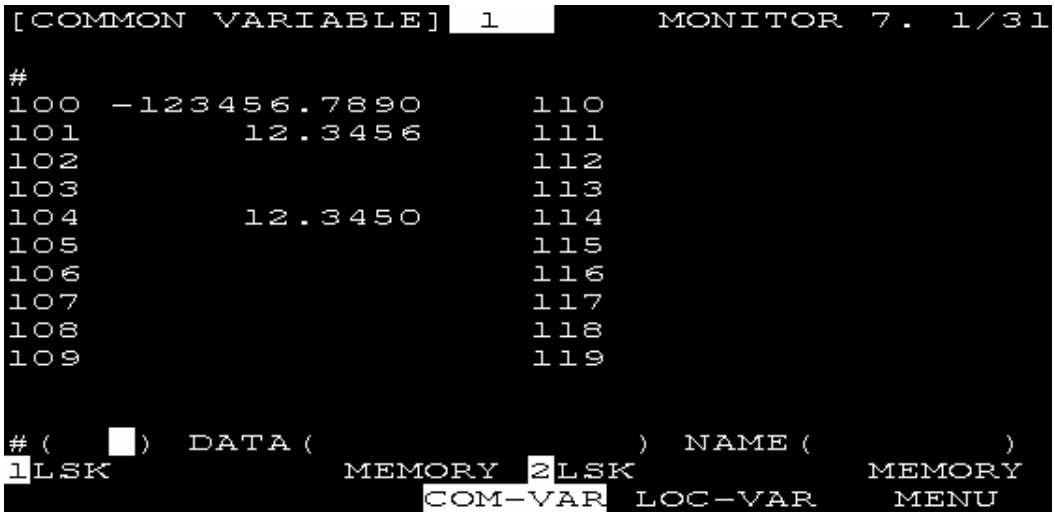


(Note 1) If one of the following operations is executed in the EDIT screen after memory search, the system enters a status in which nothing is being searched. Operation is disabled at this time. In this case, execute the search again.
- Deleting the program being searched.
- Deleting the sequence number for which the search was being executed.
- Deleting the block corresponding to the block number for which the search was being executed.

(Note 2) One block which the control unit executes in one automatic start cycle can be searched in this case. The block with ; (EOB) or sequence number only is not regarded as a one cycle execution block. Axis movement command or control command such as M, S, or T is contained in it.

2.2.5 Common Variable

When the menu [COM-VAR] is pressed, the COMMON VARIABLE screen is displayed. The common variable contents are displayed for the variable command in a machining program.
Common variable data can also be set or changed on the COMMON VARIABLE screen.
The common variable configuration varies depending on the number of variables defined in the specifications. For 100 variables, #100~#149 and #500~#549 are assigned (7-page configuration).



Display item	Explanation
# 100 -123456.7890 101 12.3456 102	The variable numbers and contents are displayed. If variable data is "null", the data display field will be blank. If the number of columns of data is too large (the data contains more than six characters in the integer part or more than four characters in the fraction part), the exponent will be used for display.

2.2.5.1 Common Variable Display

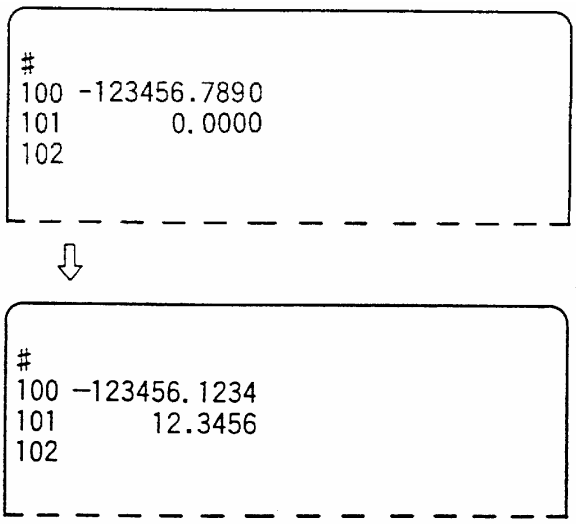
(1) When a common variable command exists, if the block is executed, the execution result is displayed.

(Example)
The following machining program is executed.

}

#101=12.3456

}



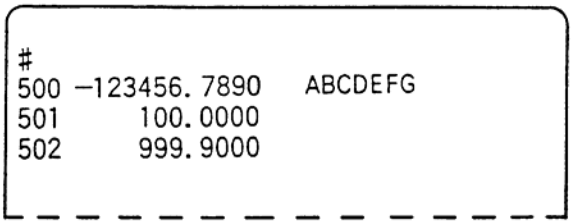
(2) When a command to set variable names for common variables #500~#519 by user macro exists, if the block is executed, the setup variable name is displayed.
Variable name setting and reference commands require the user macro specifications and are limited to 20 common variables #500~#519. The variable name is a string of up to seven alphanumeric characters beginning with an alphabetic character. For common variables #500~#519, the variable numbers, data, and variable names are displayed as shown below:

(Example)
The following machining program is executed.

}

SETVN 501 [POINTER, COUNTER];

}



2.2.5.2 Common Variable Setting

(1) Common variable data setting
To set common variable data, set the variable number in # ( ) and common variable data in DATA ( ), then press the [INPUT] key.

(2) Setting variable names of common variables #500~#519
To set a variable name, set the variable number in # ( ) and the variable name in NAME ( ), then press the [INPUT] key. Only 20 common variables #500~#519 allow variable name setting. The variable name is a string of up to seven alphanumeric characters beginning with an alphabetic character.
If both data and variable names are set for variable numbers (#500~#519), the data and variable name can be set at a time.

(3) If the [INPUT] pressed after the variable number and data (or variable name) are set, the setup data (or variable name) is displayed at the variable number position. The variable number in the setting area # ( ) is automatically incremented (to the next number) and the contents of DATA ( ) and NAME ( ) disappear.

(4) If a variable number and data (or variable name) not listed on the selected page are set, when the [INPUT] first pressed, the screen is changed to the page corresponding to the setup variable number.
If again the [INPUT] pressed, the data (or variable name) is set and displayed at the position of the corresponding variable number.

(5) Whenever the [↓] or [↑] key is pressed for the variable number displayed in # ( ), the variable number can be incremented or decremented by one.

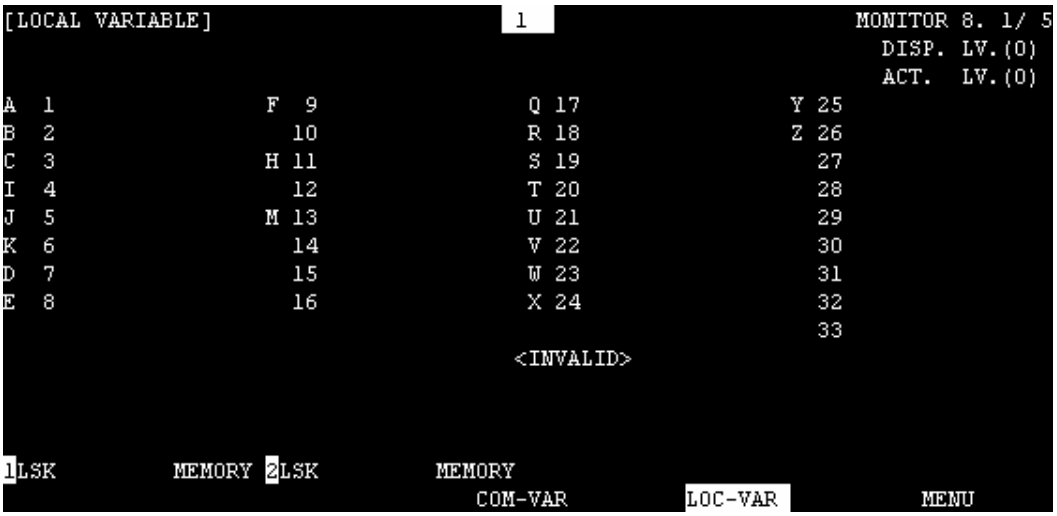
2.2.5.3 Common Variable Data Deleting

To delete all data set as common variables at a time, press the [SHIFT] [C.B CAN] keys, then press the [INPUT] key. This deletes the data displayed one screen.
In this case, data on the other screens are not deleted. If all data on all screens are desired to be deleted, repeat the above operation for all screens.
When [SHIFT] [C.B CAN] keys are pressed, only the display is deleted. When the [INPUT] key is then pressed, the variable data is deleted. This delete operation causes the common variable data to be "null".

(Note) If parameter "#1128 RstVC1" is ON, the common variable data will be cleared to "null" when the system is reset. If parameter "#1129 PwrVC1" is ON, the common variable data will be cleared to "null" when the power is turned ON.

2.2.6 Local Variable

When the menu [LOC-VAR] is pressed, the LOCAL VARIABLE screen is displayed.
Local variables #1 to #33 are provided for each user macro subprogram call level. 33-local variable data is displayed per page and five-page configuration of levels 0 to 4 is used.



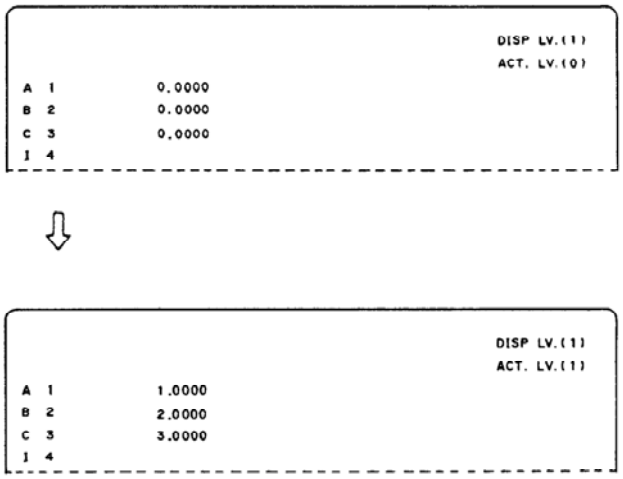
Display item	Explanation
A 1 -12345.6789 B 2 12.3450 C 3	The local variable numbers and contents are displayed. The alphabetic character preceding each local variable number is argument code. None of G, L, N, O, and P can be used as arguments and are displayed. 33 local variables (#1 to #33) exist for each user macro subprogram call level of depth. If variable data is "null", the data display field will be blank. If the number of columns of data is too large, that is, the data contains more than six characters in the integer part or more than four characters in the fraction part), the exponent will be used for display.
ACT. LV. (1)	This indicates the level of depth during user macro subprogram control execution. (0): User macro is not called. (1): User macro call level 1 (2): User macro call level 2 (3): User macro call level 3 (4): User macro call level 4
<INVALID> FEED-HOLD OVERRIDE EXACT SNGL-BLOCK MST-FIN	This indicates the modal state of the operation control status by the #3003, #3004 command. FEED-HOLD : Is displayed when command is programmed with #3004 bit 0 set to 1, indicating that feed hold is invalid. OVERRIDE : Is displayed when command is programmed with #3004 bit 1 set to 1, indicating that cutting override is invalid. EXACT : Is displayed when command is programmed with #3004 bit 2 set to 1, indicating that the G09 (block deceleration check) command is invalid. SNGL-BLOCK: Is displayed when command is programmed with #3003 bit 0 set to 1, indicating that block stop is invalid. MST-FIN : Is displayed when command is programmed with #3003 bit 1 set to 1, indicating the state of proceeding to the next block without waiting for the MST command completion signal.

2.2.6.1 Local Variable Data Display

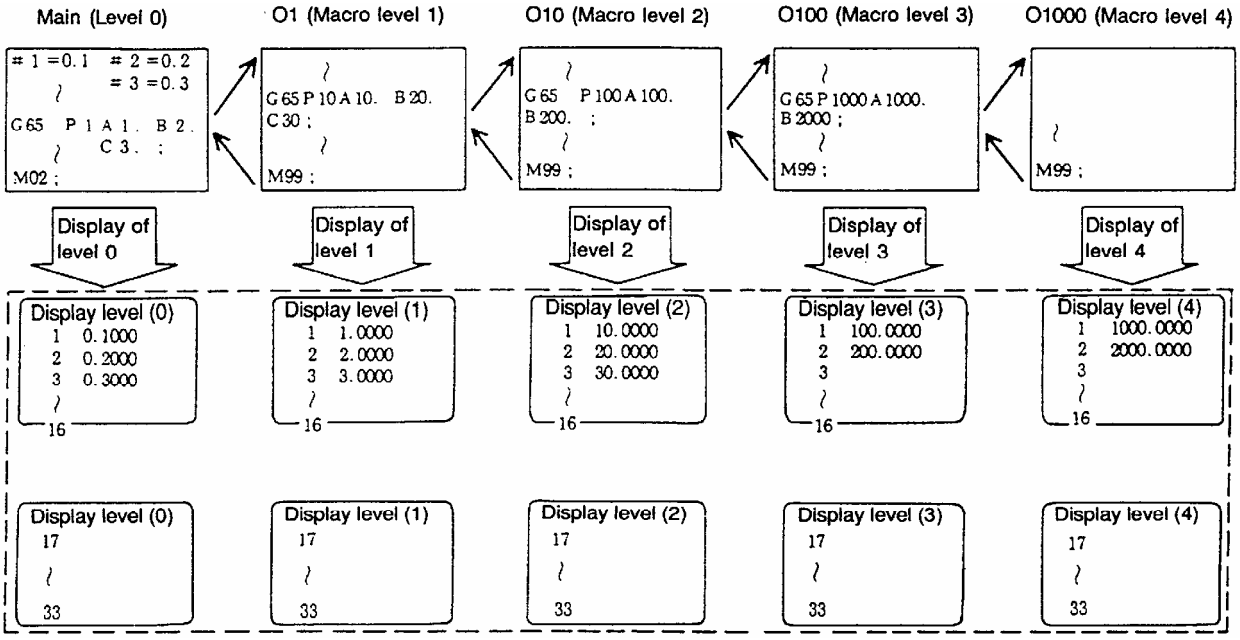
(1) When local variable #1 - #32 command exists in user macro or argument specification is made in user macro subprogram call, if the block is executed, the execution result is displayed.

(Example)
When the following machining program is executed and user macro subprogram is called, data as shown in the right is displayed on the page of local variable display level (1):

```
〈
G65  P1  A1.  B2.  C3.  ;
〈
```



(2) The relationship between the user macro subprogram call execution and display levels is as shown below:

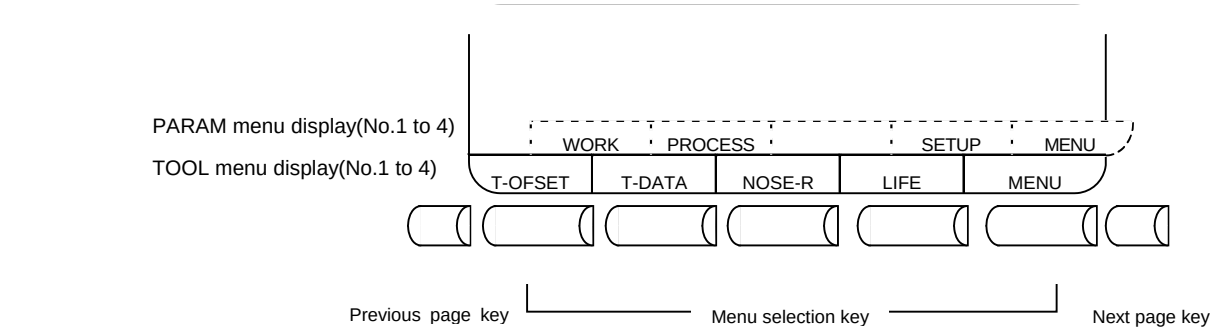


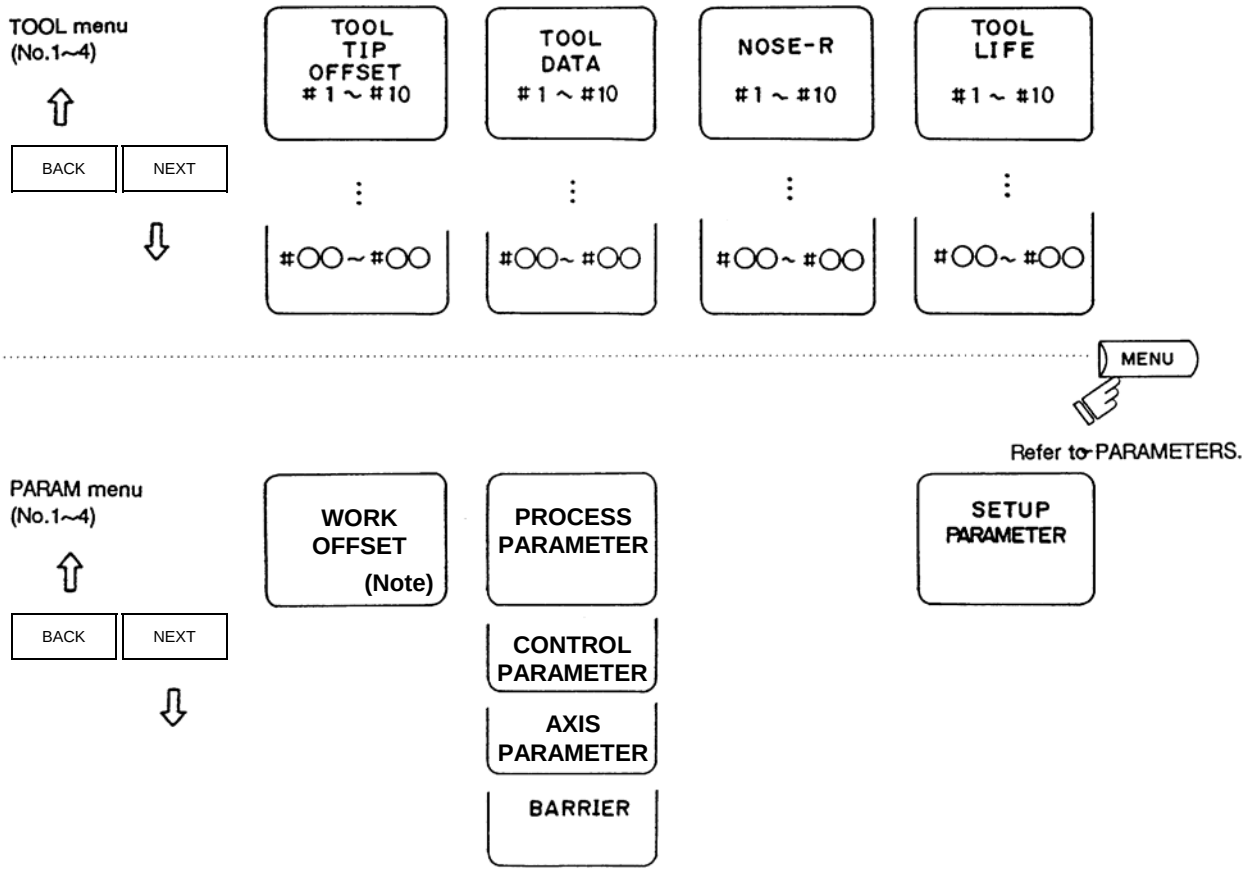
(3) A local variable display page is selected by using the page keys [BACK],[NEXT] Display can be changed as desired independently of the executing level.

(Note) The local variables are not cleared even when power is turned off. They are cleared when a macro is called.

2.3(I) Tool Offset (L system)

The following menu will appear when the function key [TOOL PARAM] is pressed.





(Note) "WORK OFFSET" is included in the TOOL menu.

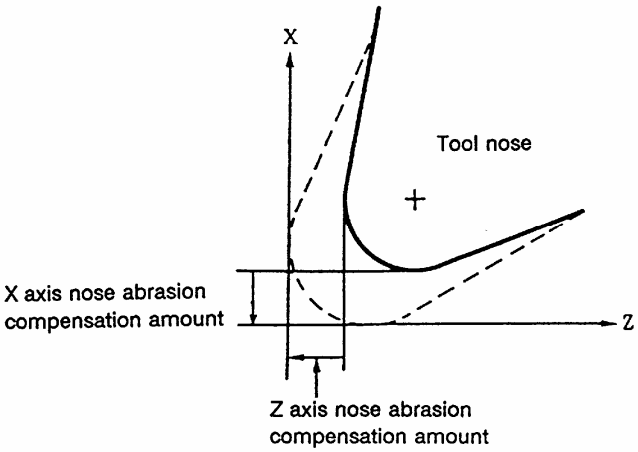
⚠ CAUTION

⚠ If a tool offset amount is changed during automatic operation (including during single block stop), the new offset is validated from the command of the next block or blocks onwards.

2.3.1 Wear Data

The TOOL TIP OFFSET screen will appear when the menu [T-OFFSET] is pressed.

Set the nose wear for each tool used. When the tool compensation No. is designated by the tool command (T command), compensation is carried out matching the tool length of the next screen.



X axis offset
X axis tool length offset + X axis wear offset

Z axis offset
Z axis tool length offset + Z axis wear offset

C axis offset (additional axis)
C axis tool length offset + C axis wear offset

Data	Function
X	X axis tool nose wear compensation
Z	Z axis tool nose wear compensation
C	Additional axis tool nose wear compensation

2.3.1.1 Setting Tool Offset Data

(1) To set the tool offset data, set the offset memory No. in # ( ), and set the offset data in the setting areas corresponding to wear data, tool length data and tool nose data. Then press the [INPUT] key.
(2) If the [INPUT] key is pressed after the offset memory No. and tool offset data are set, the tool offset data set in the corresponding offset memory No. position is displayed. The offset memory No. in # ( ) of the setting area is incremented by 1, and the contents in DATA ( ) disappear.
The No. is not incremented when parameter “#1124 ofsfix” is 1.
(3) If tool offset data and offset memory Nos. other than those in the display are set, the screen changes to the screen corresponding to the set offset memory No. when the [INPUT] key is first pressed. The offset memory is displayed when the [INPUT] is pressed again.
(4) By pressing the [↑] and [↓] keys, the offset memory No. displayed in # ( ) can be continuously incremented or decremented by one.
(5) Tool offset data setting range

Screen	Item	Function	Setting range (unit)	
			"#1003 iunit" = B	"#1003 iunit" = C
TOOL TIP OFFSET	X, Z, C	Tool wear	± 99.999 (mm)	± 9.9999 (mm)
TOOL DATA	X, Z, C	Tool length offset	±999.999 (mm)	±999.9999 (mm)
NOSE-R	R	Tool radius (nose R)	0 to 999.999 (mm)	0 to 99.9999 (mm)
	r	Tool radius (nose R) wear	0 to 99.999 (mm)	0 to 99.9999 (mm)
	P	Tool nose point	0 to 9	0 to 9

(Note) When parameter “#1019 dia” (diameter command) is set to 0, set the radius. When it is set to 1, set the diameter.

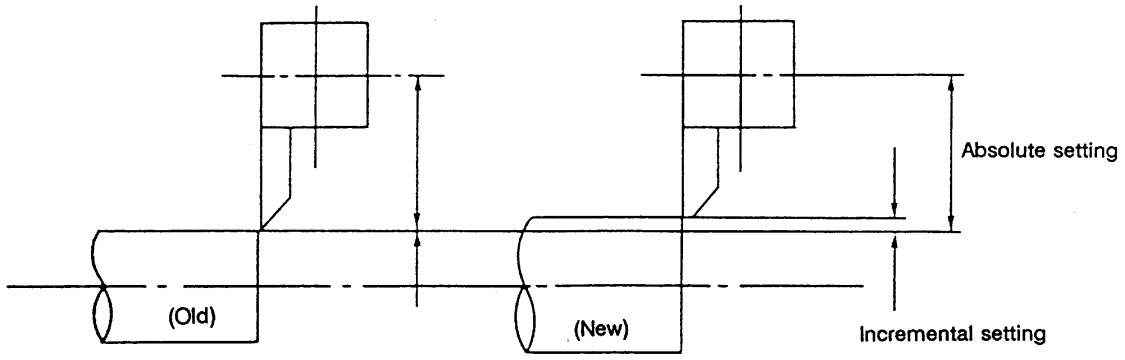
(6) Whether to use the 3rd axis or 4th axis for the additional axis' tool compensation is selected by the parameter "#1520 Tchg34".

2.3.1.2 Erasing the Tool Offset Data

(1) Erasing the display screen units
Ten sets of tool offset data units are displayed on one screen. To set all the displayed offset data to 0, press the [SHIFT] key, the [C.B CAN] key, and finally the [INPUT] key.

2.3.1.3 Tool Wear and Tool Length Data Setting Mode (incremental/absolute)

Whether tool offset data is set as an absolute value or as an incremental value is designated on the TOOL TIP OFFSET or TOOL DATA screen.



(Example) Incremental/absolute value setting

Display
#2 X -100.000

☞

Setting	Display
• Incremental value setting	
# (2) (- 0.1) ⟶	#2 X-100.100
• Absolute value setting	
# (2) (- 100.1) ⟶	#2 X-100.100

(1) Change over to the incremental setting mode

Set I in # ( ), then press the INPUT key.
(I) X ( ) Z ( )

☞

[TOOL DATA]
#I:INC. #A:ABS.

"INC." is highlighted, and the mode becomes the incremental value setting mode.

(2) Change over to the absolute value setting mode

Set A in # ( ), then press the INPUT key.
(A) X ( ) Z ( )

☞

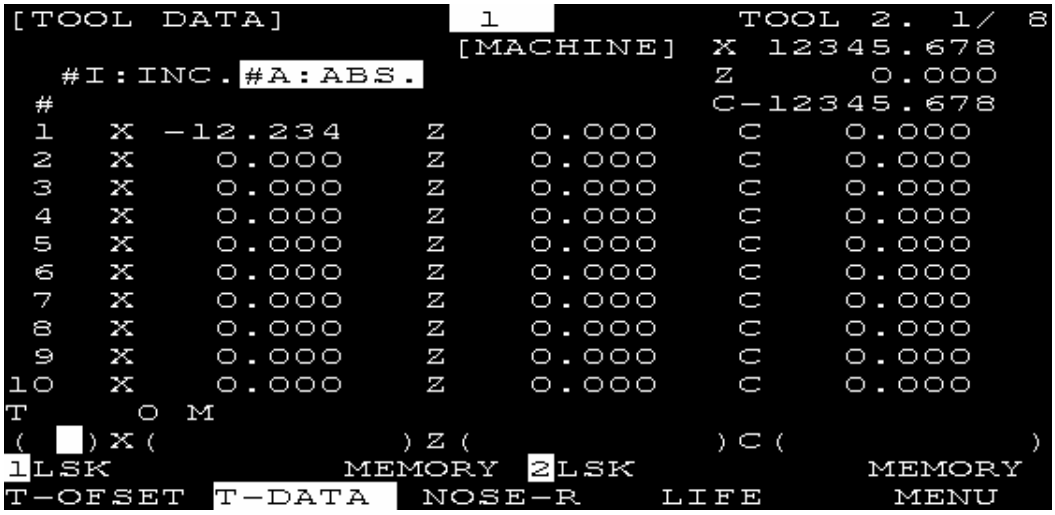
[TOOL DATA]
#I:INC. **#A:ABS.**

"ABS." is highlighted, and the mode becomes the absolute value setting mode.

(Note) The mode is held even if the screen is changed or the power is turned OFF.

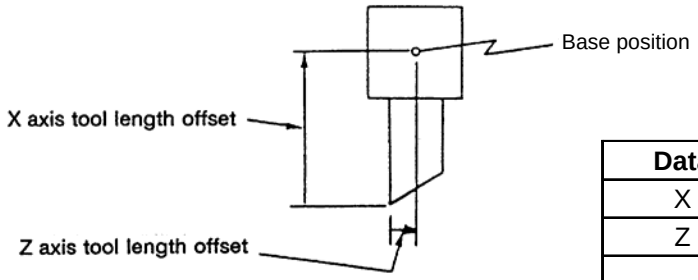
2.3.2 Tool Length Data

The TOOL DATA screen will appear when the menu [T-DATA] is pressed.



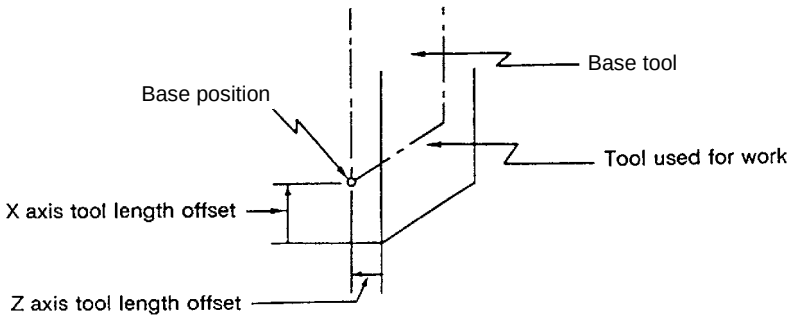
Set the tool length in respect to the programmed base position of each tool used.
When the tool compensation No. is designated by the tool command (T command), compensation is carried out matching the wear data of the previous screen. Generally, the programmed base point position is either the tool rest center position or the base tool nose position.

(1) Tool rest center position



Data	Function
X	X axis tool length compensation
Z	Z axis tool length compensation
C	Additional axis tool length compensation
MACHINE	Same value as on the MONITOR screen.

(2) Base tool nose position



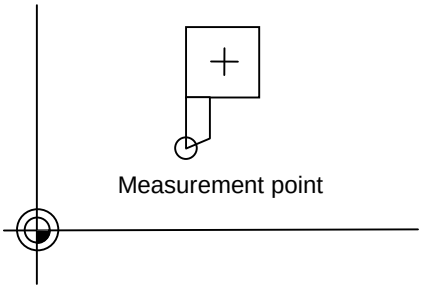
2.3.2.1 Manual Tool Length Measurement I

(1) Outline

This function automatically calculates the amount of tool length compensation, by moving the tool to the measurement point with the manual feed. There are two types of measurement methods in manual tool length measurement I: the base point method and the measurement value input method. The required method is selected by setting parameter “#1102 tlm”.

(a) Base point method

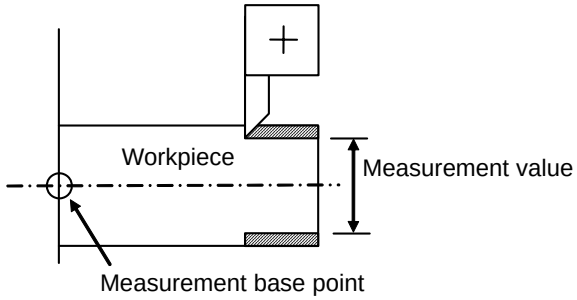
Obtain the tool length with the tool nose placed on the measurement point.



Set the measurement point in parameter “#2015 tlml” beforehand.

(b) Measurement value input method

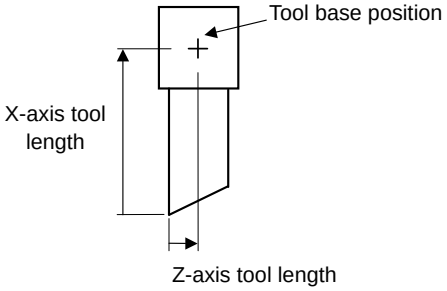
Actually cut the workpiece. Measure its dimensions, and obtain the tool length from the measured values.



The measurement base point is characteristic for each machine (the center of the chuck face, etc.).

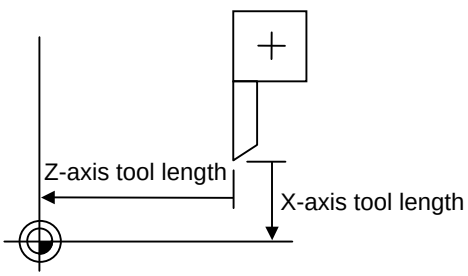
(Note) The tool length from tool length measurement I is as follows, depending on the whether the 1st reference point coordinate values have been set.

If the 1st reference point coordinate values have been set:



If the 1st reference point coordinate values have been set, the tool length is the distance from the tool's hypothetical nose to the tool base position.

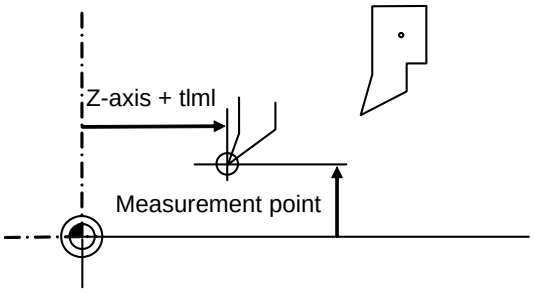
If the 1st reference point coordinate values have not been set:



If the 1st reference point coordinate values are set to 0, the tool length is the distance from the tool's hypothetical nose to the machine base position.

(2) Base point method

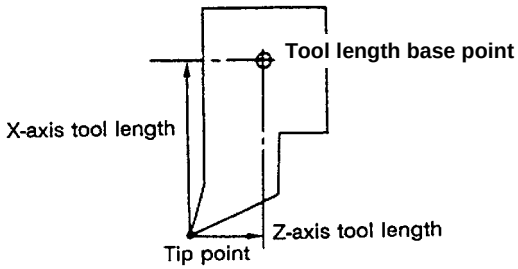
Set the type selection to the base point method. (Set “#1102 tlm” to 0).
To carry out the reference point method, a point to place the tool nose on (measurement point) is required.
Set the measurement point in parameter "#2015 tlml" beforehand.



(Note) Always set the measurement point with the radius, regardless of the diameter/radius command.
Set the measurement point in the machine coordinate system.

Tool length = Machine value - Measurement point (tlml)

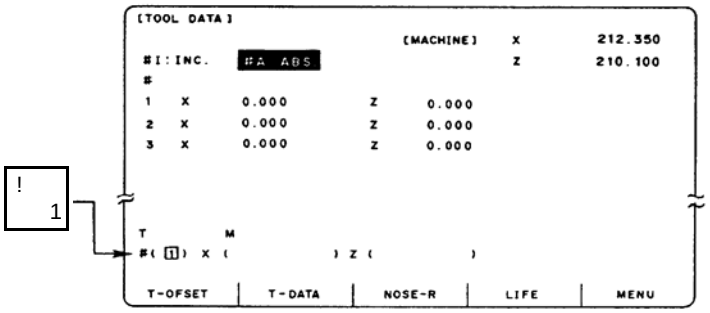
The expression above is used for automatic calculation in the base point method. When the tool nose is placed on the measuring point, the distance from the tool nose to the tool length base point is calculated.



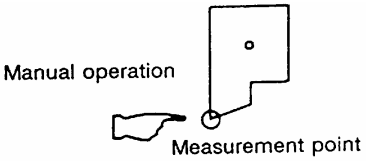
<Measuring procedure for the base point method>

(1) Select the TOOL DATA Screen.
(2) Set the tool No. to be measured in # ( ).
(Select the tool before this step. It can be selected using a manual numerical command.)

(Example) Select tool length No. "1".



(3) Manually place the tool nose on the measuring point.



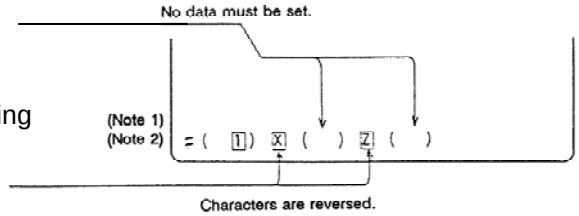
(4) Select the axis to be measured.

X axis - [X]

Z axis - [Z]

Press the address key of each axis.
The selection is canceled by pressing the same address key twice.

Measure the X and Z axes. [X] [Z]

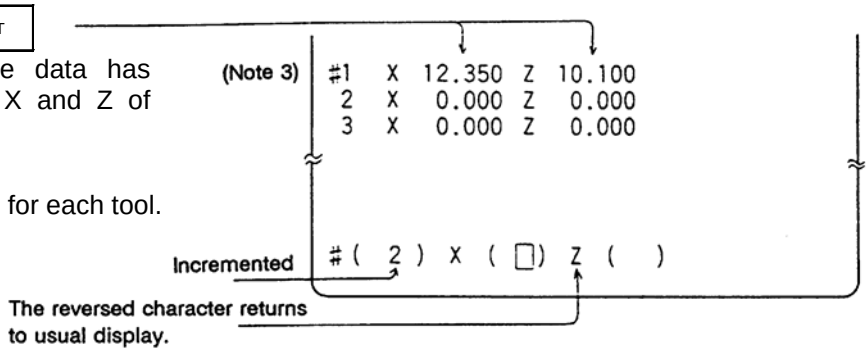


(5) The data is automatically calculated and written.
(The data is written for the axis shown in highlighted characters.)

[INPUT]

Confirm that the data has been written to X and Z of tool No. "1".

Repeat the above steps for each tool.



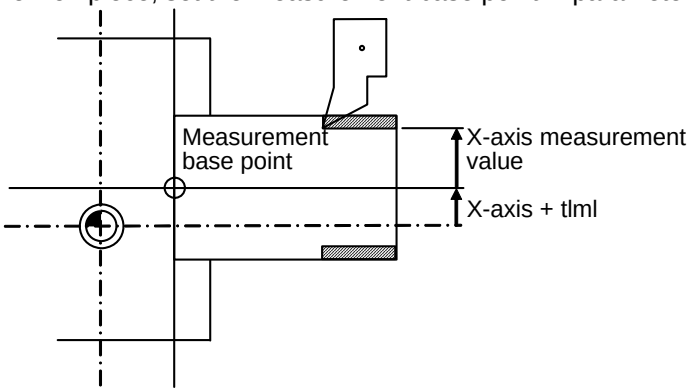
(Note 1) If the screen is changed back to the TOOL DATA screen after axis selection (after the characters are highlighted), the selection is invalidated (the characters are not highlighted).

(Note 2) If an axis having an error (reference point return incomplete axis, etc.) is selected, the characters will not be highlighted. An error message will appear.

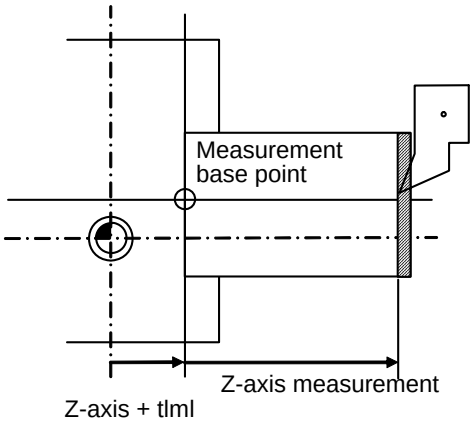
(Note 3) For a diameter command, the diameter value is written.
For a radius command, the radius value is written.

(3) Measurement value input method

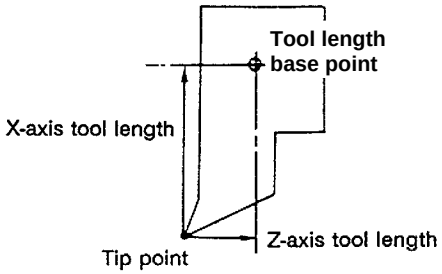
Set the type selection to the measurement value input method. (Set "#1102 tlm" to 1).
To carry out the measurement value input method, a workpiece for measuring is required. To measure the workpiece, set the measurement base point in parameter "#2015 tlml–" beforehand.



(Note) Always set the measurement point with the radius, regardless of the diameter/radius command.
Set the measurement point in the machine coordinate system.



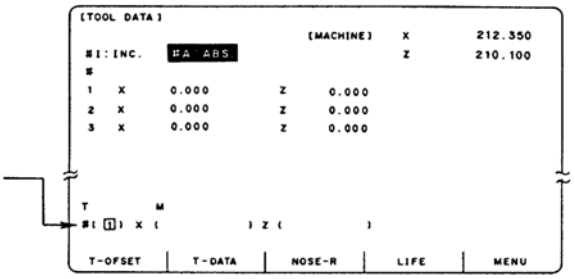
Tool length = Machine value - Measurement base point (tlml) - Measurement value
The expression above is used for automatic calculation in the measurement value input method.



<Measuring procedure for the measuring value input method>

(1) Select the TOOL DATA Screen.
(2) Set the tool No. to be measured in # ( ).
(Select the tool before this step.
It can be selected using a manual numerical command.)

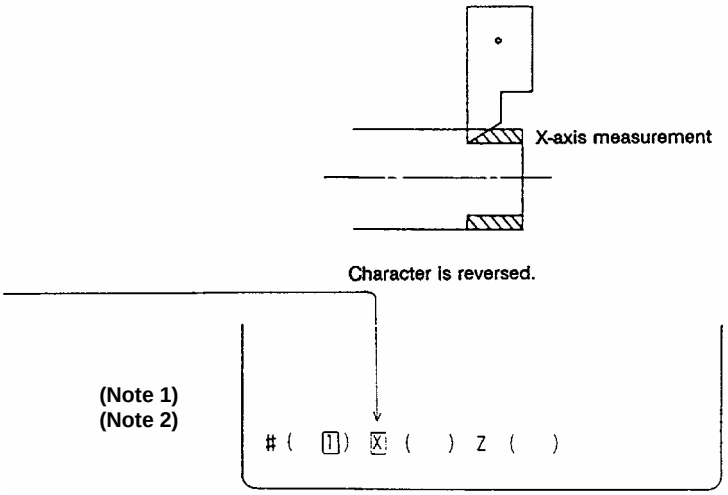
(Example) Select tool length No."1".



(3) Cut the surface corresponding to the axis to be measured.
To measure the X axis, cut the work lengthwise.
(For the Z axis, execute face turning.)
(4) Do not retract the tool at the finish point of the cutting, but press address key of the axis to be measured.

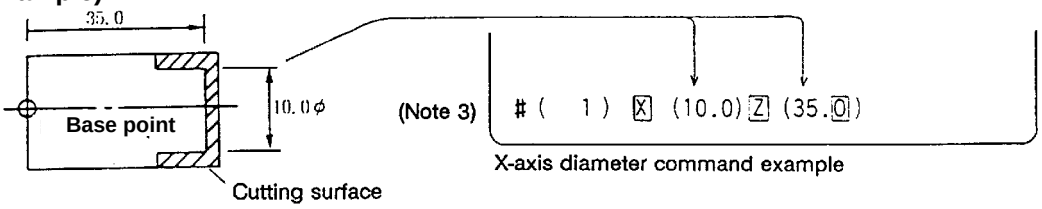
X axis measurement ... [X]

In this way the machine coordinate values of the measured axis are stored in the memory. They are canceled by pressing the same key twice.
∗Also repeat steps (3) and (4) for the Z axis.

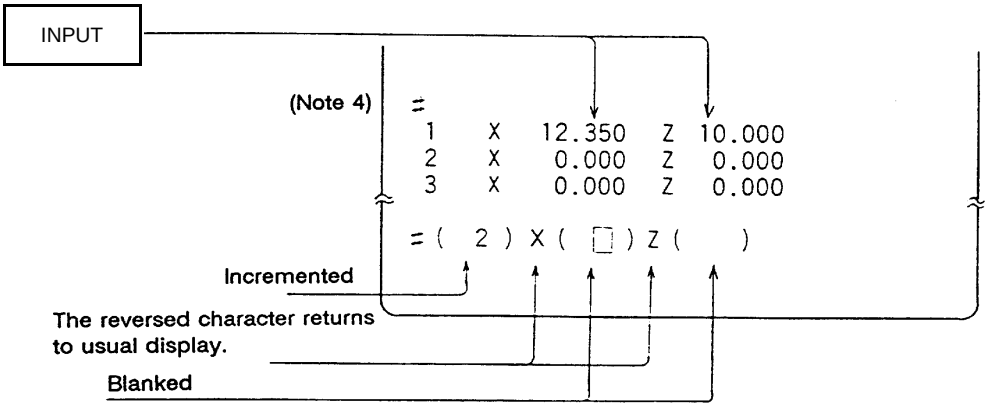


(5) Retract the tool, and stop the spindle.
(6) Measure the workpiece, and set the measurement values in the setting areas of each axis. Set the values for all axes shown in highlighted characters.

(Example)



(7) The data is automatically calculated and written.
(The data is written for the axis shown in highlighted characters.)



Repeat the above steps for each tool.

(Note 1) If the screen is changed back to the TOOL DATA screen after the characters are highlighted, the characters will return to the usual display. Retry processing, beginning with step (3) or (4).

(Note 2) If an axis having an error (reference point return incomplete axis, etc.) is selected, the characters will not be highlighted. An error message will appear.

(Note 3) For a diameter command, the diameter value is written.
For a radius command, the radius value is written.

(Note 4) An error occurs in the following cases:

- # (1) [X] ( ) [Z] (35.0) ... The X axis measurement value was not set.
- # (1)X (10.0) [Z] (35.0) ... The character was not highlighted although the X axis measurement value was set.

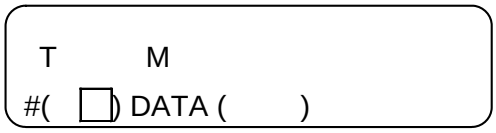
In these cases the status is held, so reset correctly and then repress [INPUT] .

2.3.2.2 Manual Numerical Command Operation on the TOOL DATA Screen (M, T)

When carrying out a manual numerical command of the TOOL OFFSET screen, the mode must first be changed from the normal data setting mode to the manual numerical command mode. M and T commands can be executed by screen operation in this mode.

(1) Changing from the normal data setting mode to the manual numerical command mode
A cursor appears in the data setting area in the normal data setting mode, but a cursor does not appear in the manual numerical command mode. Confirm that the mode has changed over by checking this difference.

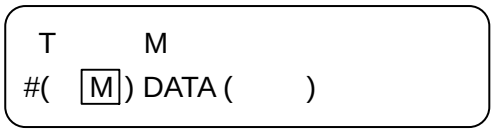
<Normal data setting mode>



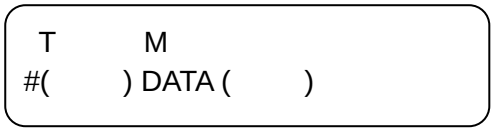
<Manual numerical command mode>

The operation is as follows:

(a) Set "M"(manual) in the first set of parentheses in the setting area.
This operation is the same for M or T commands.



(b) Press the [INPUT] key. The mode changes to the manual numerical command mode.



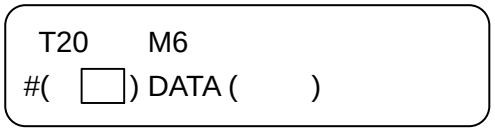
The data in the setting area is cleared, and the cursor disappears from the screen.

(2) Executing the manual numerical command ..... Carry out this step after (1) above.
1. Press the address key corresponding to the command. The display area of the corresponding command value is highlighted, and a manual numerical command input status results. Execute tool function commands with "T", and miscellaneous function commands with "M".
2. Key-input the numerical value to be commanded.
3. Press the [INPUT] key. The command is executed.

(Note) The manual numerical command operation is the same as the operation on the POSITION screen. Refer to the section on manual numerical commands for the MONITOR and POSITION screens for details.

(3) Operation for returning the mode from the manual numerical command mode to the normal data setting mode

Press the [↓] key. The normal data setting mode returns.



The cursor appears in the first set of parentheses, and the normal setting mode is enabled.

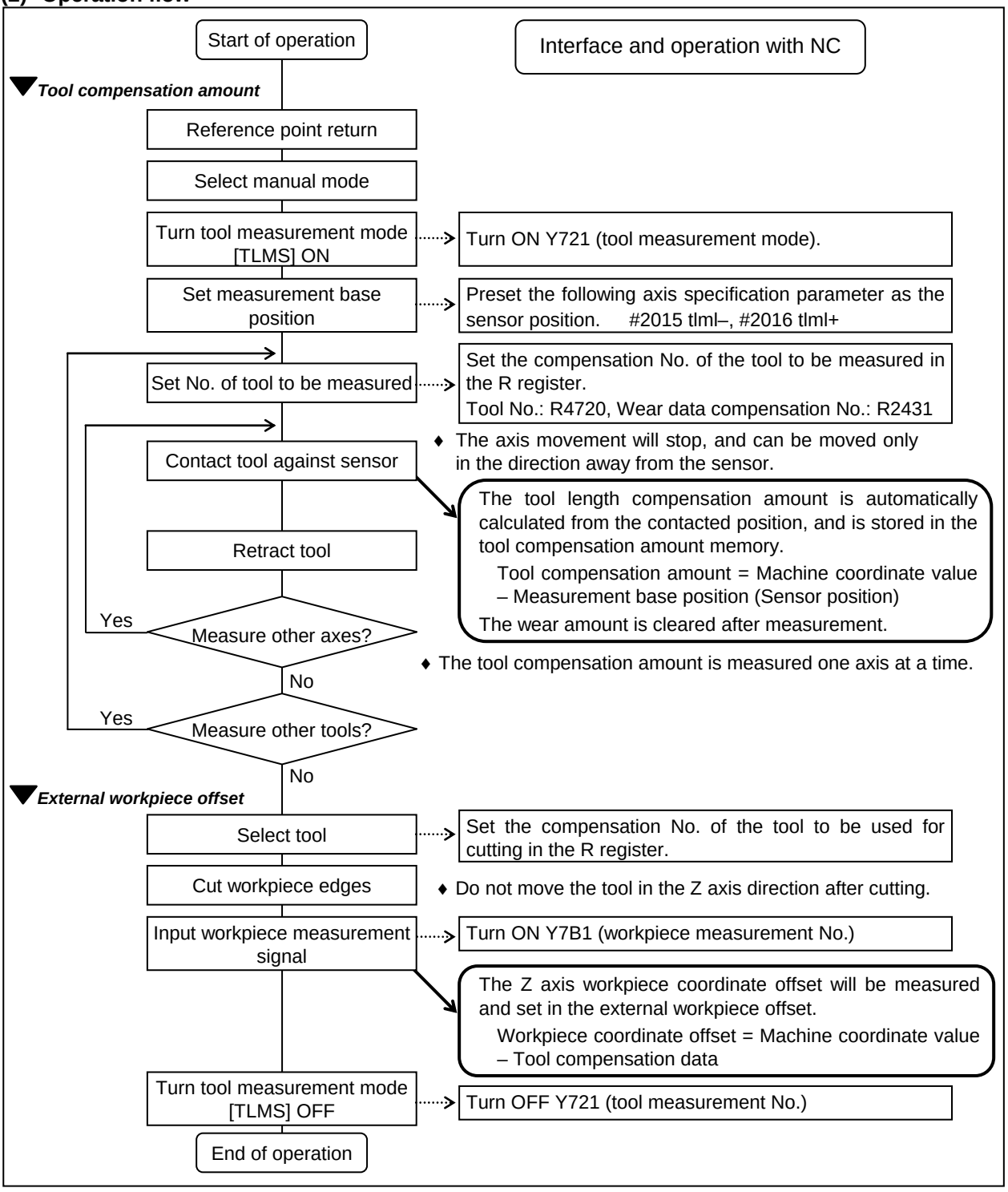
2.3.2.3 Tool Presetter

(1) Outline

By using a device having a touch sensor, the tool compensation amount can be calculated just by contacting the tool nose against the touch sensor with manual feed. The calculated results are stored in the tool compensation amount memory.

After setting the tool compensation amount for each tool, the Z axis external workpiece coordinate offset data can be set by cutting the edges of the workpiece with manual operation and inputting the workpiece measurement signal.

(2) Operation flow



(3) Explanation of operations

(a) Setting the tool compensation amount

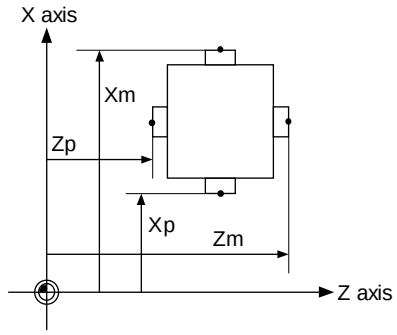
1) Zero point return
After turning the power ON, establish the coordinate system by carrying out dog-type zero point return. When using the absolute position detection specifications, carry out initialization if the absolute position is not established.

2) Select the mode
Set the mode selection switch to the manual mode (either [handle], [jog] or [rapid traverse]).

3) Input the tool measurement mode signal
Set the tool measurement mode signal to "1".
The tool measurement mode is entered with steps 1), 2) and 3).

4) Confirm measurement base position (sensor position)
The following parameter must be set before carrying out tool setter operations.

#2015 tlml–, #2016 tlml+ (sensor position) Axis specification parameter p. 2



Xm : X axis – sensor machine coordinate value (position measured by moving in – direction)
→ #2015 tlml– X axis
Zm : Z axis – sensor machine coordinate value (position measured by moving in – direction)
→ #2015 tlml– Z axis
Xp : X axis + sensor machine coordinate value (position measured by moving in + direction)
→ #2016 tlml+ X axis
Zp : Z axis + sensor machine coordinate value (position measured by moving in + direction)
→ #2016 tlml+ Z axis

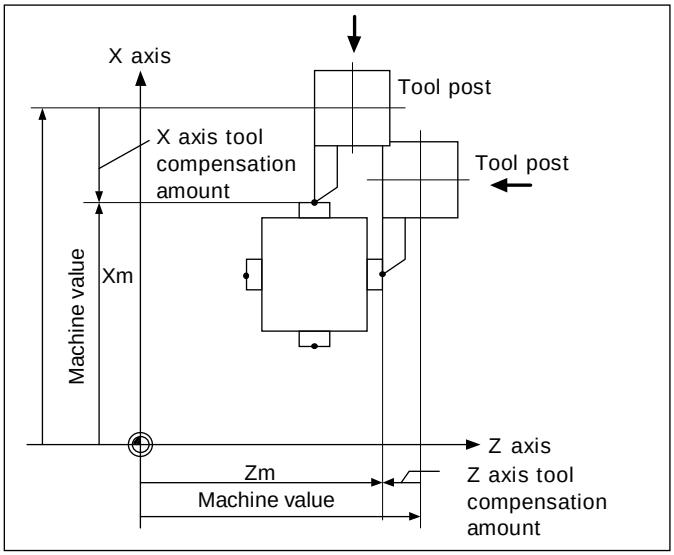
5) Select the tool
Select the tool to be measured.
Set the compensation No. of the tool to be selected as a BCD code in R4720.
Set the compensation No. of the wear data to be cleared after measurement as a BCD code in R2431.
(The tool No. data is input from the PLC to the NC.)

6) Measure tool compensation amount with sensor contact
Approach the tool nose to the sensor with manual or handle feed.
Stop the feed when the nose contacts the sensor.
The tool length compensation amount will be automatically calculated from the contacted position, and will be stored in the tool length memory.
After measuring, the wear amount of the designated compensation No. will be cleared.

(Note) The sensor contact surface is judged by the NC according to the manual axis movement direction, so measure the tool compensation amount one axis at a time.

<Details of automatic calculation expression>
The tool compensation amount is automatically calculated with the following expression.

Tool compensation amount = Machine coordinate value – Measurement base position (sensor position)



Tool compensation amount calculation diagram

7) Retract the tool
8) Set the tool compensation amount for the X axis and Z axis using steps 5) to 7).
9) Repeat steps 5) to 8) for the required tools.
10) Turn the tool measurement mode signal OFF.

This completes the measurement of the tool compensation amount.

(b) Setting the external workpiece coordinate offset data

1) Reference point return
After turning the power ON, establish the coordinate system by carrying out dog-type reference point return.
When using the absolute position detection specifications, carry out zero point initialization if the absolute position is not established.

2) Select the mode
Set the mode selection switch to the manual mode (either [handle], [jog] or [rapid traverse]).

3) Input the tool measurement mode signal
Set the tool measurement mode signal to "1".
The tool measurement mode is entered with steps 1), 2) and 3).

4) Select the tool
Issue the T command with MDI operation, etc., and select the tool.
(Note 1) Set the compensation No. of the tool to be selected in the R register (R register corresponding to the compensation No.).
(Note 2) Preset the tool length data and wear data for the tool to be used.

5) Cut workpiece edges
If the workpiece edges have not been cut, cut them slightly to flatten the workpiece edges.
(Note 1) Do not move the tool in the Z axis direction after cutting the workpiece edges.
(Note 2) If the edges do not need to be cut, position to the measurement position.

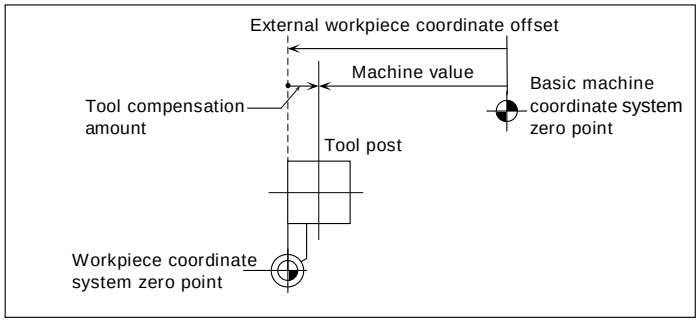
6) Set the Z axis external workpiece offset data with the workpiece measurement signal input
Turn ON the workpiece measurement signal. The Z axis external workpiece coordinate offset data will be automatically calculated from the machine value at the time the signal is turned ON and the tool compensation data of the tool used. The data will then be set.

(i) Details of automatic calculation expression
The external workpiece coordinate offset data is automatically calculated with the following expression.
(Refer to “External workpiece coordinate offset calculation diagram”)

External workpiece coordinate offset = Machine coordinate value – Tool compensation data

The tool compensation data used for the measurement is selected with the base specification parameter "#1226 aux10 bit0".

aux10 bit0	Tool compensation data
0	Tool length data + nose wear data
1	Tool length data



External workpiece coordinate offset calculation diagram

(ii) Selected tool's compensation No.
The number set in the R registers, shown in the table below, are used as the tool length and nose wear data compensation numbers for automatic calculation.

Compensation No. R registers

#1098 Tlno.	#1130 set_t	#1218 aux02 bit4	Tool length compensation No.	Nose wear compensation No.
0	0	0/1	R2446, R2447	R2446, R2447
	1	0/1		
1	0	0	R120, R121	R2446, R2447
		1	R2448, R2449	R2446, R2447
	1	0/1	R2448, R2449	R2446, R2447

(Note 1) If the compensation No. is 0, the compensation amount will be calculated as "0".

(Note 2) If the compensation No. exceeds the number of offset sets in the specifications, the "E76 TOOL No. ERROR" error will occur.

(Note 3) The details of the parameters are shown below.

#	Items		Details
1098	Tlno.	Tool length offset number	0: Tool length offset No. = Nose wear compensation No. 1: Tool length offset No. = Tool No.
1130	set_t	Display selected tool number	0: NC command 1: Command by user PLC
1218	aux02 (bit4)	Tool number selection	0: Conforms to "#1130 set_t". 1: Uses the tool number commanded by user PLC

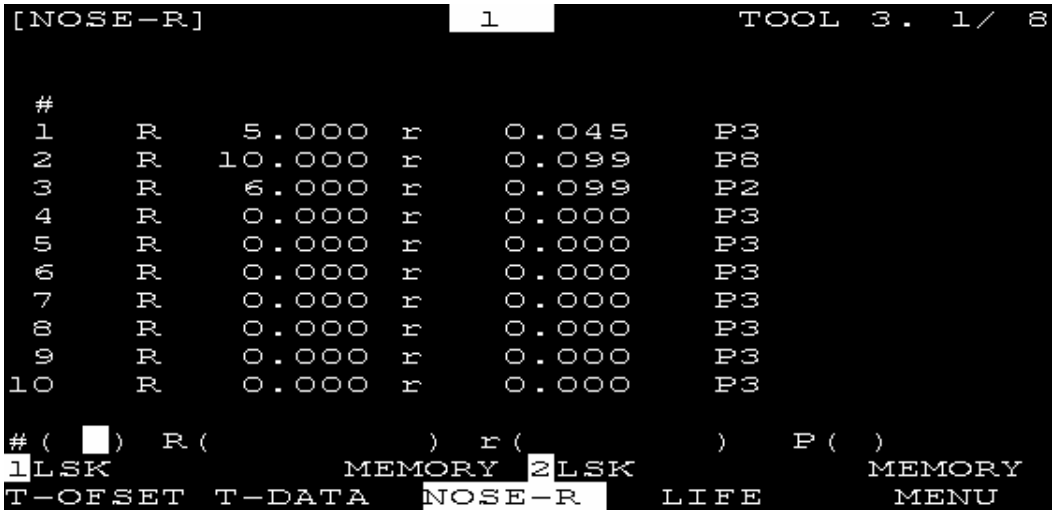
7) Turn the tool measurement mode signal OFF.
This completes the measurement of the external workpiece coordinate offset.
When carrying out this operation independently, follow steps 1) to 7), and when carrying out after measuring the tool compensation amount, carry out steps 4) to 6) between 9) and 10) of “(a) Setting the tool compensation amount”.

(4) Precautions

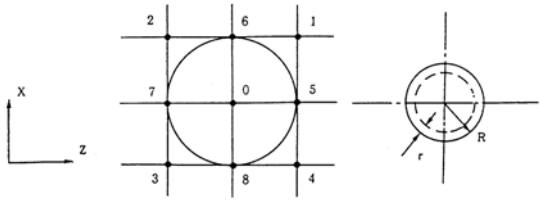
1) When entering the sensor area, the axis can move only in one direction selected from +X, –X, +Z, –Z, (+Y, –Y).
If two axes (ex. +X, –Z) are moved simultaneously, it will not be clear which contact surface was contacted, so the measurement will not be made. Note that the error "E78 AX UNMATCH (TLM)" will occur and the movement will stop for safety purposes.
2) After entering the sensor area, if the nose is contacting the sensor, the axis can be moved only in the direction away from the sensor. (An interlock is applied on the entry direction by the NC.)
The axis can move in both directions when the nose is separated from the sensor.

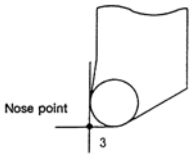
2.3.3 Tool Nose Data

The NOSE-R screen will appear when the menu [NOSE-R] is pressed.



Set the tool nose radius R (nose R), wear r, and tool nose point for each tool used. When the tool nose R compensation (G41, G42, G46) command is given, the tool nose is assumed to be a half-circular arc with radius R (R + r) corresponding to the tool No. Compensation is then carried out so that the half-circular arc contacts the designated machining program path.



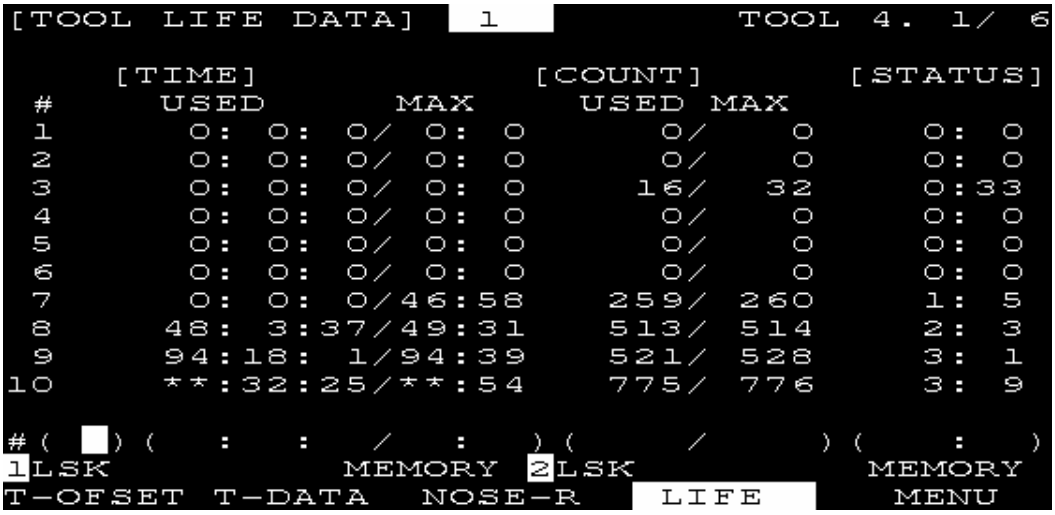


Data	Function
P0 to P8	Tool nose point
R	Tool radius (nose R) (no sign)
r	Wear (no sign)

(Note) The incremental value/absolute value setting mode changeover follows the tool length data setting mode for R, and the tool wear data setting mode for r.

2.3.4 Tool Life Management I ("#1096 T_L type" is 1)

The TOOL LIFE DATA screen will appear when the menu [LIFE] is pressed.



Tool life management is valid when parameter "#1103 T_Life" is set to 1. Tool life management is then carried out according to the tool usage time or the No. of times the tool is used (also called count). When the tool usage time reaches the service life time, or when the tool count exceeds the service life count, a tool life expiration signal (X62E) is output to the user PLC and the tool No. (#) is highlighted on the TOOL LIFE DATA screen.
Tool life management is possible for up to 80 tools (tool Nos. 1 to 80).
This function is useful for setting tool abrasion and wear data, and for knowing when to replace tools with new ones, etc.

Item		Details	Setting range
TIME	USED	The cumulative time the tool is used. This timer value is incremented during cutting.	0 : 0 to 99 : 59 (h: min)
	MAX	The tool service lifetime setting. Set the max. time the tool can be used. Seconds are discarded.	0 : 0 to 99 : 59 (h: min) (0 : 0 = no warning given)
COUNT	USED	The cumulative count the tool is used. The counter value is incremented each time the tool is used.	0 to 9999 (times)
	MAX	The tool service life count. Set the max. count the tool can be used.	0 to 9999 (times) (0 : 0 = no warning given)
STATUS	Left side	The tool life management status is indicated. 0: Not used 1: Current tool (tool being used) 2: Service lifetime (service life count) is exceeded.	0 to 2
	Right side	(Machine maker free area)	0 to 99

2.3.4.1 Tool Life Management Method

By setting the service lifetime (or service life count) to "0" for each tool, the following four tool life management methods can be selected.

Life management method	Service lifetime setting	Service life count setting
1. Time only		Set to 0.
2. Count only	Set to 0.	
3. Time and count		
4. No management	Set to 0.	Set to 0.

(1) Tool life management by time

The cutting time (G01, G02, G33, etc.) after a tool selection (T) command is carried out is incremented to the usage time corresponding to the commanded tool.
If the usage time reaches the service lifetime when a tool selection command is executed, a warning is output to the user PLC.
When the usage time reaches the service lifetime, the corresponding tool No. (#) on the TOOL LIFE DATA screen is highlighted.

(2) Tool life management by count

The count for the commanded tool is incremented when the first cutting feed starts after a tool selection (T) command is carried out. If no cutting feed is executed after the selection of a tool, the count is not incremented.
If the count equals the service life count for the commanded tool when a tool selection command is executed, a warning is output to the user PLC.
When the count exceeds the service life count (when the cutting feed starts after a tool selection command), the corresponding tool No. (#) on the TOOL LIFE DATA screen is highlighted.

(3) Tool life management by time and count

The tool life is managed simultaneously by time and count.
If the usage time reaches the service lifetime, or the count equals the service life count for the commanded tool when a tool selection command is executed, a warning is output to the user PLC.
When the usage time display reaches the service lifetime display, or when the count display exceeds the service life count display, the corresponding tool No. (#) on the TOOL LIFE DATA screen is highlighted.

(4) No management

The usage time and count are incremented, but no alarm is output to the user PLC, and the tool No. (#) on the TOOL LIFE DATA screen is not highlighted.

2.3.4.2 Conditions for Counting (incrementing)

The usage time (or count) is incremented when a cutting feed (G01, G02, G03, G33) is executed. Note that they are not incremented in the following conditions:

- When the base specifications parameter "#1103 T_Life" is OFF.
- During machine lock
- During miscellaneous function lock (input signal from the PLC)
- During dry run
- During single block operation
- When the count ON signal of the data used is OFF. (Input signal from the PLC)

2.3.4.3 Setting Tool Life Management Data

(1) To set tool life management data, set the tool No. in # ( ). Then set the tool service lifetime and service life count data in the corresponding setting areas, and press the [INPUT] key.
(2) The operations in (1) update the tool life management data display, increment the tool No. in # ( ) by 1, and deletes the service lifetime (life count) data in ( ).
(3) If a tool No. and tool life management data is set for a tool No. other than the ones displayed, the screen will change to one corresponding to the set tool No. when the [INPUT] key is pressed once. The tool life management data can be set by pressing the [INPUT] twice.
(4) The tool No. that appears in # ( ) can be continually incremented or decremented by pressing the [↑] and [↓] keys.

2.3.4.4 Erasing Tool Life Management Data in Display Screen Units

Ten sets of tool life management data appear in one screen. All the displayed tool life management data (time-used, time-max, count-used, count-max) can be set to 0 by pressing the [INPUT] key, and then pressing the [SHIFT] [C.B CAN] key and [INPUT] key.

2.3.4.5 Cautions

(1) The cumulative time (count) is incremented, even if the service lifetime (service count) is set to "0". Note that a warning (tool life expiration signal: X62E) is not output.
(2) For tool life management by time, a warning will not be output to the PLC if the usage time reaches the service lifetime during cutting. Instead, the warning will be output when the next tool selection command is issued. During that interval, the usage time will continue to increment.
(3) When there are 20 offset pairs, the No. of tools whose lifetime can be managed is 20.
(4) The TOOL LIFE screen cannot be selected in systems without the tool life management function. If the tool life management menu key is pressed, alarm "E06 NO SPEC" will occur and the screen will not change.
(5) If a tool selection (T) command is carried out during cutting feed modal, the count will be incremented at that time.

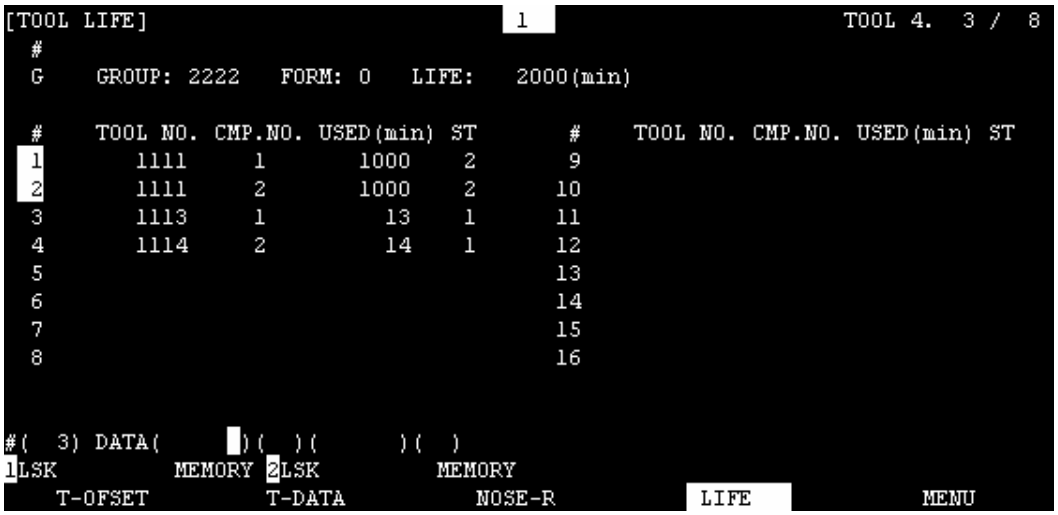
2.3.5 Tool Life Management II ("#1096 T_Ltype" is 2)

The tools used are classified into several groups. With this tool life management with spare tool function, tool life (usage time, count) is managed for each group. When a tool's life is reached, an equivalent spare tool is selected in order from the group to which that tool belongs.

(1) No. of tool life management tools : 1-part system: max. 80 tools, 2-part system: max. 40 tools/part system
(2) No. of groups : 1-part system: max. 80 tools, 2-part system: max. 40 tools/part system
(3) Group No. : 1 to 9999
(4) No. of tools per group : Max. 16 tools
(5) Service lifetime : 0 to 999999 min. (approx. 1667 hours)
(6) Service life count : 0 to 999999 times

2.3.5.1 Group Registration

(1) Set and display a group's life management information on page 1 of TOOL LIFE screen.



(Note) The (MIN) display following "LIFE" or "USED" will change according the method setting.
FORM 0: Time (MIN): Indicates that the data is displayed in minute units.
1: Count(SET): Indicates that the data is displayed in count units.

1) Selecting a display group
Select the group by setting # (G) DATA (group No.).
When the group No. is set, the tool life management information of the tools registered in that group will appear from #1 to #16. A highlighted # No. indicates that tool is a life-reached tool (or a skip tool).
To display another group, set # (G) DATA (group No.) again.

2) Registering a group
Register a group by setting # (G) DATA (group No. to be registered) (FORM) (LIFE).

- Designate a group No. from 1 to 9999.
- Set FORM with for group life management by either time or count.
  0: Time 1: Count

If the FORM setting is omitted, the method becomes "0" (time).
- Set LIFE with the service life setting value for that group's tools. (0 to 999999).

If the LIFE setting is omitted, the life setting value becomes "0".

(Note 1) The FORM and LIFE setting values can only be changed for a group being displayed. This is to prevent mistaken settings. The setting is made with # (G) DATA ( ) (FORM) (LIFE).
(Only FORM and LIFE setting values can be changed.)

(Note 2) FORM and LIFE data is common data within that group. To suppress the LIFE value of a specific tool, adjust by setting the offset value for the USED data. In this case, ST will be set to 1 (current tool), and the following new tool selection signal will not be output at tool selection.

(Note 3) The USED data will be incremented when the LIFE data is 0, but no judgment will be made when the service life count is reached.

3) Deleting a group registration

The group being displayed and its data can be deleted by pressing [SHIFT] + [C.B CAN] + [INPUT] .

4) Registering tools

Set the tools in order from the first tool to be used. If multiple compensation Nos. are used with one tool, set the tool No. and respective compensation Nos. for each compensation No.

- Tool No. : Set the tool No. (1 to 999999: differs according to the specifications)
- Compensation No. : Set the compensation No. (1 to 80: differs according to the specifications)
- USED : When the designated tool is other than a not-used tool, the initial incrementation value can be adjusted by setting the USED data. If no data is set, this value becomes 0. (Can be omitted.)
- ST : Designate whether the tool is a tool skip tool or not. (Can be omitted.) If the data is not set, or if 0 to 2 is set, the data will be automatically set according to the relation with the USED data and LIFE data.
  0: Not used tool 1: Current tool (tool being used)
  2: Normal life-reached tool 3: Tool skip tool

(Example) Setting to use multiple compensation Nos. with one tool.

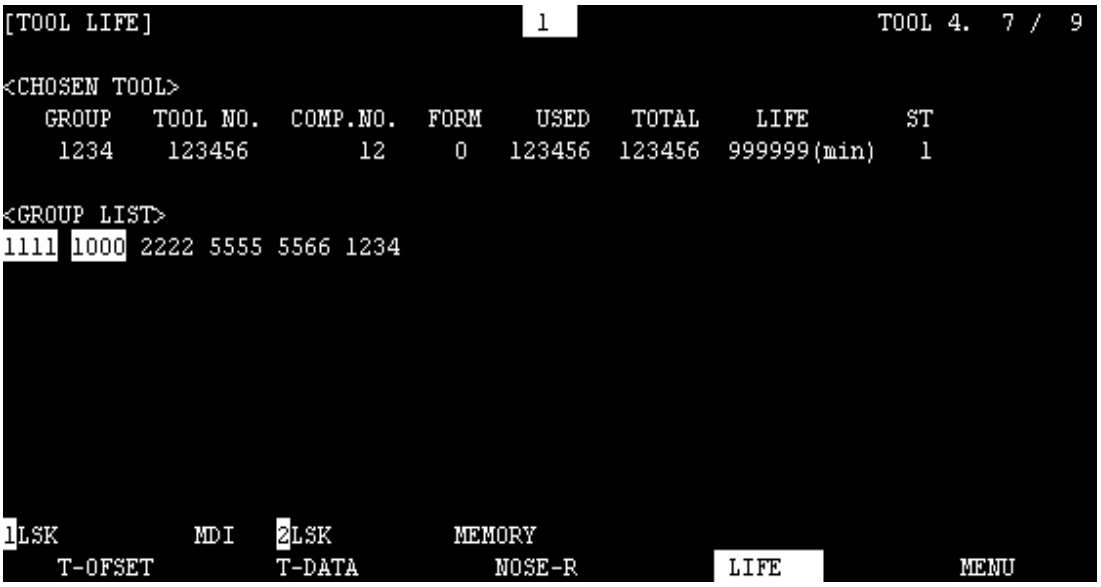
#	Tool No.	Compensation No.	
1	520000	11	..... Equivalent to a T52000011 command.
2	520000	12	..... Equivalent to a T52000012 command.
3	520000	13	..... Equivalent to a T52000013 command.

(Note) Tool life management is carried out in group units with this function. Thus, if a tool is set in a different group, the life will be managed according to the respective group, and that tool cannot be managed correctly.

5) Deleting a tool registration

Set 0 in the tool No. of the # No. to be deleted. All data of that # No. will be deleted, and the subsequent # Nos. and data will all move up a line.

(2) After the group's life management information is displayed, the life management data of the tool currently being used and the list of registered groups of tools are displayed. This page is mainly used for monitoring tool life data in group units.



1) Display details

< CHOSEN TOOL >: The life management information of the tool currently being used appears here.

FORM : The incrementation unit of the life data appears here.
0: Time 1: Count

ST : The tool status appears here.
0: Not used tool 1: Current tool (tool being used)
2: Normal life-reached tool 3: Tool skip tool

TOTAL : For tools using multiple compensation Nos., the total of the usage data for each compensation appears here. If there is only one compensation No., the data will be the same as "USED".

< GROUP LIST > : All registered group Nos. appear here.
A highlighted group No. indicates that the lives of all tools registered in that group have been reached.

2) Erasing all registered data of a group

All registered data (including the group No.) of a group can be erased by pressing [SHIFT] + [C.B CAN] + [INPUT] .

2.3.5.2 Tool Life Incrementation Methods

The tool life can be incremented either by time method or by the No. of uses (count) method.
If the USED data equals or exceeds the LIFE data as a result of incrementation, a spare tool will be selected from that tool's group by the next relevant group selection command (T****99). After that, the incrementation will be for the newly selected tool (the spare tool selected). If the life of all tools in a group is reached, and a spare tool cannot be selected, the incrementation will continue for the last tool selected.

(1) Time incrementation with the time method

The time the tool is used in the cutting mode (G01, G02, G03, G31, G33, etc.) is incremented in 100ms units.
The time is not incremented during dwell, machine lock, miscellaneous function lock, dry run or single block status.

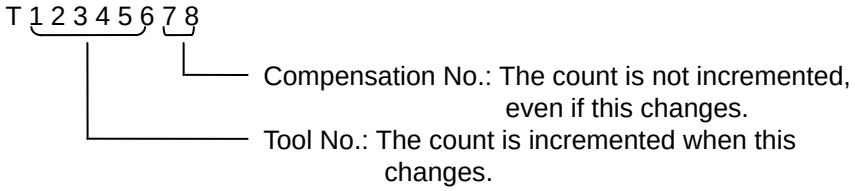
(Note) • The max. life value is 999999 min.
• The data on the TOOL LIFE screen is displayed in minute units.

(2) No. of uses (count) incrementation with the count method

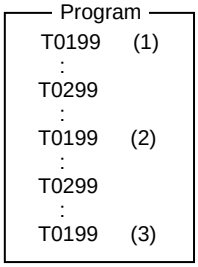
Incrementation is carried out when the No. of the tool being used changes by the execution of a tool selection command (T****99) during the cutting mode (except during machine lock, miscellaneous function lock, dry run, and single block states). (If the mode never changes to the cutting mode after the tool No. changes, the count is not incremented.)

(Note) • The max. life value is 999999 times.
• If only the compensation No. for the current tool changes, the count is not incremented.

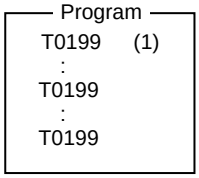
If the T code of the current tool is 12345678:



<< Operation example >>



The count for group 01 is 3 times.



The count for group 01 is 1 time.

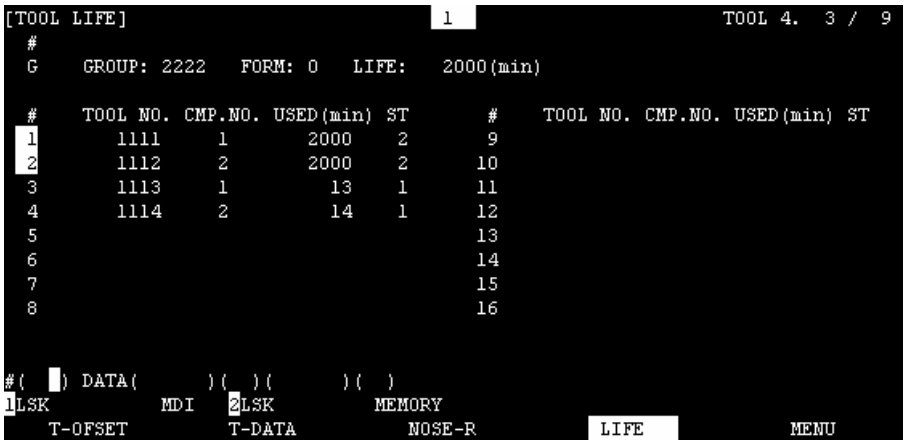
(Note) The count is for one program execution.
If the program is executed again after resetting the count will be incremented.

(3) Incrementation when using one tool with multiple compensation Nos.

With this function, each registered T No. (tool No. + compensation No.) has independent USED data, so the count for a tool using multiple compensation Nos. is incremented for each compensation No. Thus, life management for that tool's USED data is carried out with the total of the USED data for each compensation. Because of this, when only one # No. is looked at on the screen, the tool status (ST) may be 2 (life-reached tool), although that tool's USED data has not yet reached the life of the tool. The total of the currently selected tool's USED data appears in "TOTAL" of the <CHOSEN TOOL> column on page where the life management data of the tool currently used is displayed.

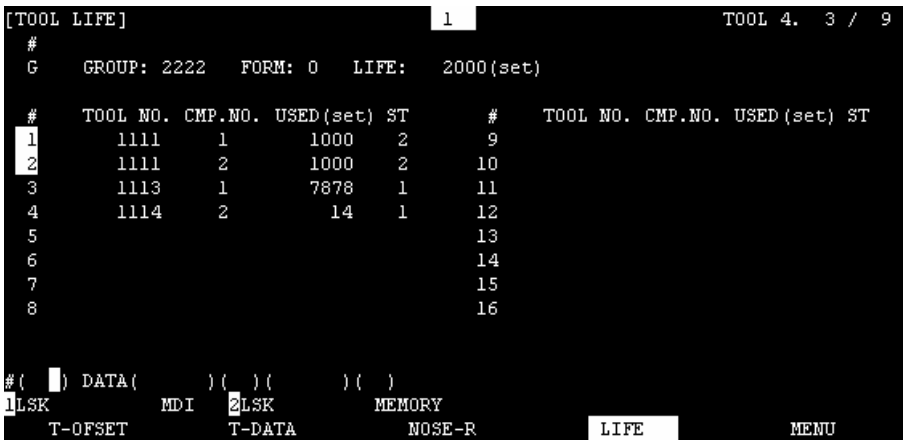
Example of the screen display when using multiple compensation Nos.

• Time method (life: 2000 min.)



* The life of tool 1111 is the total usage time of #1 to #2.

• Count method (life: 2000 times)



* The life of tool 1111 is the total count of #1 to #2.

2.3.5.3 Parameters

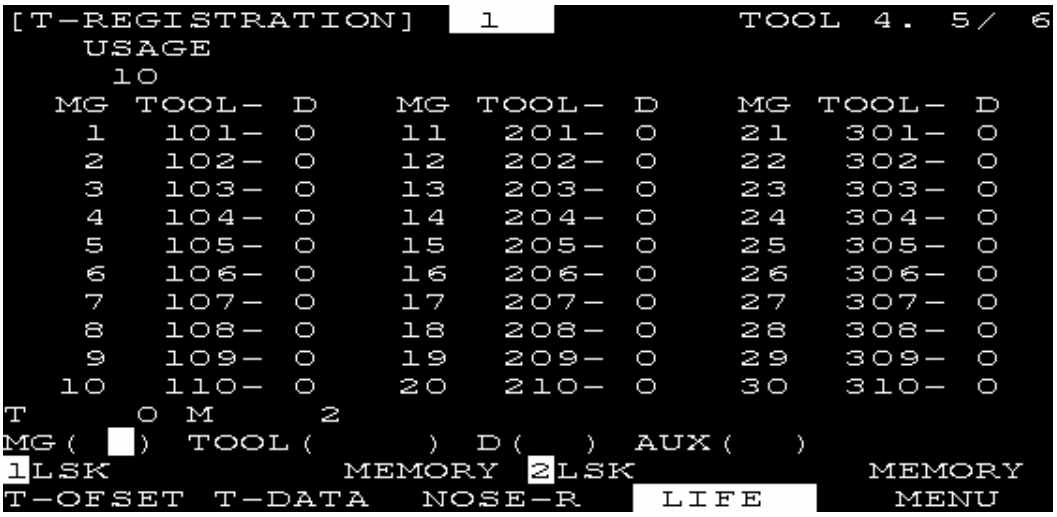
The tool life management specifications will differ according to parameter "#1096 T_Ltype" and "#1106 Tcount". Confirm the explanation for the relevant setup parameter data item.

2.3.6 Tool Registration

The T-REGISTRATION screen will appear when the menu [LIFE] is pressed, and the screen is changed using the [NEXT] key. The use of this screen differs according to the user PLC, so refer to the instruction manual issued by the maker for details.

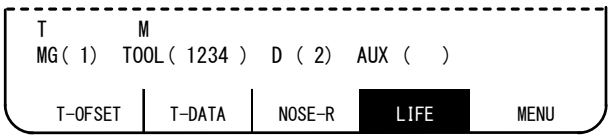
2.3.6.1 Outline of Functions

(1) Tools used can be registered in the magazine pot.
(2) When the magazine pot and the tool No. are changed by a tool selection command or a tool replacement command, the new tool No. is displayed.
(3) Random data can be set in AUX ( ) in the setting area and processed as a sequence with the user PLC.
(4) Tools can be registered in USAGE on the upper portion of the screen. The displayed name and displayed No. can be changed.
(5) The No. of tool registrations differs according to the specifications, but a max. of 80 tools can be registered, with a max. of 4 digits in the tool Nos.
(6) Tools can be selected by a manual numerical command.

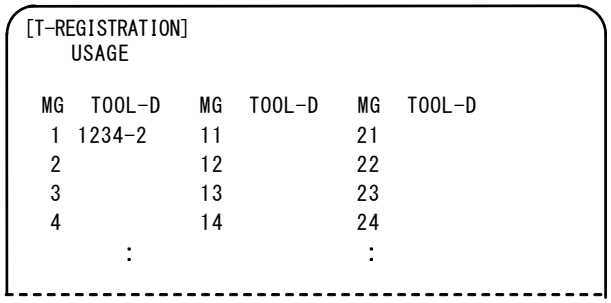


2.3.6.2 Tool Registration in the Magazine Pot

Set “1” in MG ( ),
“1234” in TOOL ( ), and
“2” in D ( ).



Press the [INPUT] key.



The tool No. and data in D appear in the designated magazine pot, and the magazine No. in MG ( ) is incremented by 1. The data in the other ( ) dis- appears.
When a No. other than the magazine No. in the data display area is set, the screen changes as follows: When the [INPUT] key is pressed the 1st time, the screen corresponding to the magazine No. appears. When the [INPUT] is pressed the 2nd time, the data set in the data area appears.

(Note) Refer to the instruction manual issued by the maker for data on the function and purpose of the data in D.

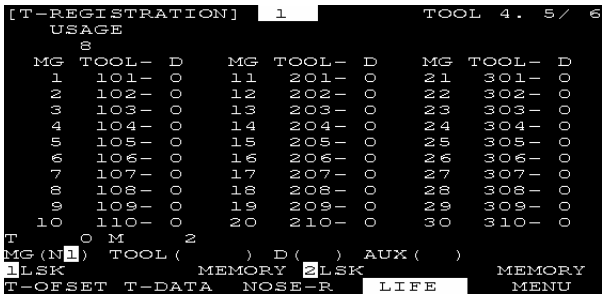
2.3.6.3 Tool Registration in the Spindle, Standby and Indexing Areas

These commands are used to change the display data when the tool No. set in the magazine pot differs with the displayed tool No.

Set to USAGE MG (N0) TOOL ( )

Set “N0” in MG ( ), and “8” in TOOL ( ).

Press the [INPUT] key.



"8" appears under USAGE in the data display area, and the display in the data setting area changes to MG (N1).

(Note) Although the title display in the upper portion of the screen differs according to the maker, the data is always set by an input of N0.

2.3.6.4 Deleting Tool Registration Data

Set CL in MG ( ).

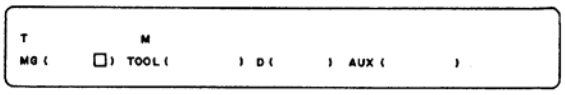
Press the [INPUT] key.	All data displayed in USAGE and MG1 to MGn is cleared to 0.

2.3.6.5 Manual Numerical Command Operation (M, T) on the TOOL REGISTRATION Screen

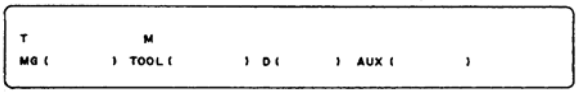
To carry out manual numerical commands on the TOOL REGISTRATION screen, the mode must first be changed from the normal data setting mode to the manual numerical command mode. M and T commands can be executed by screen operation in the manual numerical command mode.

(1) Changing from the normal data setting mode to the manual numerical command mode
A cursor appears in the data setting area in the normal data setting mode, but a cursor does not appear in the manual numerical command mode.
Confirm that the mode has changed over by checking this difference.

<Normal data setting mode>

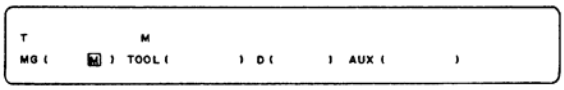


<Manual numerical command mode>

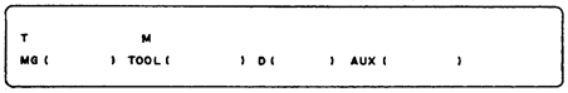


The operation is as follows.

(a) Set "M" (manual) in the first set of parentheses in the setting area.
This operation is the same for M or T commands.



(b) Press the [INPUT] key. The mode changes to the manual numerical command mode.



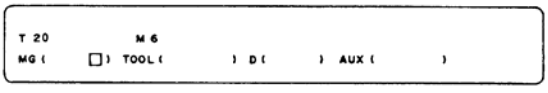
The data in the setting area is cleared, and the cursor disappears from the screen.

(2) Executing the manual numerical command ..... Carry out this step after (1) above.
a) Press the address key corresponding to the command. The display area of the corresponding command value is highlighted, and a manual numerical command input status results. Execute tool function commands with "T", and miscellaneous function commands with "M".
b) Key-input the numerical value to be commanded.
c) Press the [INPUT] key. The command is executed.

(Note) The manual numerical command operation is the same as the operation on the POSITION screen. Refer to the section on manual numerical commands for the MONITOR and POSITION screens for details.

(3) Operation for returning the mode from the manual numerical command mode to the normal data setting mode

Press the [↓] key. The normal data setting mode returns.



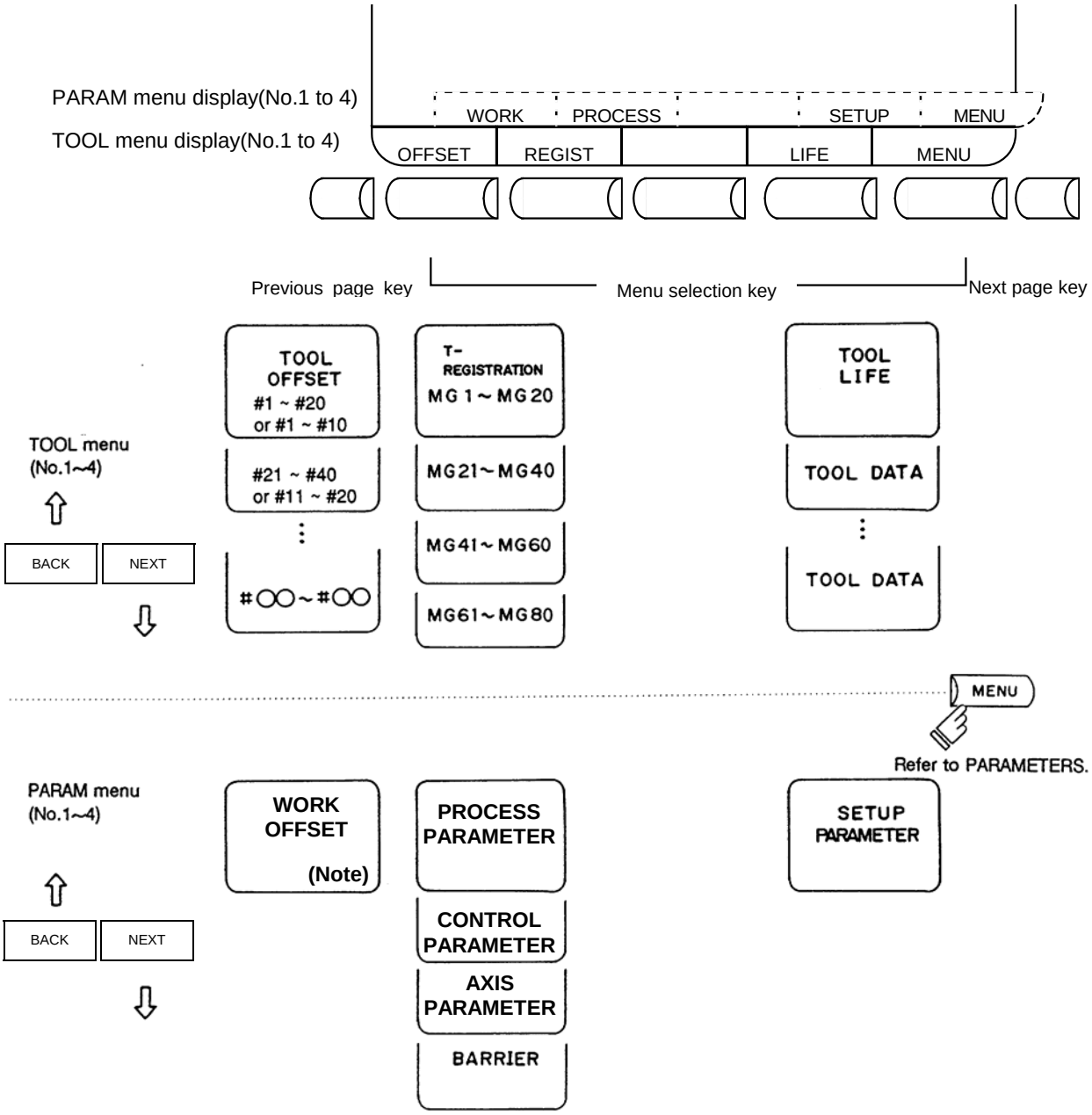
The cursor appears in the first set of parentheses, and the normal setting mode is enabled.

2.3.7 Workpiece Coordinate

Refer to "2.3.4 Workpiece Coordinate" in "2.3(II) Tool Offset (M system)".

2.3(II) Tool Offset (M system)

The following menu will display if the function selection key [TOOL PARAM] is pressed.



(Note) "WORK" is included in the TOOL menu.

CAUTION

If a tool offset amount is changed during automatic operation (including during single block stop), the new offset is validated from the command of the next block or blocks onwards.

2.3.1 Tool Offset

When the menu [OFFSET] is pressed, the TOOL OFFSET screen is displayed.

(1) Tool offset memory (type I: parameter "#1037 cmdtyp" = 1)
Form compensation memory is not distinct from abrasion compensation memory. Set the sum amount of form compensation and abrasion compensation.
Offset data is common to the tool length, tool offset, tool diameter, and cutter compensation.

(2) Tool offset memory (type II: parameter "#1037 cmdtyp" = 2)
Set the shape compensation amount and wear compensation amount separately. The form compensation amount is separated into the length dimensions and diameter dimension.
Of offset data, the length dimension data is used for tool length and the diameter dimension data is used for tool diameter cutter compensation.

Tool offset memory type I

```
[TOOL OFFSET]      1          TOOL 1. 1/ 5
#A:ABS. #I:INC.  [MACHINE]   Z -12345.678
                    SURFACE #0=    50.000
    #
     1     12.000          11    300.000
     2     50.000          12     50.000
     3    100.000          13    250.000
     4     30.000          14     50.000
     5    100.000          15    150.000
     6     60.000          16     80.000
     7     20.000          17    200.000
     8    150.000          18    150.000
     9     20.000          19    500.000
    10    150.000          20    100.000
T     0 M    66
#(   ) DATA(            )
1LSK          MEMORY 2LSK          MEMORY
OFFSET  REGIST             LIFE      MENU
```

Tool offset memory type II

```
[TOOL OFFSET]      1          TOOL 1. 1/10
#A:ABS. #I:INC.  [MACHINE]   Z -12345.678
                    SURFACE #0=    50.000
   #      LENG     WEAR      RADIUS    WEAR
    1   120.000    0.020      53.000    0.099
    2   100.000    0.004      30.000    0.000
    3   200.000    0.000      60.000    0.010
    4    20.000    0.005     150.000    0.008
    5    20.000    0.530     150.000    0.059
    6   300.000    0.032      50.000    0.111
    7   250.000    0.000      50.000    0.000
    8   150.000    0.006      80.000    0.009
    9   200.000    0.000     150.000    0.003
   10   500.000    0.667     100.000    0.888
T     0 M    66
  :          :          :          :
1LSK          MEMORY 2LSK          MEMORY
OFFSET  REGIST             LIFE      MENU
```

Tool offset data can be set in either absolute or incremental value.

Display item	Description
#A: ABS #I: INC	The valid setting mode, either absolute or incremental mode, is displayed in reverse video. Before setting data, check that the setting mode is proper.

2.3.1.1 Tool Offset Data Setting

(1) For type I
To set tool offset data, set the offset memory number in # ( ) and offset data in DATA ( ), then press the [INPUT] input key.
(2) For type II
To set tool offset data, set the offset memory number in # ( ) and offset data in the setting area corresponding to LENG, WEAR, RADIUS and WEAR, then press the [INPUT] key.
(3) If the [INPUT] key is pressed after the offset memory number and tool offset data are set, the tool offset data set in the offset memory number position is displayed, the offset memory number in the setting area # ( ) is incremented by one, and the contents of DATA ( ) disappear. At the time, the cursor moves to the right end of the same setting field as the input time.
(4) If tool offset data is set with an offset memory number not contained in the displayed offset memory numbers, the screen changes to the screen corresponding to the setup offset memory number when the [INPUT] key is first pressed. When the [INPUT] key is pressed again, the tool offset data set in the offset memory number position is displayed.
(5) The offset memory number displayed in # ( ) can be consecutively incremented or decreased by one by pressing the [↑] or [↓] key.
(6) To set the incremental mode, enter "I" in # ( ), then press the [INPUT] key.
In incremental mode, the set data is added to the data indicated in the display area. To cancel the incremental mode, enter "A" in # ( ), then press the [INPUT] key; the absolute mode is set. (Refer to "2.3.1.3 Tool Offset Data Setting Modes (Absolute and Incremental)".)

2.3.1.2 Tool Offset Data Clear

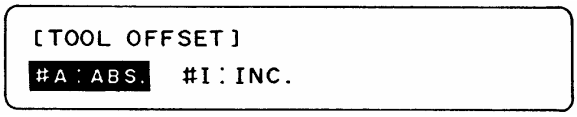
(1) Clear in display screen units
20 sets of tool offset data (10 sets for type II) are displayed on one screen. To clear all displayed offset data, press the [SHIFT] key, then press the [C.B CAN] and [INPUT] keys.

2.3.1.3 Tool Offset Data Setting Modes (Absolute and Incremental)

(1) Absolute value setting

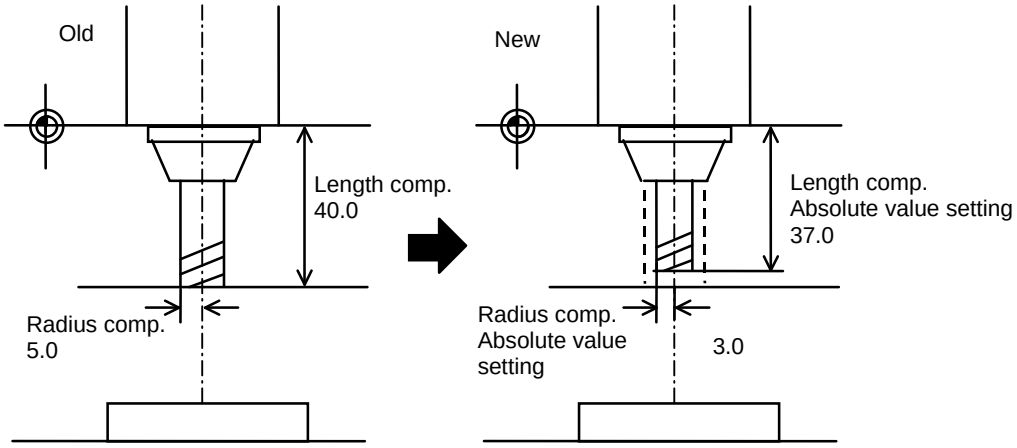
Change to the absolute value setting mode as follows:

Enter A in # ( ), then press the input key. # (A) DATA ( )



#A: ABS is displayed in reverse video indicating that the absolute value setting mode is valid.

Example of setting tool offset data in absolute mode



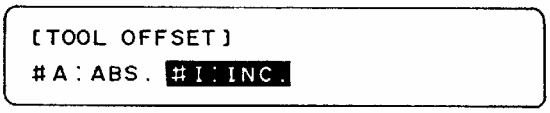
(#3 length compensation data)

Display	Setting	Display
#3 40.000	# (3) (37)	#3 37.000

(2) Incremental value setting

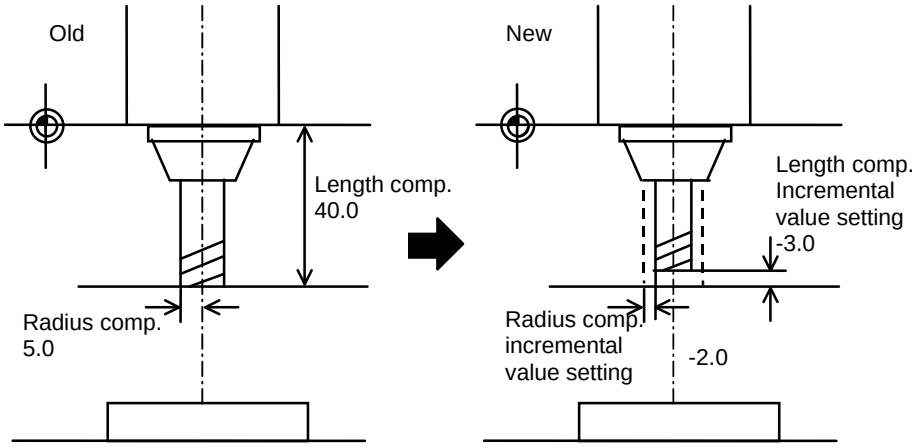
Change to the incremental setting mode as follows:

Enter I in # ( ), then press the input key. # (I) DATA ( )



#I: INC is displayed in reverse video indicating that the incremental mode is valid

Example of setting tool offset data in incremental mode



(#3 length compensation data)

Display #3 40.000	→	Setting # (3) (-3)	Display #3 37.000

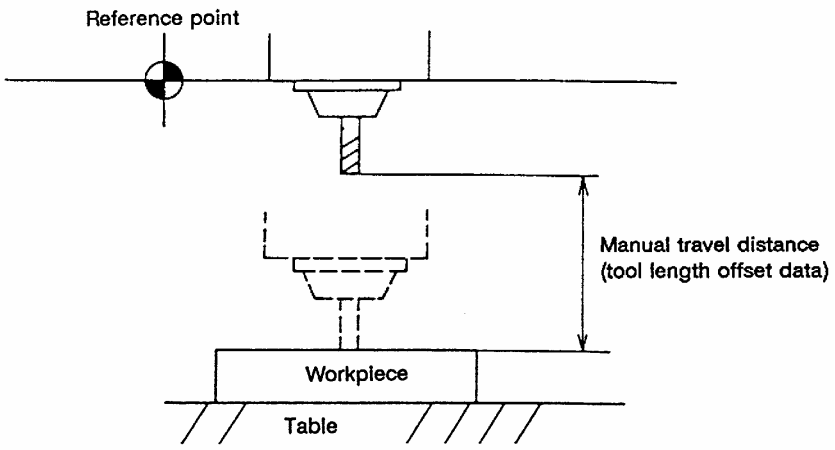
The mode thus set is retained even after the screen is changed or after power has been turned off.

2.3.1.4 Manual Tool Length Measurement

By moving a tool manually from the reference to measurement point, the travel distance from the reference to measurement point can be measured and set as tool offset.

(1) Tool length measurement I

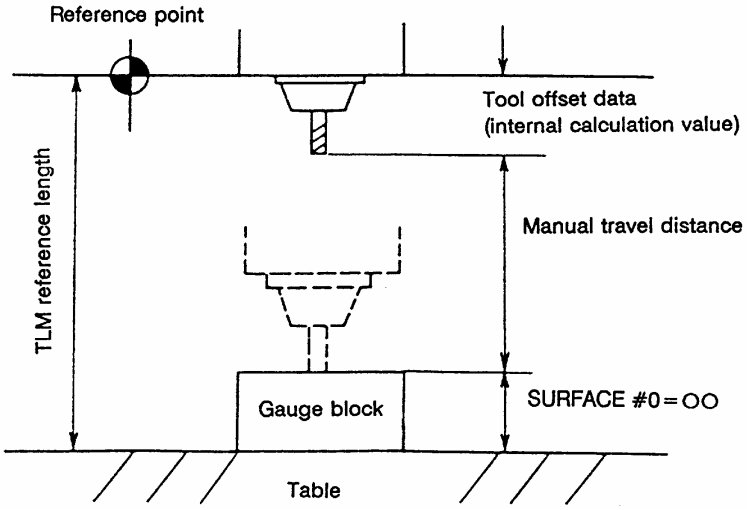
When the tool is placed in the reference point, the distance from the tool tip to measurement point (work top end) can be measured and set as tool offset data.



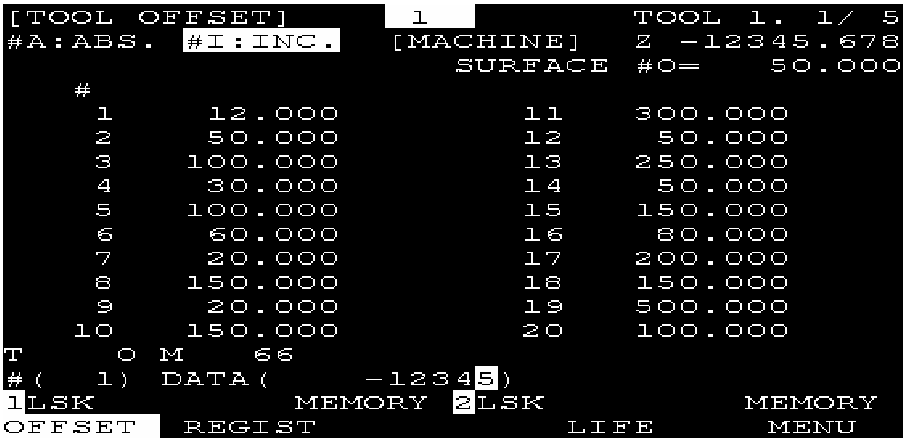
When TLM reference length (#1102 tlm) = 0 and SURFACE #0 = 0 are set, tool length measurement I mode is set.

(2) Tool length measurement II

When the tool is placed in the reference point, the distance from the reference point to tool tip can be measured and set as tool offset data.



(3) Tool offset data setting by tool length measurement



1) Tool length measurement I

START

↓

Position the tool to the reference point by making return to reference point, etc.

↓

Select a measurement tool.

↓

Set the absolute value setting mode.

↓

Turn on the machine operation board TLM switch.

↓

Move the tool to the measurement point by making jog feed or manual handle feed.

↓

Upon completion of measurement, specify the tool offset number.

Setting and display on the TOOL OFFSET screen

Select the TOOL OFFSET screen.

Enter "A" in # ( ), then press the [INPUT] key. "#A: ABS" is displayed in reverse video.

To message "TLM" is displayed on the TOOL OFFSET screen.

0 is displayed in DATA ( ) field.

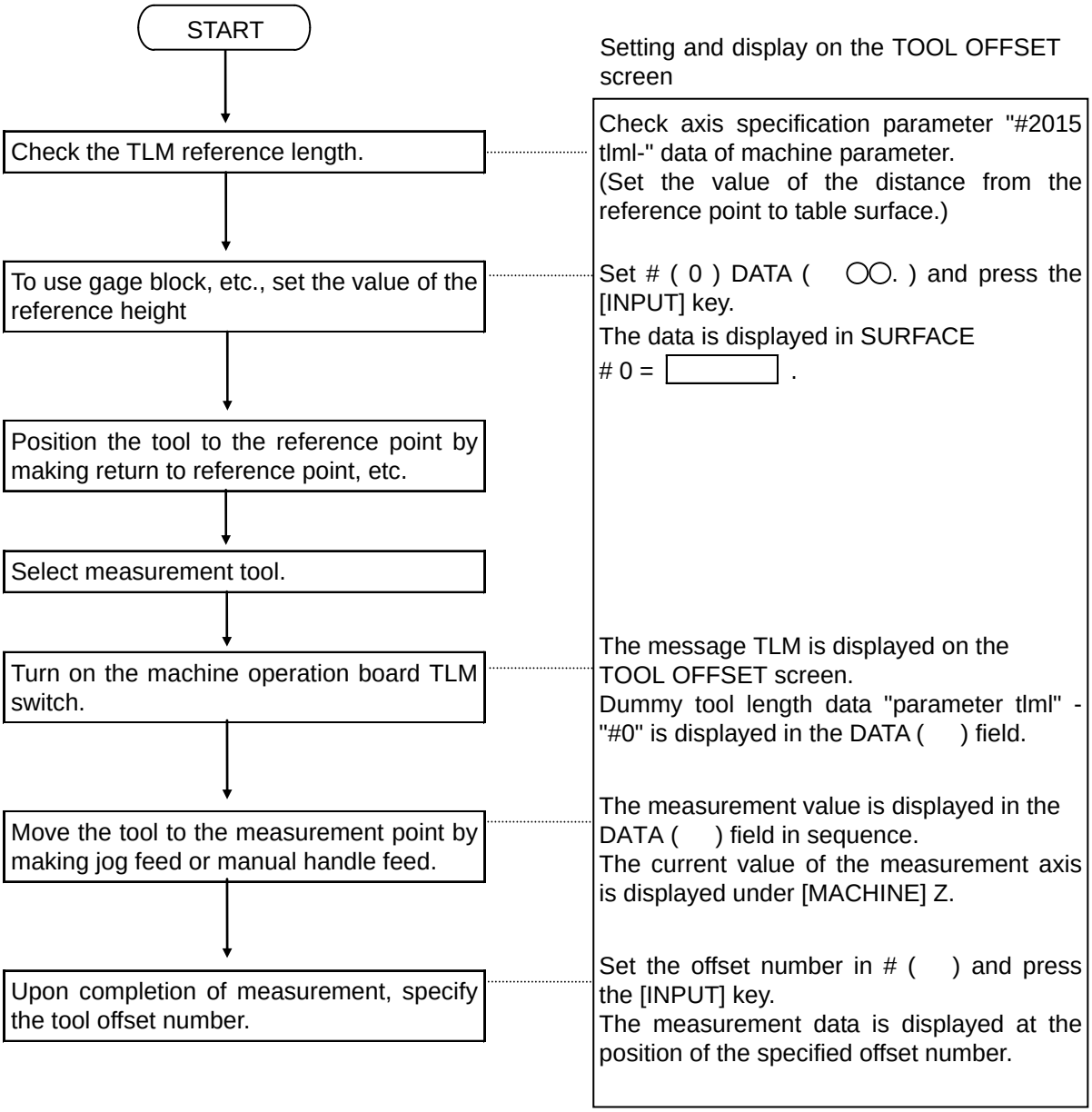
The measurement value is displayed in DATA ( ) field in sequence.
The measurement value is also displayed at [TLM].
The current value of the measurement axis is displayed at [MACHINE].

Set the offset number in # ( ) and press the [INPUT] key.

The measurement data is displayed at the position of the specified offset number.

(Note) For operation procedure, also refer to "Chapter III Machine Operation".

2) Tool length measurement II

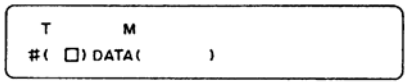


2.3.1.5 Manual Numeric Command Operation on the TOOL OFFSET Screen (M, T)

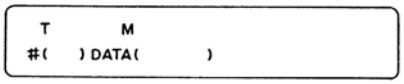
To execute a manual numeric command on the TOOL OFFSET screen, first change the mode from usual data setting to manual numeric command. The M and T commands can be executed by screen operation in manual numeric command mode.

(1) Changing the mode from usual data setting to manual numeric command
In the usual data setting mode, the cursor is displayed in the data setting field.
It is not displayed in manual numeric command mode. By checking this difference, make sure that the mode has changed.

<Normal data setting mode>

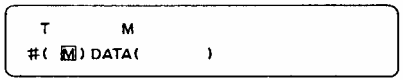


<Manual numerical command mode>

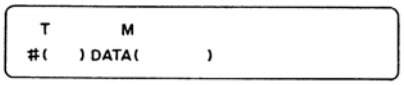


Change the mode by the following operations:

(a) Set "M"(Manual) in the first parenthesis pair of the setting field.
This operation is necessary regardless of the command (M, T).



(b) Press the [INPUT] key. The mode changes to manual numeric command.



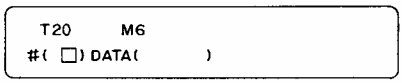
Data is cleared from the setting field. The cursor is also cleared from the screen.

(2) Execution the manual numeric command ... Execute this after operation (1) above.
a) Press the address key corresponding to the command. The corresponding command value display field is highlighted, and the manual numeric command input mode is activated. To execute the tool function, input "T". To execute the miscellaneous function, input "M".
b) Input the specified numerics from keys.
c) Press the [INPUT] key. The command is executed.

(Note) The manual numeric command operation is the same as the operation for the POSITION screen. See "2.2.1.2 Manual Numerical Value Command (S, T, M)" in POSITION of MONITOR screen for details.

(3) Returning the mode from manual numeric command to usual data setting

(a) Press the [↓] key. The usual data setting mode returns.



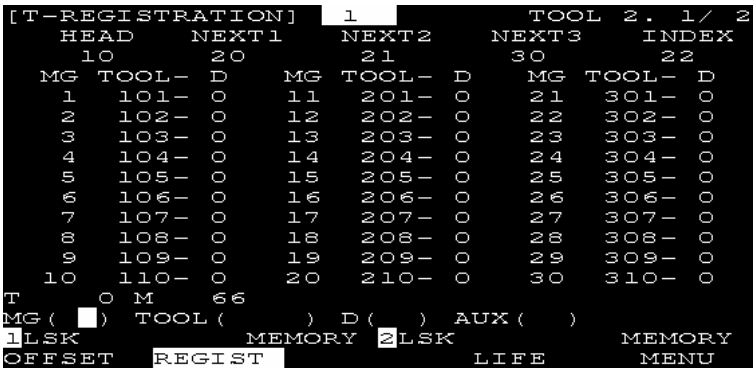
The cursor is displayed in first parenthesis pairs. After this, usual data setting is enabled.

2.3.2 Tool Registration

When the menu [REGIST] is pressed, the TOOL REGISTRATION screen is displayed. The use of this screen varies depending on the user PLC. For details, refer to the appropriate manual issued by the machine manufacturer.

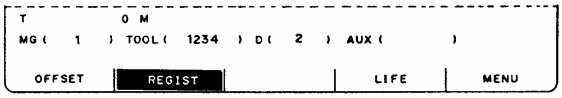
2.3.2.1 Function Outline

(1) The used tools can be registered in magazine pots.
(2) When magazine pots and tool numbers are changed by the tool selection or tool replacement command, new tool numbers are displayed.
(3) Any data can be set in setting area AUX ( ) and sequence processing can be performed by using user PLC.
(4) Tools can be registered under HEAD, NEXT 1 to NEXT 3, and INDEX displayed on the screen top. The display names and the number of display pieces can also be changed.
(5) Although the number of registered tools varies depending on the specifications, a maximum of 80 tools can be registered (the maximum number of digits of a tool number is four.)
(6) Tool selection and head replacement can be made by using manual numeric commands.

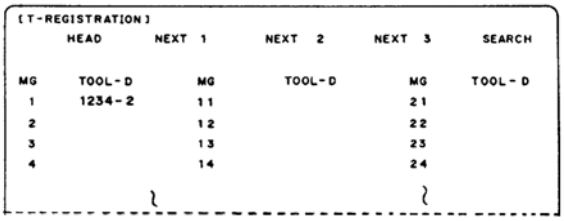


2.3.2.2 Tool Registration in Magazine Pot

Set “1” in MG ( ),
“1234” in TOOL ( ), and
“2” in D ( ).



Press the [INPUT] key.



The tool number and data in D are displayed in the specified magazine number area. The magazine number in setting area MG ( ) is incremented by one and the data in other parenthesis pairs disappears.
If a number other than magazine numbers listed in the data display area is set, the screen is changed to the screen corresponding to the setup magazine number when the [INPUT] key is first pressed. When the [INPUT] key is pressed again, the data set in the area is displayed.

(Note) For the functions and purpose of data in D, refer to the appropriate manual issued by the machine manufacturer.

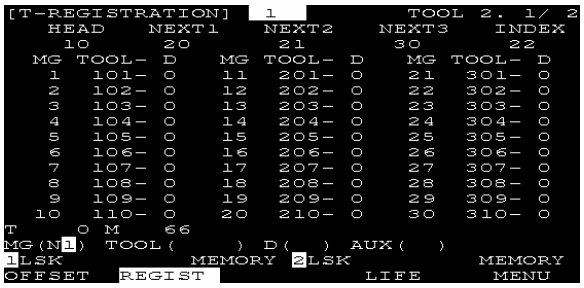
2.3.2.3 Tool Registration in HEAD, NEXT, and INDEX

This function is used to change display data when the tool number set in magazine pot differs from the displayed tool number.

Set in HEAD MG (SP) TOOL ( )
Set in NEXT 1 MG (N1) TOOL ( )
Set in NEXT 2 MG (N2) TOOL ( )
Set in NEXT 3 MG (N3) TOOL ( )
Set in INDEX MG (N4) TOOL ( )

Set “SP” in MG ( ) and "10" in TOOL ( ).

☞



Press the [INPUT] key.

"10" is displayed below HEAD in the data display area and a change is made to MG (N1) in the data setting area.

(Note) Although the title display on the screen top (HEAD, NEXT 1 to NEXT 3, INDEX) varies depending on the machine manufacturer, data is set by using SP And N1 to N4.

2.3.2.4 Tool Registration Data Clear

Set “CL” in MG ( ).

Press the [INPUT] key.

☞

All data displayed in HEAD, NEXT 1 to NEXT 3, INDEX, and MG1 to MGn is cleared.

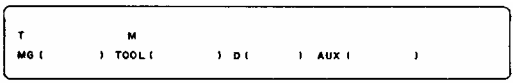
2.3.2.5 Manual Numeric Command Operation on the TOOL REGISTRATION Screen (M, T)

To execute a manual numeric command on the TOOL REGISTRATION screen, first change the mode from usual data setting to manual numeric command. The M and T commands can be executed by screen operation in manual numeric command mode.

(1) Changing the mode from usual data setting to manual numeric command
In the usual data setting mode, the cursor is displayed in the data setting field. It is not displayed in manual numeric command mode. By checking this difference, make sure that the mode has changed.

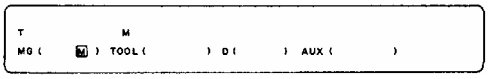
<Normal data setting mode>

<Manual numerical command mode>

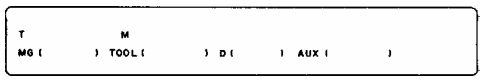


Change the mode by the following operations:

(a) Set "M"(Manual) in the first parenthesis pair of the setting field.
This operation is necessary regardless of the command (M, T).



(b) Press the [INPUT] key. The mode changes to manual numeric command.

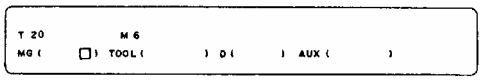


Data is cleared from the setting field.
The cursor is also cleared from the screen.

(2) Execution of the manual numeric command ... Execute this after operation (1) above.
a) Press the address key corresponding to the command. The corresponding command value display field is highlighted, and the manual numeric command input mode is activated. To execute the tool function, input "T". To execute the miscellaneous function, input "M".
b) Input the specified numerics from keys.
c) Press the [INPUT] key. The command is executed.

(Note) The manual numeric command operation is the same as the operation for the POSITION screen. See "2.2.1.2 Manual Numerical Value Command (S, T, M)" in POSITION of MONITOR screen for details.

(3) Returning the mode from manual numeric command to usual data setting
(a) Press the [↓] key. The usual data setting mode returns.



The cursor is displayed in first parenthesis pairs. After this, usual data setting is enabled.

2.3.3 Tool Life

When the menu [LIFE] is pressed, the TOOL LIFE screen is displayed.
The TOOL LIFE screen consists of the HEAD, NEXT, GROUP LIST screen and TOOL LIFE data screen.

2.3.3.1 Function Outline

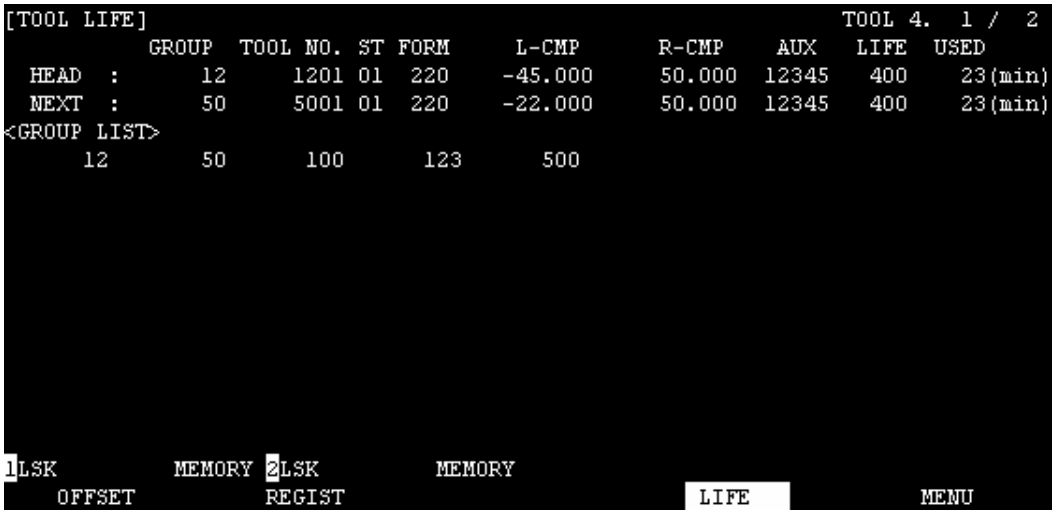
The specifications for TOOL LIFE changes depending on parameter "#1096 T_ Ltype" and "#1103 T_Life" to "#1106 T count". Confirm the explanations described in the appropriate section in setup parameter.

(1) TOOL LIFE I ("#1096 T_Ltype" = 1)
The use time or count of the programmed tool is accumulated and the tool use state is monitored.

(2) TOOL LIFE II ("#1096 T_Ltype" = 2)
TOOL LIFE II is provided by adding the spare tool selection function to TOOL LIFE I. A spare tool is selected among programmed tool commands. Tool offset and tool radius compensation are performed for the selected tool.

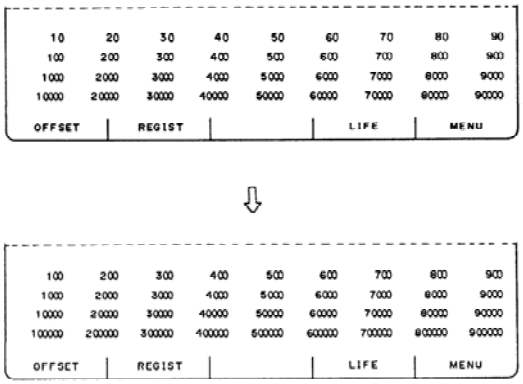
2.3.3.2 TOOL LIFE Screen Data Display

(1) HEAD, NEXT, GROUP LIST screen
(Display only. No data can be set).



Display item	Explanation
HEAD NEXT	The tool numbers and TOOL LIFE data of the tools in HEAD and NEXT are displayed. When TOOL LIFE is ineffective, only the tool numbers are displayed.
GROUP LIST	The group numbers registered as TOOL LIFE data are displayed.

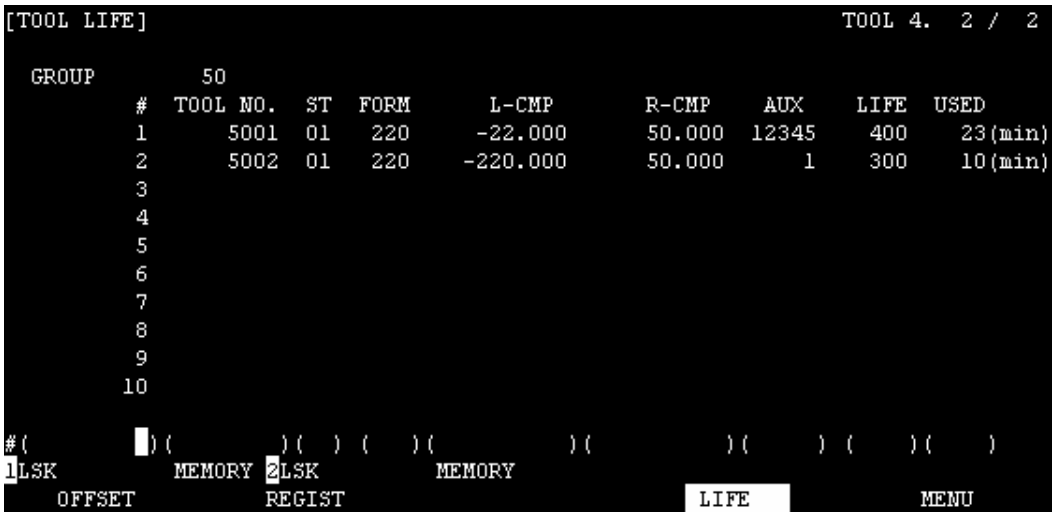
A maximum of 90 group numbers are displayed on one screen. If the number of the group numbers exceeds 90, the screen is scrolled every line by using the [↑] or [↓] key.

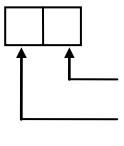


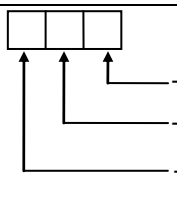
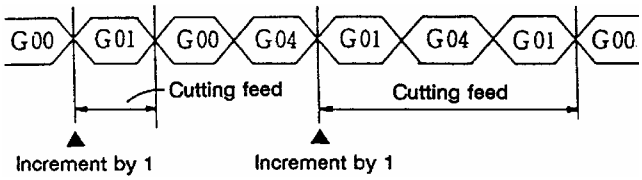
One-line scroll by pressing the [↓] key.

(2) TOOL LIFE data screen

The TOOL LIFE data displays tool data in group units. If the number of lines displayed exceeds one screen area, press the [NEXT] or [BACK] key to scroll the screen.
The data to control the life of a group of tools can be displayed and set.



Display item	Explanation	Setting range
GROUP	Group number of the tools which carry out tool life control.	1 to 99999999
#1 to #10	These are data setting numbers, not magazine pot numbers.	
TOOL NO.	Number given to each tool. A maximum of 400 tool numbers can be registered depending on the specifications.	1 to 99999999
ST	 Tool Status 0: Unused tool. Normally, it is set to 0 when the tool is replaced with a new tool. 1: Used tool. It is set to 1 when actual cutting is begun. 2: Normal life tool. It is set to 2 when the use data (time, count) exceeds the life data. 3: Tool error 1 tool 4: Tool error 2 tool **(Note)** 3 and 4 depend on the machine manufacturer specifications.	

Display item	Explanation	Setting range
FORM	 a. Tool life control mode 0: Use time controlled by the time during which cutting feed is performed. 1: Mount count Controlled by the number of times the tool is used as a single tool. 2: Work count Work count The work count is made whenever a rapid traverse feed (G00 etc.) command is replaced by a cutting feed command (G01, G02, G03, etc.). However, rapid traverse feed or cutting feed commands inducing no movement are ignored.  b. Tool radius compensation data format 0: Compensation number Compensation data in tool data is handled as compensation number. It is replaced with the compensation number commanded in a machining program for compensation. 1: Addition compensation mode Compensation data in tool data is handled as addition compensation amount. It is added to the compensation amount indicated by the compensation number commanded in a machining program for compensation. 2: Direct compensation mode Compensation data in tool data is handled as direct compensation amount. It is replaced with the compensation amount indicated by the compensation number commanded in a machining program for compensation. c. Tool length compensation data format 0: Compensation number 1: Addition compensation amount 2: Direct compensation amount The functions are the same as in b. above.	

Display item	Explanation	Setting range
L-CMP R-CMP	These depend on the data format specified in FORM.	Compensation number: 1 to 400 Addition compensation amount: ±1 to 99999.999 Direct compensation amount: ±1 to 99999.999
AUX	This depends on the machine manufacturer specifications.	0 to 65535
LIFE	Life of each tool. It is displayed in the use time (minutes), mount count (the number of times the tool has been mounted on the spindle), or work count (the number of times drilling has been performed) as specified in FORM. If it is set to 0, life infinity is specified.	Use time: 0 to 4000 (min) Mount count: 0 to 9999 (times) Work count: 0 to 9999 (times)
USE	Use data of each tool is displayed in the form as specified in FORM (a. Tool life control mode). **(Note)** Use data is not counted during machine lock, miscellaneous function lock, dry run, or single block mode.	Use time: 0 to 4000 (min) Mount count: 0 to 9999 (times) Work count: 0 to 9999 (times)

(Note) The No. of uses/No. of mounts depends on the model.

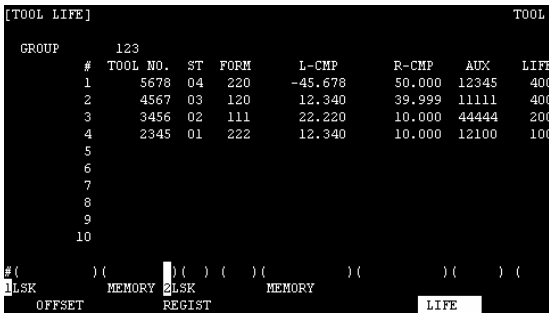
2.3.3.3 TOOL LIFE Data Display and Setting (TOOL LIFE Data Screen Page 2)

(1) Data display

When the menu key [NEXT] is pressed on the HEAD, NEXT, GROUP list screen (previously described), the TOOL LIFE screen is displayed.
The data in the group previously set is displayed. If no data is set, the screen is displayed with blank in the data area.

Set G123 in # ( ).

Press the [INPUT] key.



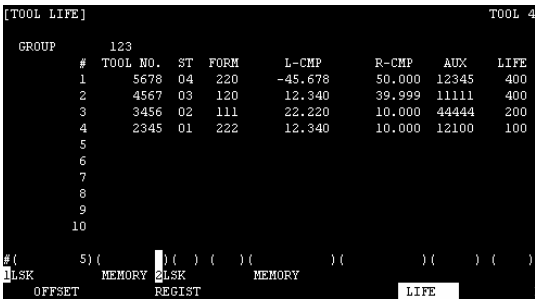
The tool data registered in group 123 is displayed in the registration order. If the data exceeds one screen, the remaining data can be seen by using the [NEXT] key.

(2) Data registration

Select TOOL LIFE data screen for the group in which data is to be registered.

Set G○○○ in # ( ), then press the [INPUT] key.

Set necessary data of TOOL NO. to USED in the corresponding parenthesis pairs in the ascending order of the (#) numbers, then press the [INPUT] key.

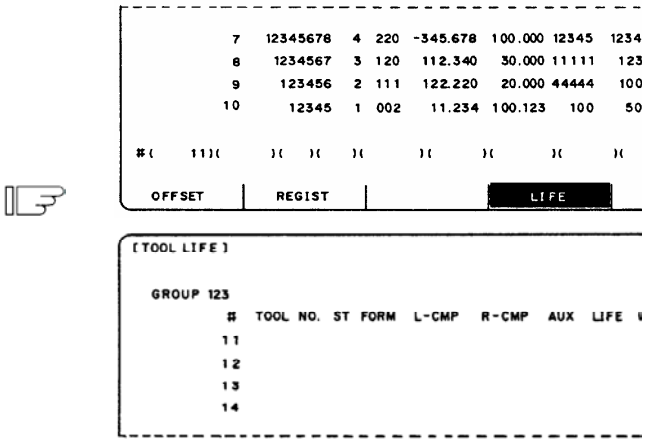


The data setting number incremented by one is set in # ( ).

(Note 1) If TOOL NO. and ST (status) are not set, setup data becomes invalid.
(Note 2) A single tool cannot be registered in more than one group.

To register data exceeding the number of data pieces that can be displayed on a screen, set the # number only. New data can be registered.

Set “11” in # ( ), then press the [INPUT] key.



(3) Data change

1) Display the TOOL LIFE data screen for the group in which the tool whose data is to be changed is registered.
2) Set the # number of the data to be changed and new data in given parenthesis pairs of the setting area, then press the [INPUT] key.
3) After setting, the # number is incremented by one and is set in setting area ( ).
4) To change data under # number not displayed on the screen, change the screen by using the [NEXT] or [BACK] key or setting the number in setting area # ( ).
5) By changing a registered tool number to 0, the tool can be deleted.

(4) Deletion in group units

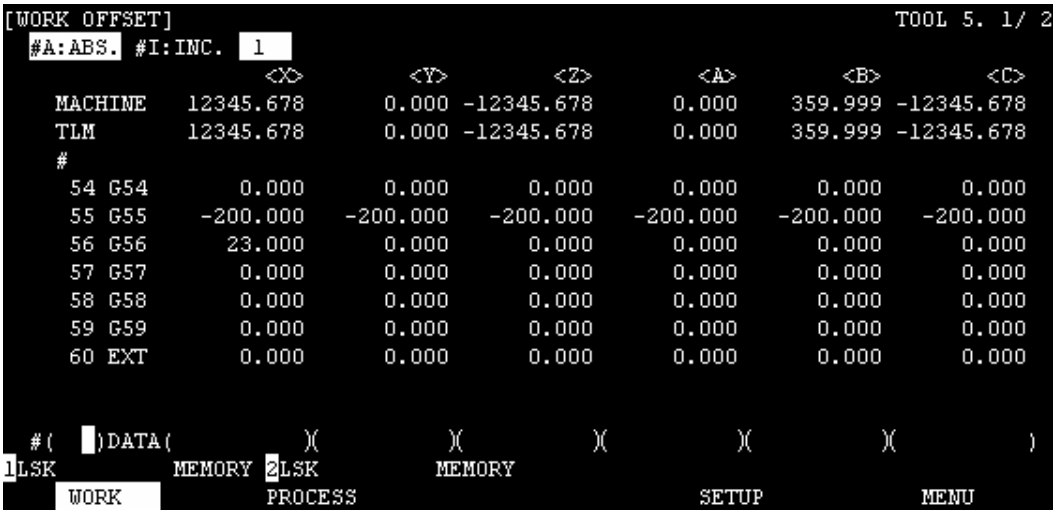
To delete all data in one group, select the TOOL LIFE data screen for the group to be deleted and press the [SHIFT] key, [C.B CAN] then [INPUT].

2.3.3.4 Clear of All TOOL LIFE Data (HEAD, NEXT, GROUP LIST Screen Page 1)

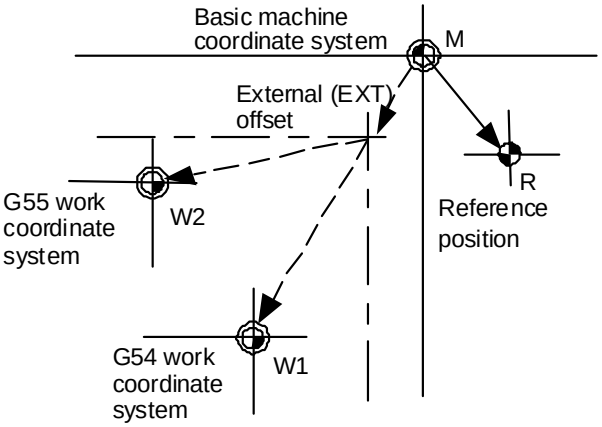
To clear all data, select the HEAD, NEXT, GROUP LIST screen and press [SHIFT] key, [C.B CAN], then [INPUT].

2.3.4 Workpiece Coordinate

Pressing the menu [WORK] displays the WORK OFFSET screen.
The workpiece coordinate system offset data can be set or displayed for the number of axes.



Display item	Explanation
#A: ABS. #I : INC.	The currently effective one of the setting modes (absolute and incremental) is displayed in reverse video. Before setting data, check the mode.

#	Parameter	Explanation	Setting range (unit)
54 55 56 57 58 59 60	G54 offset G55 offset G56 offset G57 offset G58 offset G59 offset EXT offset	Specify the workpiece coordinate system from G54 to G59 and external workpiece coordinate offset. Workpiece coordinate system offset data can be specified in absolute or incremental values. 	±99999.999 (mm)

2.3.4.1 Setting Workpiece Coordinate System Offset Data

(1) Enter the number corresponding to the workpiece coordinate system in # ( ), put offset data in DATA ( ), then press the [INPUT] key. This defines workpiece coordinate system offset data.
(2) The workpiece coordinate system offset data thus defined is then displayed at the position of the workpiece coordinate system and the number in # ( ) changes to the next setting number and the data in DATA ( ) disappears.
(3) The number displayed in # ( ) is incremented and decremented by one when pressing the arrow keys [↑] [↓].
(4) Typing "I" in # ( ) and pressing the [INPUT] key puts the setting mode to the incremental mode. Data set in the incremental mode is added to the displayed data.
Typing "A" in # ( ) and pressing the [INPUT] key cancels the incremental mode and restores the absolute mode.

2.3.4.2 Setting External Workpiece Coordinate System Offset Data

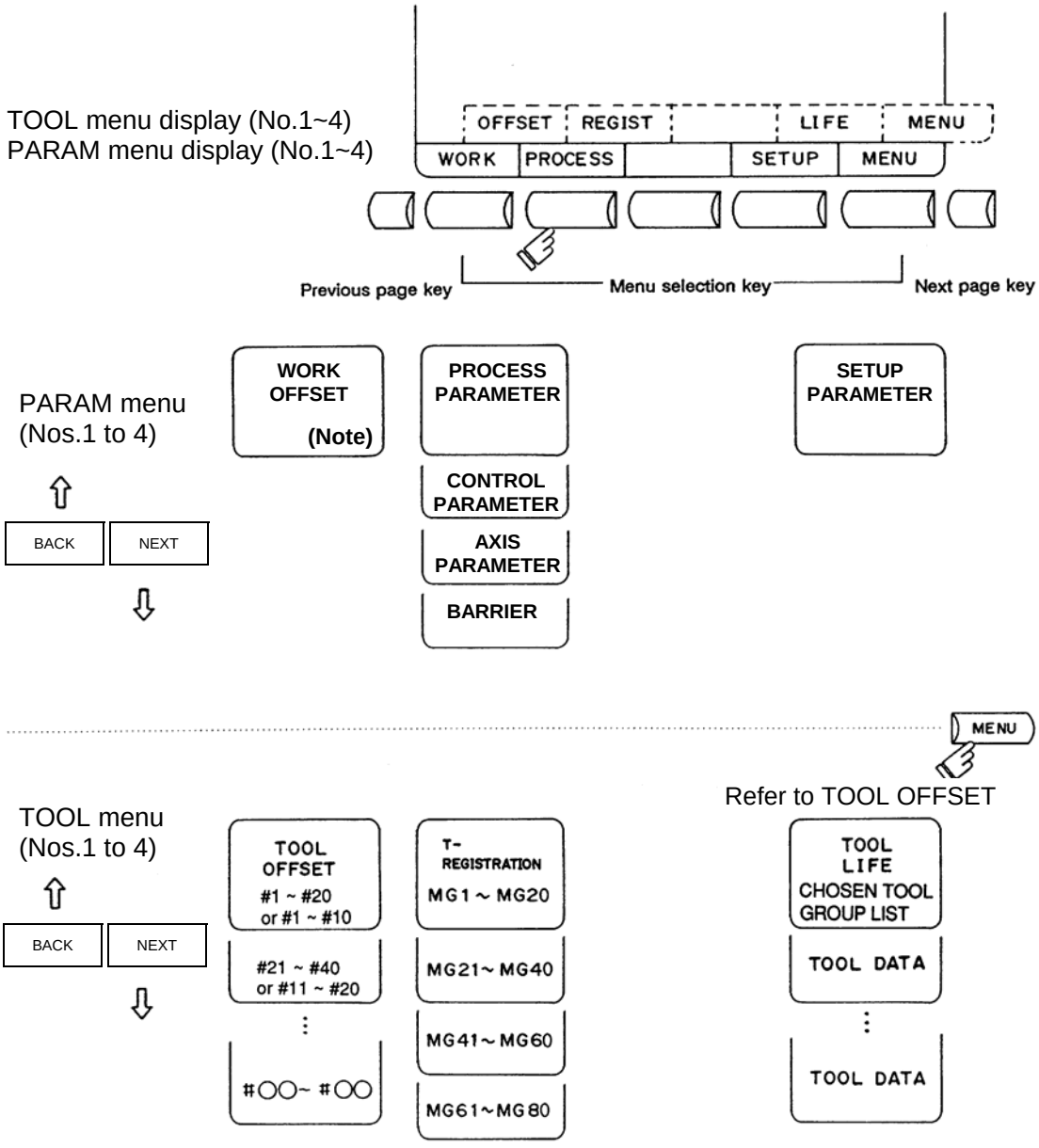
By measuring the coordinate system deviation with an external touch sensor, etc., all workpiece coordinate systems G54 to G59 can be offset.
External workpiece coordinate system data can be defined in one of two ways: inputting external data directly to the external offset (EXT) or entering it into the setting field on the screen (EXT).
The setting method is the same as for workpiece coordinate system offset data.

2.3.4.3 Displaying Machine Position Data

As with the POSITION screen, data of each axis displayed at the [MACHINE] on the WORK OFFSET screen indicates the current machine position in reference to the zero point on the basic machine coordinate system; it cannot be changed on this screen.

2.4 Parameters

When the function selection key [TOOL PARAM] is pressed, the following menu appears:
TOOL menu is displayed after the power is turned on. To display PARAM menu, use menu [MENU] on the TOOL screen.



(Note) "WORK" is included in the TOOL menu. Refer to the "Tool Offset" section.

(Note) Selection of inch/metric unit
When setting value of "#1041 I_inch" is changed, the unit of length is changed after the power is turned ON again.
Among parameters concerning length, following items are not changed automatically, therefore change the setting values to match the new unit system when the unit system is changed.

Tool compensation amount (Tool length compensation amount, tool wear compensation amount and tool nose compensation amount)		
Workpiece coordinate offset		
Machining parameter	#8004 SPEED	#8013 G83n
	#8005 ZONE r	#8016 G71 MINIMUM
	#8006 ZONE d	#8017 G71 DELTA-D
	#8009 DSC. ZONE	#8018 G84/G74n
	#8010 ABS. MAX.	
	#8011 INC. MAX.	
	#8012 G73n	
Axis parameter	#8204 OT-CHECK-N	
	#8205 OT-CHECK-P	
	#8206 TOOL CHG.P	
	#8209 G60 SHIFT	
Barrier data	#8300 to #8306	
Basic specification parameter	#1084 RadErr	

"#8004 SPEED" is 10 inches/min unit for the inch system.

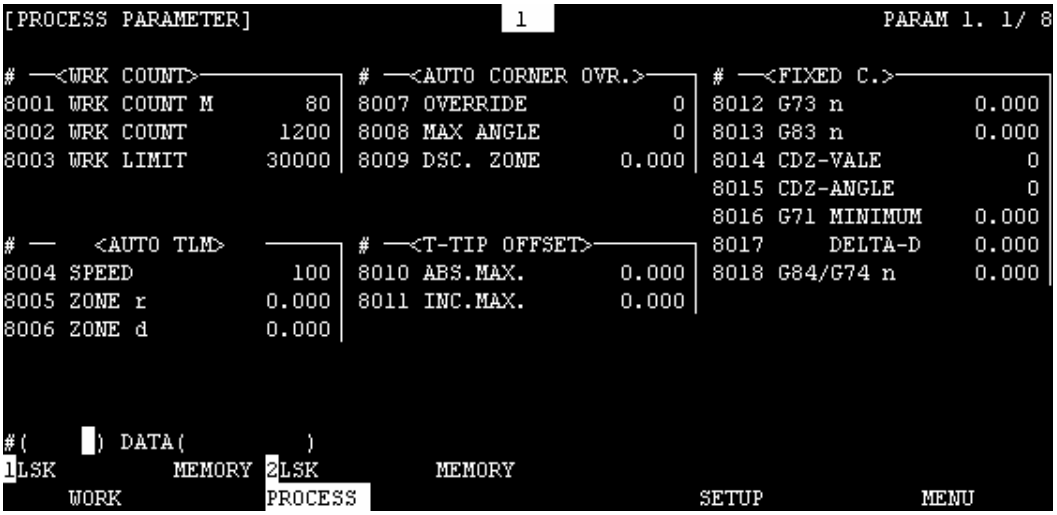
The parameter input setting units are as follows.

Input unit "#1003 iunit"	**Linear axis "#1017 rot"=0**		**Rotary axis "#1017 rot"=1**
	Machine constant:mm "#1040 M_inch"=0	**Machine constant:inch "#1040 M_inch"=1**	
B	0.001 mm	0.0001 inch	0.001°
C	0.0001 mm	0.00001 inch	0.0001°

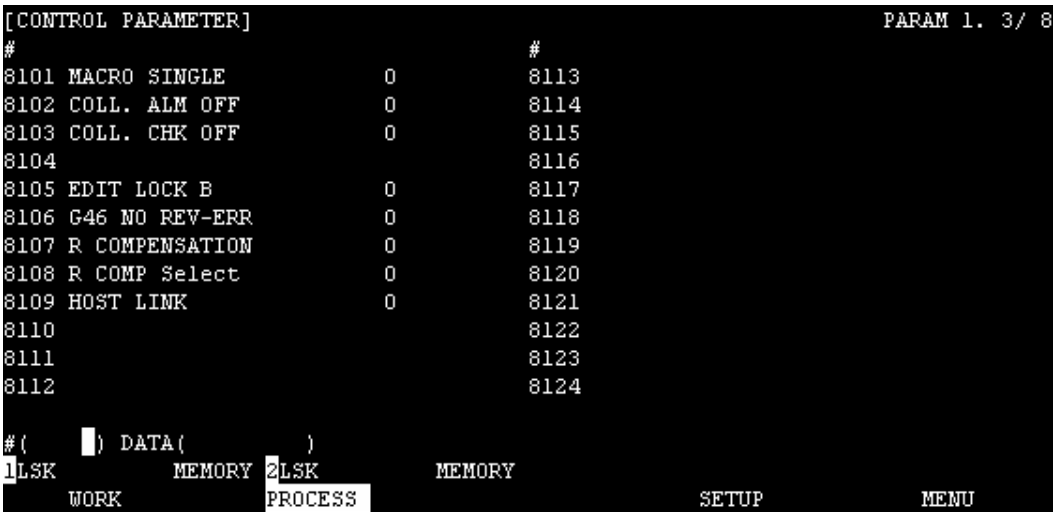
2.4.1 Machining Parameters

Pressing the menu [PROCESS] displays the PROCESS PARAMER screen.
Refer to "Appendix 8. List of User parameters" for details of each parameter.

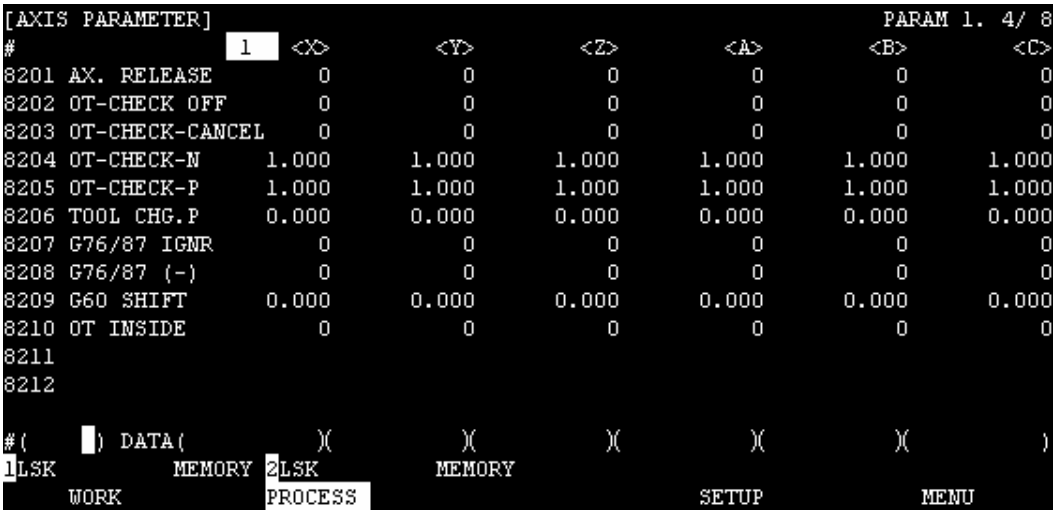
2.4.1.1 Process Parameters



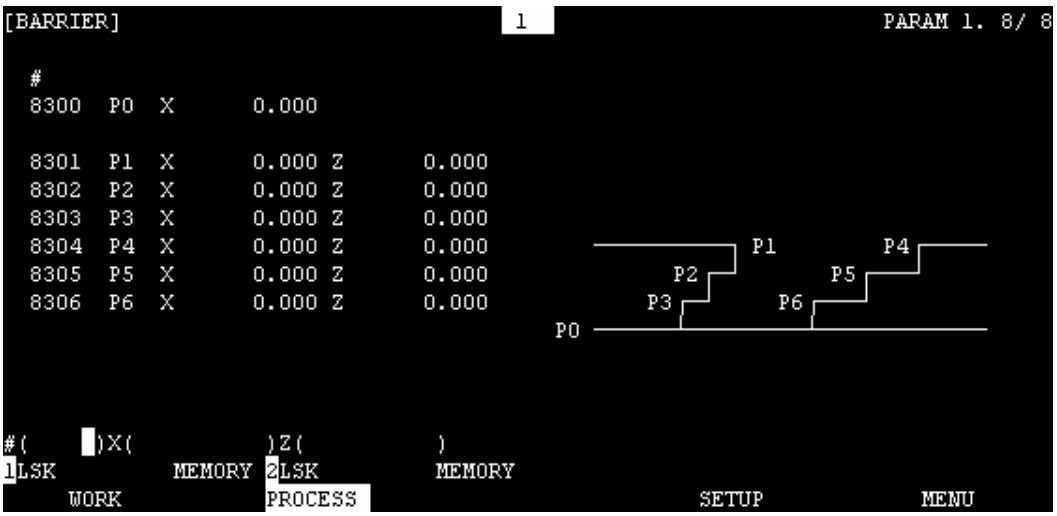
2.4.1.2 Control Parameters



2.4.1.3 Axis Parameters

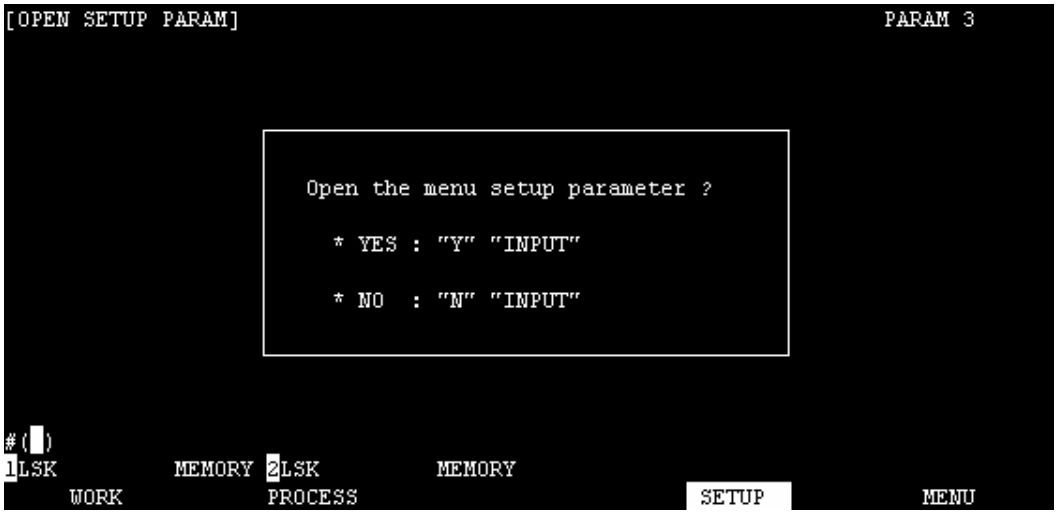


2.4.1.4 Barrier Data



2.4.2 Setup Parameters

Pressing the menu [SET UP] displays the OPEN SETUP PARAM screen.
The system's basic parameters are normally hidden as setup parameters to prevent mistaken operations and to simplify the display.
The setup parameters can be displayed and set by making a declaration to open the setup parameters on this screen.



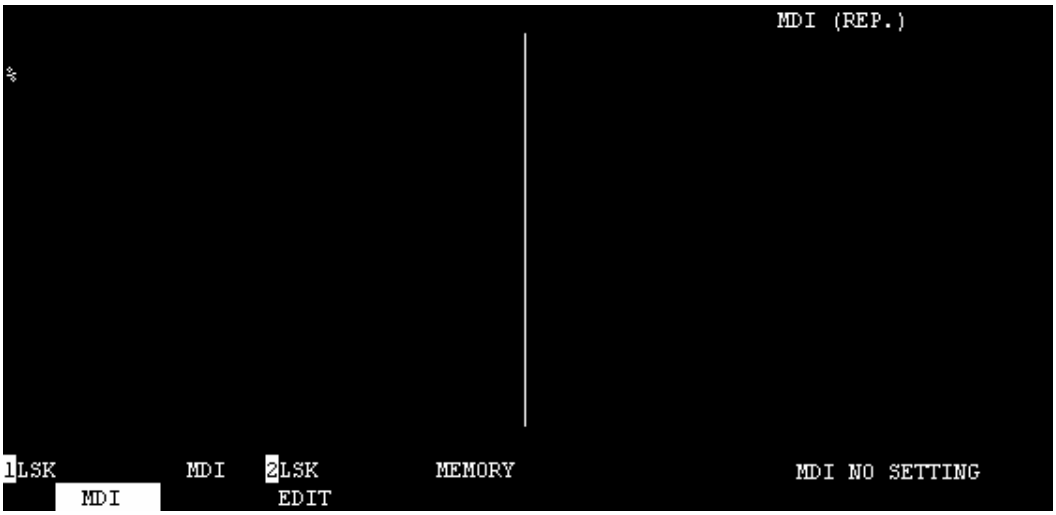
1) Select the setup parameter.
   Key-in "Y" in # ( ), and then press [INPUT].
   The normally hidden setup parameter menu will display when the menu changes over.
   The required menu can be selected to display and set the setup parameters.
2) Cancel the setup parameter selection.
   Key-in "N" in # ( ), and then press [INPUT].
   The setup parameter menu will disappear.

(Note) The setup parameters are not displayed when the power is turned on.

Refer to “Setup Manual” for details on the setup parameters.
Always turn the power OFF after selecting the setup parameters.

2.5 Program

MDI Screen



Edit Screen

2.5.1 Function Outline

(1) Function outline

When the function selection key [EDIT MDI] is pressed, the EDIT or MDI screen appears.

The EDIT screen enables you to add, delete, or change the machining program contents stored in memory. It also enables you to register a new program number in memory and prepare a new program on the screen.

The MDI screen enables you to set, correct, or erase MDI data. It also enables you to register a program prepared as MDI data in memory as a machining program.

(2) Display when the screen is selected

After the power is turned ON, the first screen when selecting [PROGRAM] depends on the value set in the parameter "#1139Edtype".

"0": MDI screen

"1" or "2": Edit or MDI screen is switched automatically by the selected operation mode

To edit a machining program on the EDIT screen, use the menu key to change the screen. No programs to be edited are called on the initial edit screen. Perform [SERCH] or [PROGRAM] operation. To edit a program already registered in memory, perform [SERCH] operation. To register a new program in memory, perform [PROGRAM] operation.

If the MDI screen is selected, MDI data can be entered as it is without operation such as a search.

If [EDIT MDI] screen operation is interrupted and any other function is executed, the previous screen selected (MDI or EDIT) will appear and the previous data will be displayed by again selecting the [EDIT MDI] screen. Then, the data input or edit operation can be continued.

(3) Fixed cycle program edit

To edit a fixed cycle program, set a given parameter.

The EDIT screen can be used to edit a fixed cycle program by setting 1 in parameter #1166 "fixpro".

(4) Editing macro operators

If a character string that matches a macro operator exists in the machining program (including a comment statement), it is automatically converted into the corresponding intermediate code during editing. This may cause a string different from that entered to be displayed during editing.

(Example) ATN → ATAN
SQR → SQRT
RND → ROUND

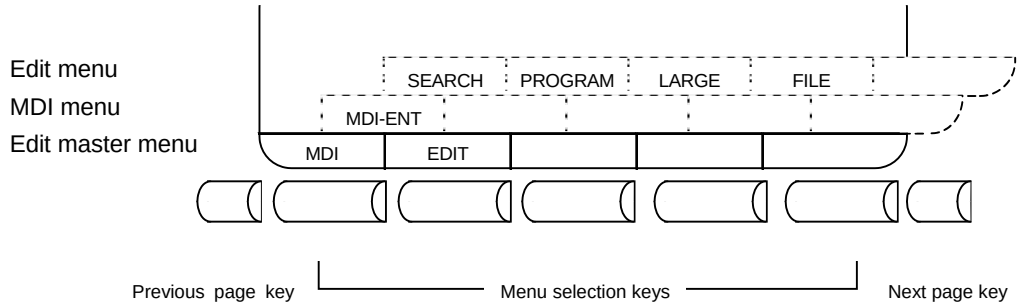
2.5.2 Edit type

There are two edit types for the edit screen: "Screen edit" and "Word edit". Select the type with parameter "#1139 Edtype".
The menu type for the word edit can be selected by the parameter "#1227 aux11/bit6".
Only "Screen edit" is available for MDI screen's edit type.

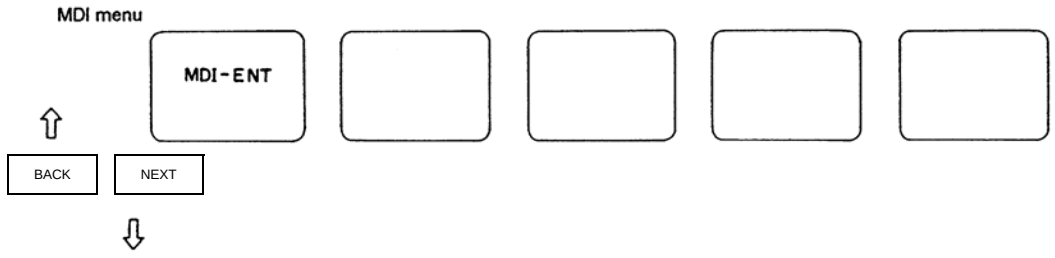
#1139	#1227/bit6	MDI Screen	Edit screen	Edit/MDI screen automatic switch
0	-	Screen Edit	Screen edit	No
1	-	Screen Edit	Screen edit	Yes
2	0	Screen Edit	Word edit, Menu type 1	Yes
3	1	Screen Edit	Word edit, Menu type 2	Yes

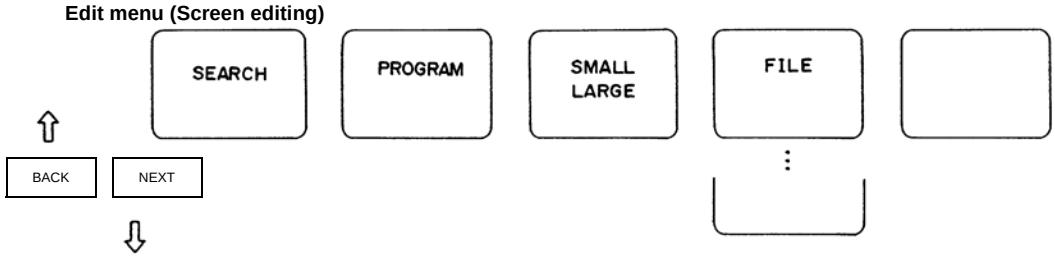
2.5.3 Menu Function

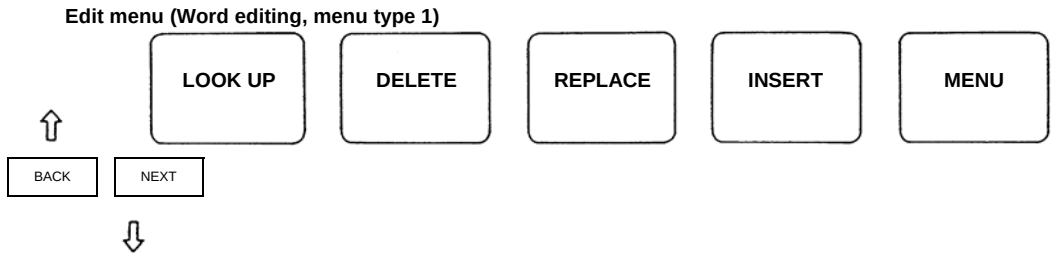
Pressing the function selection key [EDIT MDI] displays the following menu.

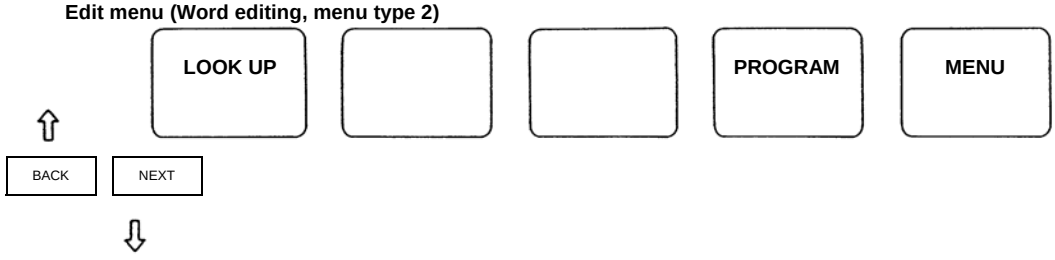


Selecting [MDI] or [EDIT] displays the following menu:



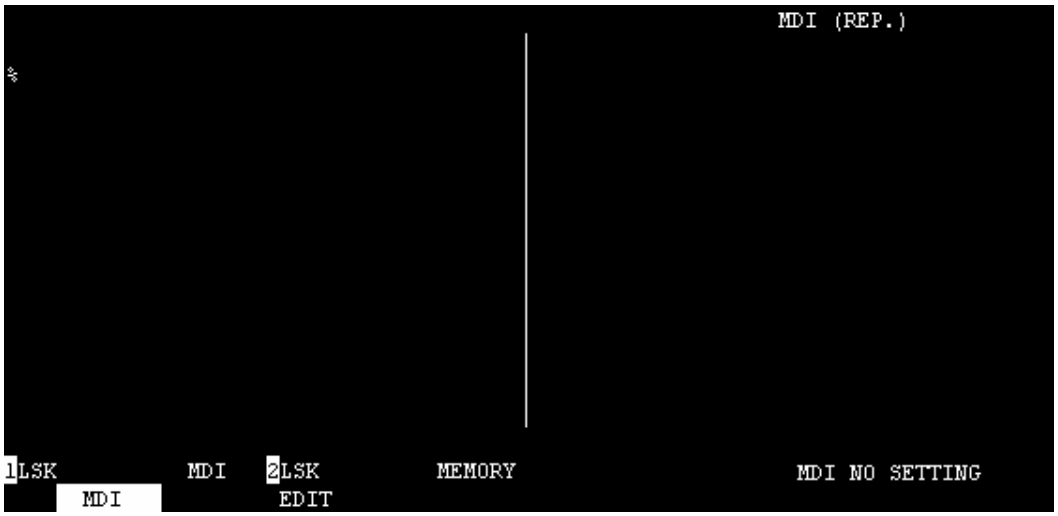






2.5.3.1 MDI Screen Menu Function

(1) Menu when [MDI] is selected (Only when "#1139 Edtype" is set to "0".)



Menu	Function
[MDI]	Reverse display of MDI menu means that MDI screen is selected. MDI data can be set on the MDI screen.
[EDIT]	Use this key to change the MDI screen to the EDIT screen.

(2) MDI screen extension operation menu

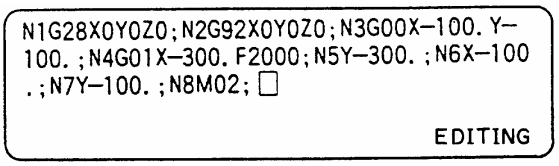
Menu	Function
MDI-ENT	MDI data can be registered in memory as a machining program.

Extension operation menu is also highlighted when it is selected. When one extension operation menu is selected, its corresponding extension operation is enabled and MDI data cannot be set. When no extension operation menu is selected, MDI data can be set.
When an extension operation menu key is once pressed, the extension operation menu is selected. When the key is again pressed, the extension operation menu is unselected. At normal completion of setting processing, automatically it becomes unselected.

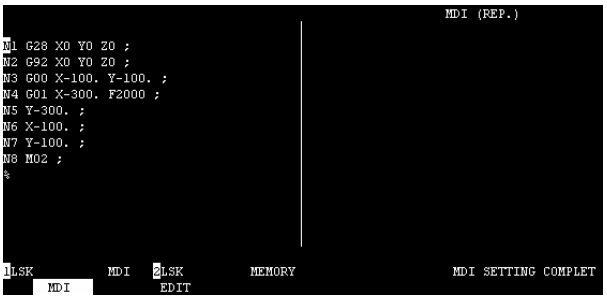
(3) MDI data setting

(1) Enter MDI data by pressing the data keys in sequence according to the machining program listing.

```
N1   G28   X0 Y0 Z0 ;
N2   G92   X0 Y0 Z0 ;
N3   G00   X−100. Y−100. ;
N4   G01   X−300.   F2000;
N5   Y−300. ;
N6   X−100. ;
N7   Y−100. ;
N8   M02    ;
```



Press the [INPUT] key.



(2)
1) The data is written into the MDI memory area.
2) It is displayed on every line per block.
3) The message MDI SETTING COMPLET is displayed and MDI operation is enabled. The running start position is the starting block of data. The cursor is displayed in the starting block.

⚠ CAUTION

⚠ Because of key chattering, etc., during editing, "NO NOS. FOLLOWING G" commands become a "G00" operation during running.

(Note 1) If the [INPUT] key is not pressed, data is simply displayed on the screen and is not actually stored in memory. Be sure to press the [INPUT] key.

(Note 2) See “2.5.3 Program Edit Operation” for details of key operation to set MDI data.

(Note 3) Check the MDI SETTING COMPLET message before starting MDI operation. If the EDITING or MDI NO SETTING message is displayed, MDI operation cannot be started. If the [INPUT] key is pressed at the time, the MDI SETTING COMPLET message is displayed.

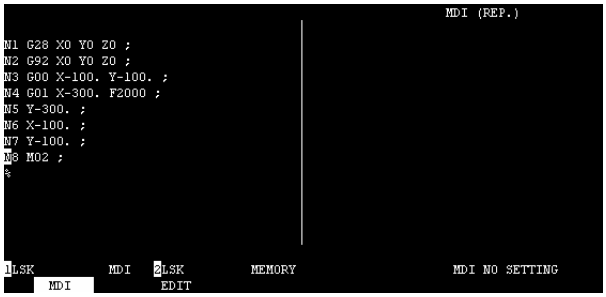
(4) Setting the MDI running start position

To start processing with a halfway block after setting MDI data, specify the starting block. First, set the data according to "Setting MDI Data." At this time, the running start position is set in the starting block of data. If it is desired to be changed, move the cursor to the head of the block to be defined as the starting position. Then, press the [INPUT] key.

(Example) When the block containing M02 is desired to be executed.

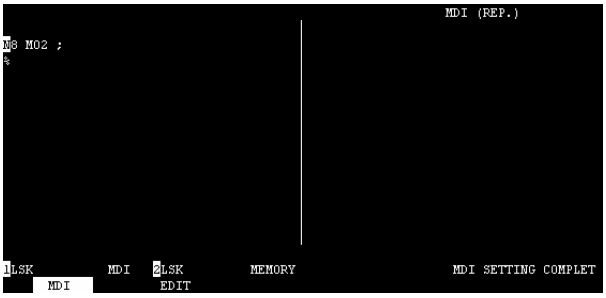
Move the cursor to the head of the block to be defined as the starting position.

1) The "MDI NO SETTING" status returns.



Press the [INPUT] key.

1) MDI running is enabled, beginning with the specified block.
2) The specified block is displayed at the top of the screen head with "MDI SETTING COMPLET" displayed.

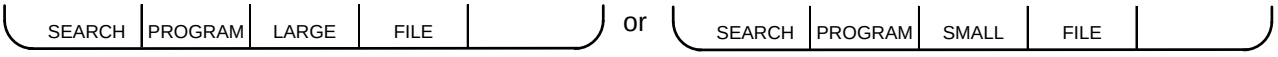


2.5.3.2 EDIT Screen Menu Function

(1) Menu when [EDIT] is selected (Only when "#1139 Edtype" is set to "0".)

Menu	Function
[EDIT]	Reverse display of EDIT menu means that EDIT screen is selected. Machining program can be set on the EDIT screen.
[MDI]	Use this key to change the EDIT screen to the MDI screen.

(2) EDIT screen extension (Screen edit) operation menu (When "#1139 Edtype" is set to "0" or "1".)



Menu	Function
SEARCH	1. Any desired character string can be searched. 2. Program number and sequence number for edit can be searched.
PROGRAM	New machining programs can be prepared and stored on the screen.
FILE	1. A list of the machining programs registered in memory can be checked. 2. Comments can be set.
LARGE	40 characters are displayed in one line on the screen.
SMALL	80 characters are displayed in one line on the screen.

Extension operation menu is also highlighted when it is selected. When one extension operation menu is selected, its corresponding extension operation is enabled and programs cannot be edited. When no extension operation is selected, program can be edited.
When an extension operation menu key is once pressed, the extension operation menu is selected. When the key is again pressed, the extension operation menu is unselected. At normal completion of setting processing, automatically it becomes unselected.

[Edit program call]

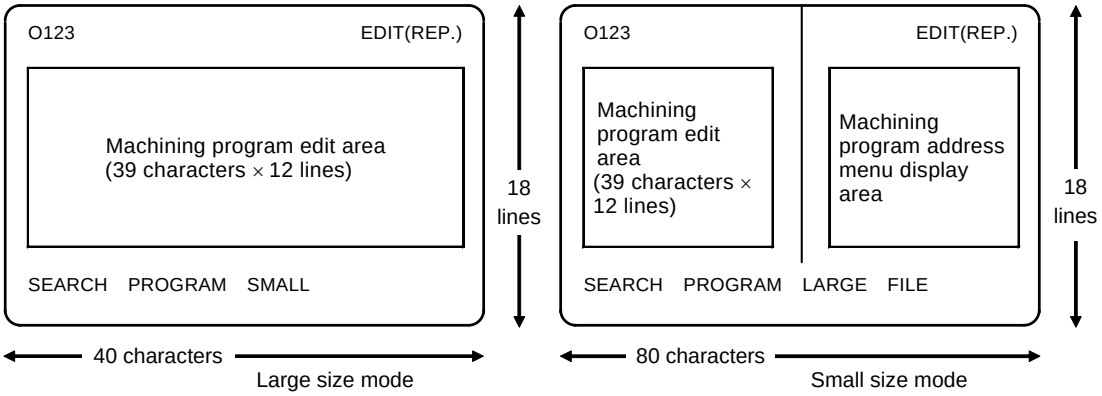
To edit a program on the EDIT screen, first press the extension operation menu [SEARCH] or [PROGRAM]. To edit an already stored program in memory, press [SEARCH] To store a new program in memory, press [PROGRAM]. For details, see 5.5.

Once the program edit operation begins, the operation is as follows: If another function screen is operated during program edit operation and then the EDIT screen is reselected, the previously edited data will be displayed. In the following cases, the system enters the status in which nothing has been called. Thus, retry data search before edit operation.

- The program being edited on the EDIT screen is condensed by the condense function. The EDIT screen is then selected.
- The program being edited on the EDIT screen is merged with another program by the merge function. The EDIT screen is then selected.

[Large-size mode/small-size mode]

The EDIT and MDI screens can be switched between the large-size and small-size modes.



In large-size mode, data search and program creation are enabled.

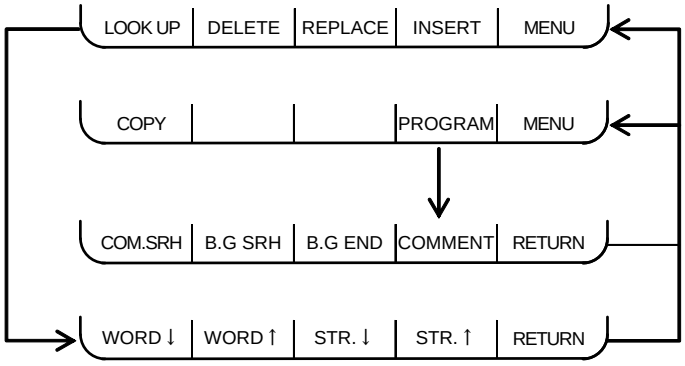
The FILE menu is not available; refer to the data input/output program list to check the stored programs.

(Note 1) Switching the mode in the EDIT screen automatically changes the mode in the MDI screen.

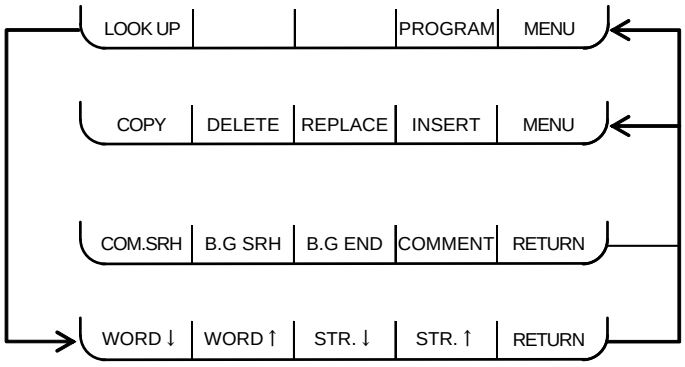
(Note 2) During editing (while message EDITING is displayed on the lower right of the screen), menu [LARGE] and [SMALL] are disabled, i.e., pressing it does not change the mode. To change the mode, the [INPUT] key must be pressed to end editing.

(Note 3) The mode thus set is held after the screen is changed or after power is turned off.

(3) Extension operation menu for edit screen (Word editing) (when #1139 is set to "2" or "3")
Menu type 1



Menu type 2



Menu	Function
LOOK UP	Switch the menu to select the direction of search.
DELETE	Delete a word where the cursor is placed. (Deleted word will appear for edit buffer area.)
REPLACE	Replace a word where the cursor is placed with edit buffer data. (Edit buffer data will not be cleared.)
INSERT	Insert edit buffer data after a word where the cursor is placed. (Edit buffer data will not be cleared.)
COPY	Copy a word where the cursor is placed to edit buffer.
PROGRAM	Switch the menu to search for a program. The searched program and a list of programs are displayed.
COM.SRH	Search the program No., sequence No. or block No. in the machining programs registered in the memory to operate automatically.
B.G SRH	Search the program No., sequence No. or block No. for the background edit. New machining program can be registered if a program No. set is not registered in the memory.
B.G END	Finish the background edit.
COMMENT	Set an outline for the machining program's functions, specifications and usages as a comment.
RETURN	Return to the top menu
WORD↓	Search for downward. The same word as the search data is searched and the cursor is moved to that word. (The search data will not be cleared.)
WORD↑	Search for upward. The same word as the search data is searched and the cursor is moved to that word. (The search data will not be cleared.)
STR.↓	Search for downward. The same string as the search data is searched and the cursor is moved to that word. (The search data will not be cleared.)
STR.↑	Search for upward. The same string as the search data is searched and the cursor is moved to that word. (The search data will not be cleared.)

2.5.4 Program Edit Operation

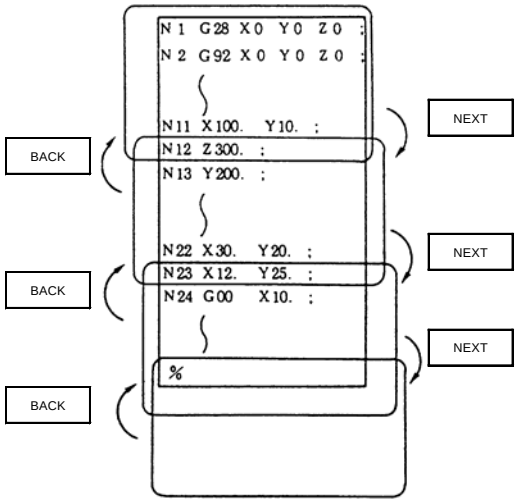
Edit operation differs between screen editing and word editing.

#1139	MDI Screen	Edit screen
0,1	Screen editing	Screen editing
2	Screen editing	Word editing

2.5.4.1 Common Operation Between Screen Editing and Word Editing

[Data display update (one screen scrolling)]

Data display on the screen can be updated in screen units by using the page key [BACK] or [NEXT].
When the [NEXT] key is pressed, the data displayed at the screen bottom is moved to the screen top; when the [BACK] key is pressed, the data displayed at the screen top is moved to the screen bottom.



For example, assume that data is displayed as shown in the right

```
N1 G28   X0   Y0   Z0 ;
N2 G92   X0   Y0   Z0 ;
N3 G00   X-300. Y-300.;
N4 G01   X-200. F2000 ;
N5 Y-200.;
N6 X200.;

N12 Z300.;
```

Press the [NEXT] key. ☞

```
N12 Z300.;
N13 Y200.;
N14
N15
      :
N22 X30. Y20.;
N23 X12. Y25.;
```

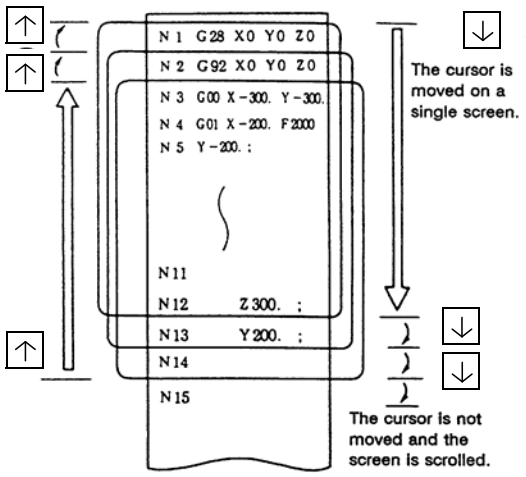
1) If a program is edited on the screen, the message as “IGNORE CHANGE? (Y/N)” appears.
   To ignore the changed contents to scroll by a whole of editing area, select "Y" key.
   To cancel the change, select "N" key.

```
N1 G28 X0 Y0 Z0;
N2 G92 X0 Y0 Z0;
N3 G00 X-300. Y-400.;
N4 G01 X-200. F2000;
N5 Y-200.;
N6 X200.;
   :
N11 X100. Y10.;
                                EDITING
```

```
N1 G28 X0 Y0 Z0;
N2 G92 X0 Y0 Z0;
N3 G00 X-300. Y-400.;
N4 G01 X-200. F2000;
N5 Y-200.;
N6 X200.;
 ...
N11 X100. Y10.;
                    IGNORE CHANGE? (Y/N)
```

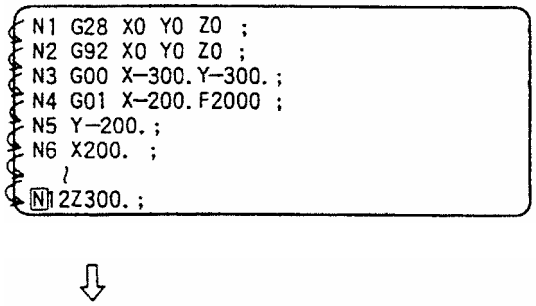
[Data Display Update (One Line Scroll)]

Data display on the screen can be updated in line units by using the [↑] or [↓] key.
If the [↓] key is pressed when the cursor is placed at the screen bottom or if the [↑] key is pressed when the cursor is placed at the screen top, display is scrolled one line.

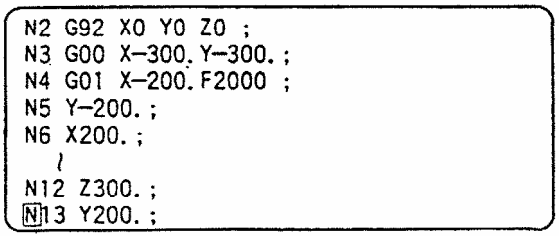


The cursor is moved downward each time the [↓] key is pressed.

1) Whenever the key is pressed, the cursor is moved downward one line.



2) If the key is pressed when the cursor reaches the screen bottom, display data is scrolled up one line. The cursor remains at the screen bottom.
3) If the key is furthermore pressed, the display data is scrolled up one line and new data is displayed at the screen bottom.
4) In contrast, whenever the [↑] key is pressed, the cursor is moved upward one line. If the key is pressed when the cursor reaches the screen top, the display data is scrolled down one line and the previous block data is displayed at the screen top.



2.5.4.2 Screen Editing

[Data Change]

A machining program can always be edited unless it is run in memory mode.
For example, when the data to be edited is displayed as shown in the right, let's try to change the

N7 X123.; block to

↓

G03 Y200. J100.;

(1)

Move the cursor to the data to be replaced.

EDIT (REP.)
N1 G28 X0 Y0 Z0;
:
N7 X123.;
M02;
%

(2)

Set new data
G03 Y200. J100. ;

N7 G03Y200.J100.;
M02;
% EDITING

1) Each time a character is set the cursor is automatically moved one column to the right.
2) When data is entered by using the keys, the message EDITING is displayed.
Note that if the cursor reaches the end of editing area (left side area for the small-character mode), the keys are ignored and the message as “EDIT (CAN’T REPLACE)” appears.

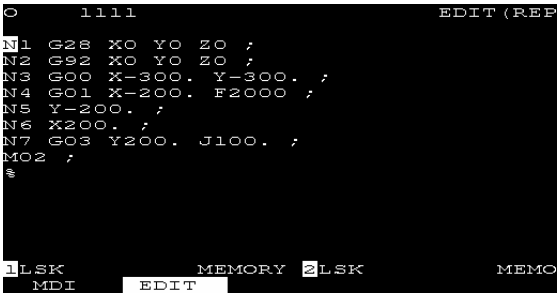
(3) After completion of correction, press the [INPUT] key.

N7 G03 Y200. J100.;
M02;
%

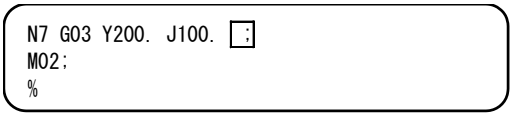
1) The new data is written into memory.
2) The new data is also displayed with each work being both preceded and followed by space code.
3) The cursor is moved to the top of the block on the screen.
4) When the data has been written into memory, the EDITING message disappears.

[Data Insertion]

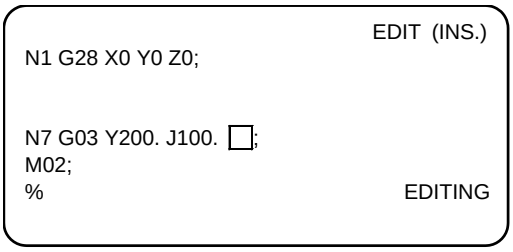
For example, let's try to insert data F500 in the block N7 G03 Y200. J100.;.



(1) Move the cursor to the character following the position in which the data is to be inserted.

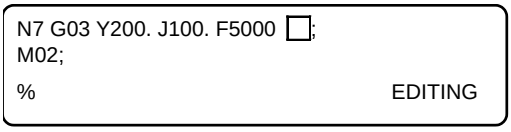


(2) Press the [INS] key.



1) The characters to the right of the cursor are moved to the right.
   The EDITTING message appears and editing state turns “EDIT (INS.)”.
2) Data can be inserted in the position indicated by the cursor.
3) When there is no space in the editing area (the left half of the display area) after the cursor, no more data can be inserted. The message "EDIT(CAN'T INSERT)" appears and the input keys are ignored.

(3) Insert the data.
F5000



1) When the key for the character to be inserted is pressed, the character is set in the position indicated by the cursor.
2) Each time one character is inserted, the cursor is automatically moved one column to the right and the characters to the right of the cursor are also moved to the right.
3) Any number of characters can be consecutively inserted by repeating 1) and 2) above. However, when there is no space to the right of the cursor on the screen, no more data can be inserted.
   The input keys are ignored and the message as “EDIT (CAN’T INSERT)” appears.
4) When a cursor is used while inserting operation, replace mode is re-entered and the editing state turns “EDIT (REP.)”.

(4) After completion of correction, press the [INPUT] key.

1) The new data is written into memory.
2) The new data is also displayed with each word being both preceded and followed by space.
3) The cursor is moved to the top of the block corrected on the screen.
4) When the data has been written into memory, the EDITING message disappears. The editing state turns “EDIT (REP.)”

```
                                        EDIT (REP.)
N1 G28 X0 Y0 Z0;
        :
N7 G03 Y200. J100. F5000 ;
M02;
%
```

[Deletion of One Character]

For example, let's try to delete the character “0” to change F5000 in the block N7 G03 Y200. J100. F5000.; to F500.

(1) Move the cursor to the position of the character to be deleted.

N7 G03 Y200. J100. F5000 ;
M02;
%

(2) Press the [DEL] key.

EDIT (REP.)
N1 G28 X0 Y0 Z0;
~
N7 G03 Y200. J100. F500 ;
M02;
% EDITING

1) The character 0 is deleted.
2) The cursor is automatically moved one column to the right and the message EDITING is displayed.

(3) After completion of correction, press the [INPUT] key.

N7 G03 Y200. J100. F500;
M02;

%

1) The new data is written into memory.
2) The characters to the right of the deleted character, (; in this case) are moved to the left.
3) The cursor is moved to the top of the on the screen.
4) When the data has been written into memory, the EDITING message disappears.

[Deletion of One Block]

For example, let's try to delete the entire block " N7 G03 Y200. J100. F500 ;".

(1) Move the cursor to the position of the block to be deleted. ☞

```
N6 X200.;
N7 G03 Y200. J100. F500;
M2;
%
```

(2) Press the [C.B CAN] key. ☞

1) Data in the entire block is deleted and the message EDITING is displayed.

```
                                    EDIT (REP.)
N1 G28 X0 Y0 Z0;
        :
N6 X200.;

M02;
%                                   EDITING
```

(3) After completion of correction, press the [INPUT] key. ☞

1) The data after correction is written into the memory.
2) The blocks following the deleted data block (M02; and % in this case) are moved forward for display.
3) The cursor is displayed at the top of the screen.
4) When the data in the block has been deleted from memory, the EDITING message disappears.
5) Even if a whole of the block is not viewed in the editing area, this block will be deleted.

```
N6 X200.;
M02;
%
```

[Deletion of Data on One Screen]

For example, assume that data is displayed as shown in the right. Let's try to delete all blocks (sequence numbers 1 to 12) displayed on the screen.

(1) Press the [SHIFT][C.B CAN] key. ☞

1) The full screen becomes blank.
(The left side area for the small-character mode)
2) The cursor to the upper left corner of the screen and the message EDITING is displayed.

(2) Press the [INPUT] key. ☞

1) The data displayed on the entire screen is deleted from memory.
2) Display is started at the data following the deleted data.
3) When the data has been deleted from memory, the EDITING message disappears.
4) When a block is viewed from halfway at the top line of the editing area, only the viewed part will be deleted and EOB will be added automatically.
5) When a whole of the block is not viewed at the bottom of the editing area, only the viewed part will be deleted.

2.5.4.3 Word Editing

[Foreground/background Edit]

"Background edit mode" and "Foreground edit mode" are available for the edit mode.

In the background edit mode

(1) The background edit mode is valid from after BG SRH is pressed to the BG END is pressed.
(2) “BACK GROUND EDITING” is displayed on the screen.
(3) Program indexing is carried out if INPUT key is pressed during the background editing.
(4) Program edit is valid while a program is executed if the program to edit is not running.
(5) Background edit mode will terminate if “COM.SRH” is operated from the edit screen during the background edit.
(6) Background edit mode will not terminate if a program, which is not for the background edit, is externally searched, search&start, or operation searched from other than edit screen during the background edit. However, the background edit mode will terminate if it is the program for the background edit.

(Note) BG SRH cannot be operated for the program if it is the target for COM. SRH or in operation. (The error message "E190 FORE EDITING" will be displayed.)

In the foreground edit mode

1) The foreground editing mode is a status where the program display during operation signal (Y72C) is turned OFF, and the background editing mode is invalid.
2) When the system is not running (operation stopped), the edit cursor successively moves to the various steps being executed in automatic operation.
3) Cursor movement is possible in the foreground editing mode, even in a write-protected status.
4) Machining programs in an operation stop status can be edited in single block mode.
5) The foreground editing mode is entered when the power is turned ON. If there is a program that is already being operation searched, that program will become the foreground editing program.
6) Program indexing is carried out with a reset when not in operation.

(Note1) "EDIT POSSIBLE" is displayed on the screen when editing is possible, "EDIT IMPOSSIBLE" is displayed when editing is not possible.
"EDIT IMPOSSIBLE" is displayed in the fixed cycle mode during feed hold or single block stop.

Other than edit mode in the foreground

When the program display during operation signal (Y72C) is turned ON, the program in operation is displayed on the left side of the screen.

⚠ CAUTION

⚠ Resetting (search) the operation start position is operated when operation signal (Y72C) is turned ON (terminate the foreground edit mode).

If the cursor for edit is placed at the head of the block, that block is the operation start position. If it is placed at other than the head of the block, the head of the machining program is the operation start position. Always make sure the operation start position before starting up automatically.

[Handling of Various Keys during Word Editing]

Various keys during word editing

Key data	Edit area (Left side)	Edit buffer (Right side)	Contents
Cursor key (↑,↓,←,→)	○	×	→ : Move the cursor to the next word in the forward direction. ← : Move the cursor to the next word in the backward direction. ↓ : Move the cursor to the first word of the next block.。 ↑ : Move the cursor to the first word of the previous block.。
BACK NEXT	○	×	NEXT : This key changes the screen to the next page in one screen units, and moves the cursor to the first word. BACK : This key changes the screen to the previous page in one screen units, and moves the cursor to the first word.
C.B	○	×	The same as the menu key [DELETE].
INS	○	×	The same as the menu key [INSERT].
Alphabetical character, numeric symbol keys (0 to 9, A to Z etc.)	×	○	Enter characters in the edit buffer/search data. The edit buffer/search data is cleared as the alphabetic, numeric, or symbol key are entered.
DEL	×	○	Delete the last character entered in the edit buffer and search data. (This key functions the same as the Back Space key.)
CAN	×	×	Invalid
INPUT	○	×	When the cursor is at the head of the block: That block is searched. The operation starts from the designated block. When the cursor is at a position besides the head of the program block: The top of the program is searched. Operation starts from the head of the program.

(Note) Since there is no key equivalent of [REPLACE], use the menu key for replace operation.

Word character judgement method

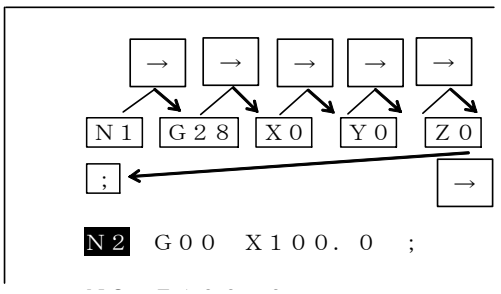
(1) When one of the following character is used for the first character, it is handled as a word.
A to Z
( ) , # / ! % ; [ ]

(2) Macro statement is handled as word characters.
Macro statement

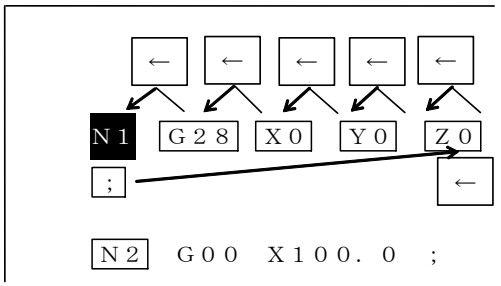
ABS	DPRINT	IOF	RND
ACOS	ELSE	ION	ROUND
AND	END	LE	SETVN
ASIN	EQ	LN	SIN
ATAN	EXP	LT	SQR
ATN	FIX	MOD	SQRT
BCD	FUP	NE	TAN
BIN	GE	OR	THEN
BPRINT	GOTO	PCLOS	WHILE
COS	GT	POPEN	XOR
DO	IF	READB	

[Search by word unit]

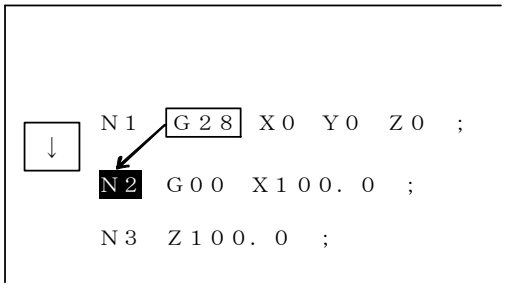
(1) [→] key
Move the cursor in the forward direction.



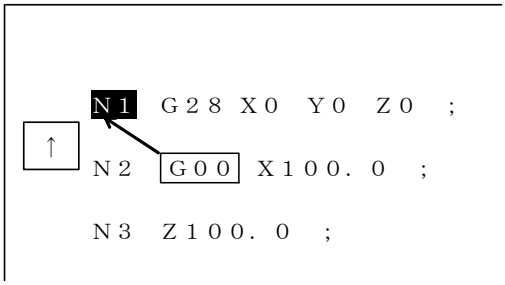
(2) [←] key
Move the cursor in the backward direction.



(3) [↓] key
Move the cursor to the first word of the next block.



(4) [↑] key
Move the cursor to the first word of the previous block.



[Word search]

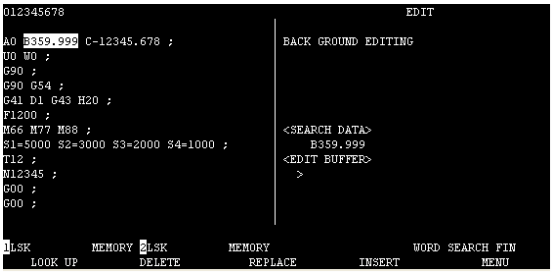
The word matching the search data is searched, and the cursor moves to that word. (The search data is not cleared.)

Use [WORD↑] and [WORD↓] keys for word search.

(1) Press the menu [LOOK UP]
The cursor moves to <SEARCH DATA>. (Edit data will not be cleared.)

(2) Enter the keys for the word data to be searched.
(Example) B359.99
Up to 11 characters can be designated for the search data.

(3) Select upward search or downward search
(Example) [WORD↓]



(a) The message "SEARCH EXECUTION" appears during the search.
(b) The word matching to the search data is searched rather than the word highlighted with the cursor.
(c) The cursor moves to the searched word.
(d) The search data is not cleared.
(e) The screen returns to the 1st menu after the search is completed. (The search data is not cleared.)
(f) The screen returns to the 1st menu if the menu key [RETURN] is pressed.

(Note 1) The message "NO CHARACTERS" appears on the screen if the word cannot be found in the designated direction. If the same menu key is pressed, the search starts again from the top for the downward search and from the end of the program for the upward search.
(Note 2) If a word character is entered for the search data after the menu key is pressed, the search buffer is cleared and then the character is entered.
(Note 3) The search data is valid until ";" (EOB). Only one block can be searched at a time.
(Note 4) The [C.B] and [INS] keys are invalid for the search menu.

[Character string search]

The same character string as the search data is searched and the cursor will move to that word.(The search data will not be cleared.)

Use [STR.↑] and [STR.↓] keys for character string search.

(1) Press the menu [LOOK UP]

The cursor moves to <SEARCH DATA>.

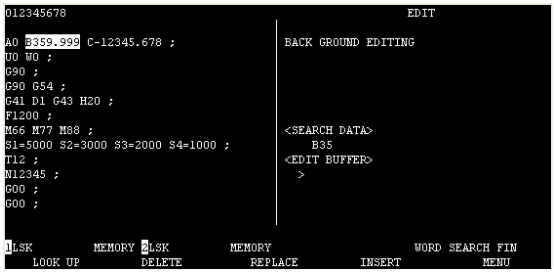
(2) Enter the keys for the word data to be searched.

(Example) B35

Up to 11 characters can be designated for the search data.

(3) Select upward search or downward search.

(Example) [STR.↓]



(a) The message "SEARCH EXECUTION" appears during the search.
(b) The character string relevant to the search data is searched rather than the word highlighted with the cursor.
(c) The cursor moves to the searched word.
(d) The search data is not cleared.
(e) The screen returns to the 1st menu after the search is completed.
(f) The screen returns to the 1st menu if the menu key [RETURN] is pressed.

(Note 1) The message "NO CHARACTERS" appears on the screen if the word cannot be found in the designated direction. If the same menu key is pressed, the search starts again from the top for the downward search and from the end of the program for the upward search.

(Note 2) Matching is checked within the No. of designated character strings, regardless of the character strings before and after the ones designated. For example, even if G2 is designated, the character strings such as G20 to G29, G200 onward, etc., become search targets.

(Note 3) During the character string search, macro statements are not handled as one word data, so the operation differs from the usual character string search. For example, if "GO" is designated for "GOTO" and a character string search is executed, the cursor is placed at [GOTO].

(Note 4) If a word character is entered for the search data after the menu key is pressed, the search buffer is cleared and then the character is entered.

(Note 5) The search data is valid until ";" (EOB). Only one block can be searched at a time.

(Note 6) The [C.B] and [INS] keys are invalid for the search menu.

[Deleting word]

The word where the cursor is placed can be deleted.

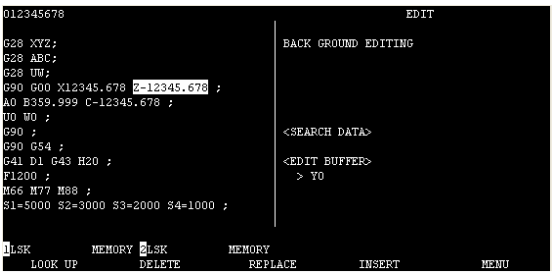
(1) Search the word to be deleted. ☞
[→][←][↑][↓]
[BACK][NEXT]

(2) Press the menu [DELETE]. ☞

(a) The word where the cursor is placed is deleted.

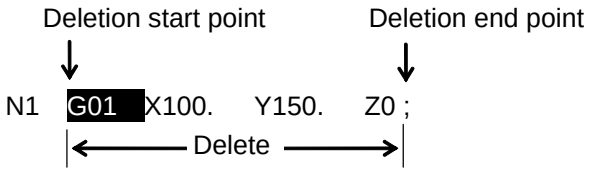
(b) The cursor is moved to the next word.

(c) Deleted word is set to the <EDIT BUFFER>.

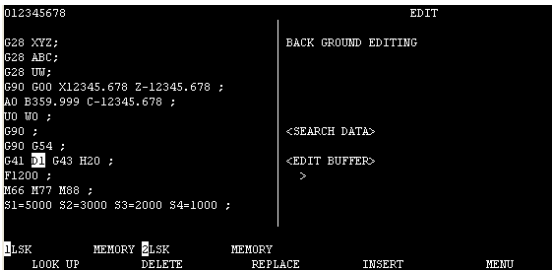


[Deleting a line]

Delete the data from where the cursor is currently placed to [EOB] (;).

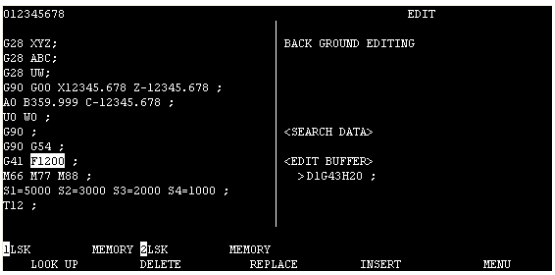


(1) Search the first word of the line to be deleted. ☞
[→][←][↑][↓]
[BACK][NEXT]



(2) Enter [EOB] key.

(3) Press the menu [DELETE]. ☞
(a) The data from the word where the cursor is placed to EOB (;) is deleted.
(b) The cursor moves to the first word of the next page.
(c) Deleted word is set to the <EDIT BUFFER>.

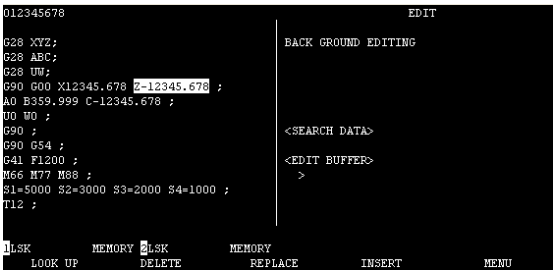


(Note 1) Only the EOB (;) key entered in the EDIT BUFFER area is valid.
(Note 2) Up to 96 characters, starting from the first word, of the deleted line are set in <EDIT BUFFER>.
(Note 3) After the line is deleted, up to 96 characters can be added in the <EDIT BUFFER>.

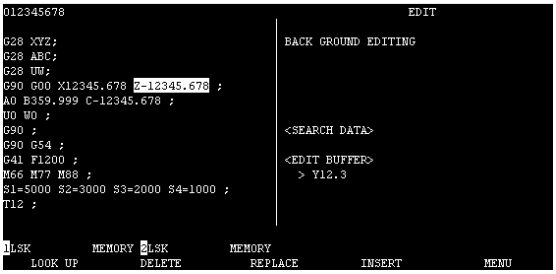
[Replacing words]

The word where the cursor is placed can be replaced with the edit buffer data.
(Edit buffer data will not be cleared.)

(1) Search the word to be replaced. ☞
[→][←][↑][↓]
[BACK][NEXT]



(2) Enter the word to be replaced to <EDIT BUFFER>. ☞
(Example) Y12.3

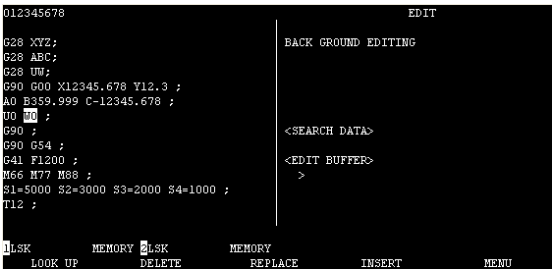


(3) Press the menu [REPLACE]. ☞
(a) The word where the cursor is placed is replaced with the word in <EDIT BUFFER>.
(b) The cursor is placed in the replaced word.
(c) The edit buffer will not be cleared.

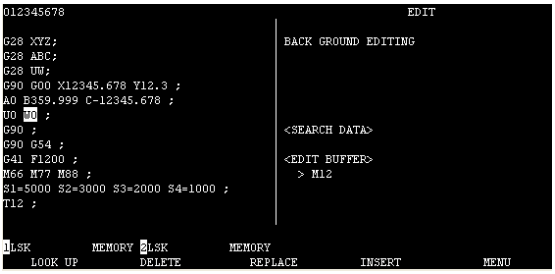
[Inserting words]

When inserting a word immediately after the word where the cursor is placed.
(The edit buffer data will not be cleared.)

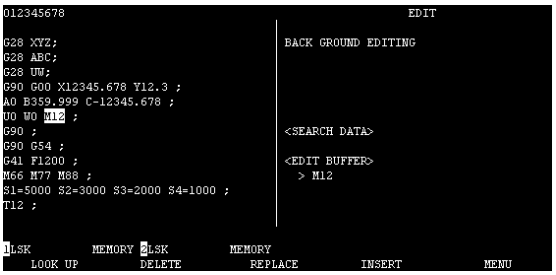
(1) Search the word immediately before insertion point. ☞
[→][←][↑][↓]
[BACK][NEXT]



(2) Enter the word to be inserted in <EDIT BUFFER>. ☞
(Example) M12

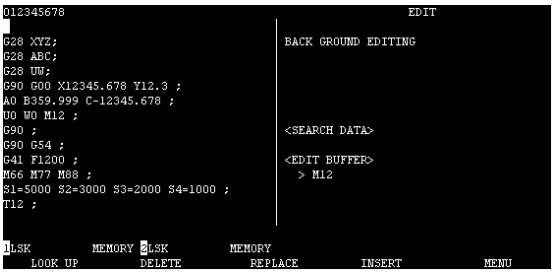


(3) Press the menu [INSERT]. ☞
(a) The edit buffer data will be inserted after a word where the cursor is placed.
(b) The cursor moves to the inserted word.
(c) The edit buffer data will not be cleared.

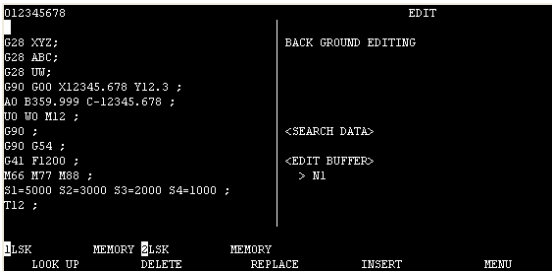


When inserting to the first word
Adding a function to insert a word before the first word of the program.
(The edit buffer data will not be cleared.)

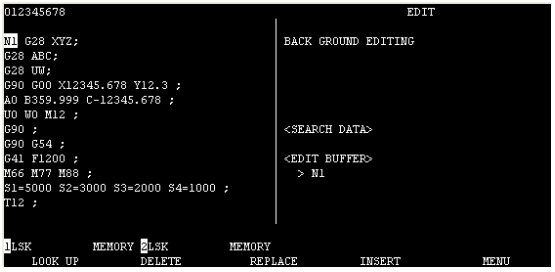
(1) Move the block cursor to the empty line of the head of the program.
[→][←][↑][↓]
[BACK][NEXT]
With this state, deleting and replacing operation are ignored. When [NEXT] is pressed, word insertion function will be cancelled.



(2) Enter the word to be inserted in <EDIT BUFFER>.
(Example) N1

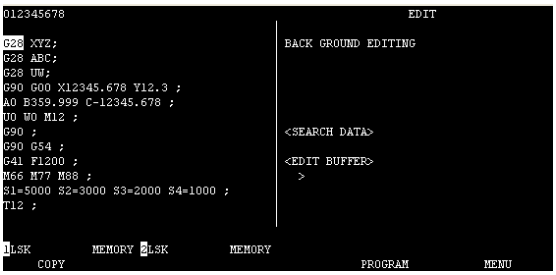


(3) Press the menu [INSERT].
(a) The edit buffer data will be inserted as the first word of the program.
(b) The cursor moves to the inserted word.
(c) The edit buffer data will not be cleared.

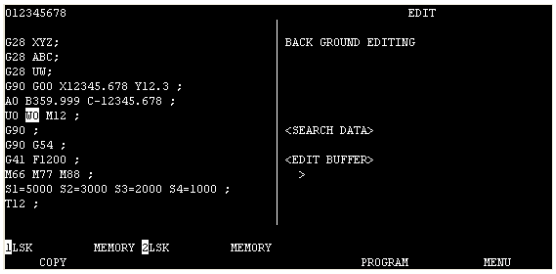


[Copying words]

The word where the cursor is placed can be copied to the <EDIT BUFFER>.



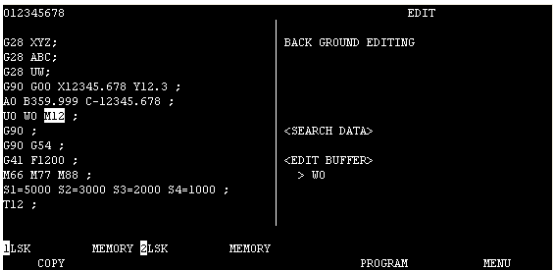
(1) Search the word to be copid. ☞
[→][←][↑][↓]
[BACK][NEXT]



(2) Press the menu [COPY]. ☞

(a) The word where the cursor is place is set to <EDIT BUFFER>.

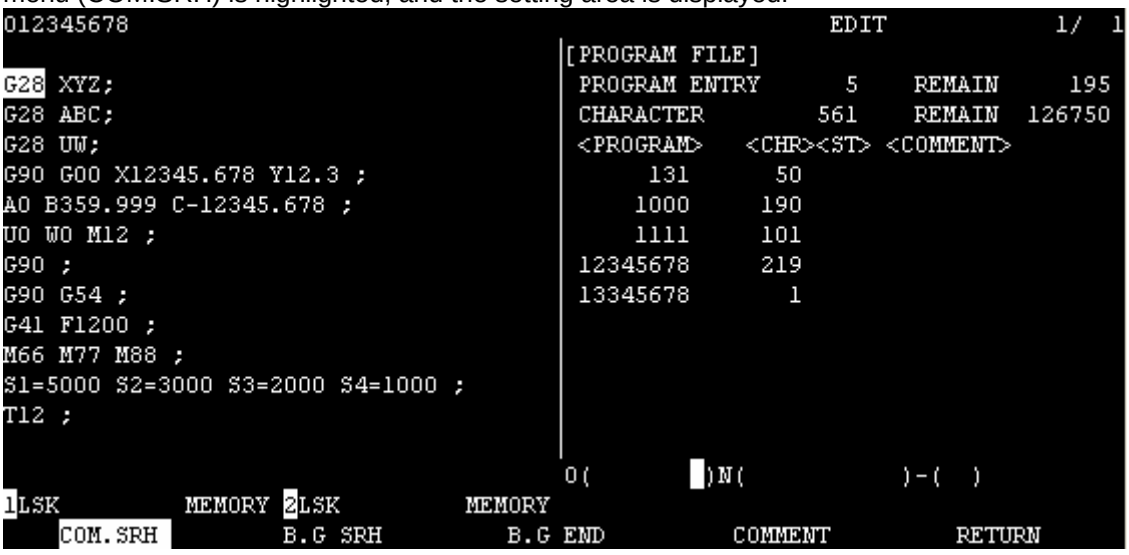
(b) The cursor moves to the next word.



(Note 1) "%" will not be copied.

[Program display]

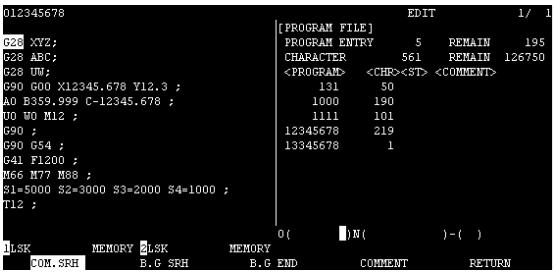
When the menu key [PROGRAM] is pressed, the searched program appears on the left side of the screen, and a list of programs registered in the memory appears on the right side of the screen. The operation search menu (COM.SRH) is highlighted, and the setting area is displayed.



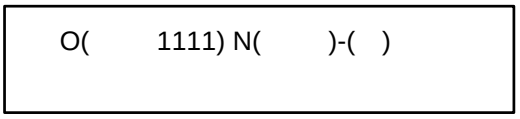
Display items	Contents
O12345678	Display the searched program No.
G28 XYZ; G28ABC; . . .	Display the searched program.
PROGRAM ENTRY and REMAIN	The No. of programs already registered as userd machining programs appears in the PROGRAM ENTRY column. The remaining No. of programs that can be registered appears in the REMAIN column. The total of "PROGRAM ENTRY" and "REMAIN" is the maximum No. of programs. This figure is determined by the specifications.
CHARACTER and REMAIN	The No. of characters already registered as user machining programs appears in the CHARACTER column. The remaining No. of characters that can be registered appears in the REMAIN column. The value in the REMAIN column is displayed in 250-character units.
<PROGRAM>	This item shows the Nos. of the machining programs already registered. The Nos. are displayed in order from the smallest No. The display range is from 1 to 99999999 and displayed in ascending order.
<CHR>	This item shows the No. of characters registered in the memory for the corresponding machining program No
<ST>	Not used.
<COMMENT>	An outline of the machining program functions, specifications, applications, etc., can be displayed as a comment in this item. The comment can be set with up to 18 alphanumeric or symbol characters.

[Deleting a program]

A program to execute automatic operation can be deleted from the machining programs registered in the memory.

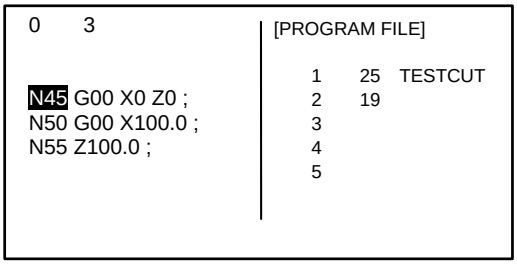


(1) Set the program No. to be deleted.
(Example)
O(1111) N( )−( )

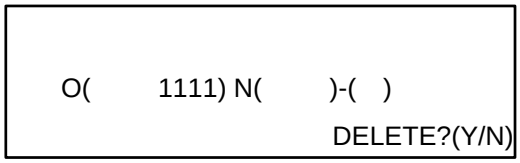


(2) Press [C.B].

(a) 1) [PROGRAM FILE] will be updated.
2) "DELETE? (Y/N)" is displayed.



(b) "DELETE? (Y/N)" is displayed.



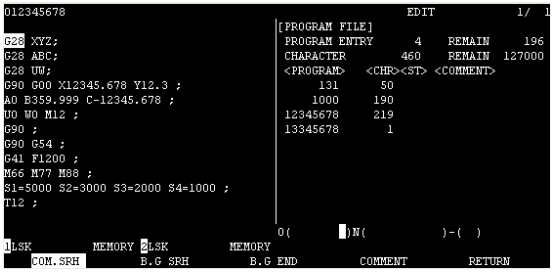
(3) Press [Y].

(4) Press [INPUT] to delete.

(a) Start deleting.

(b) When the designated program No. is found, that program will be deleted.

(c) Press [RETURN] to go back to the word edit screen (the first menu).



(Note 1) If the data is found in "N ( ) - ( )" when deleting the program, these values are ignored.

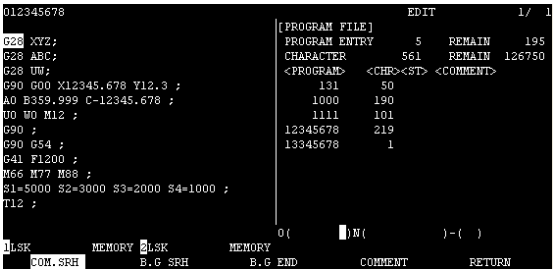
(Note 2) When the sub-program which is called from the main program currently being executed is designated, it can be deleted as long as it is not running. However the executing program's operation is not guaranteed. An error will occur if a running sub-program is designated to delete.

(Note 3) Batch deletion for the setting area is not possible.

(Note 4) When a program is deleted, [PROGRAM FILE] will be updated. But the program display column will not be update.

[Newly creating programs]

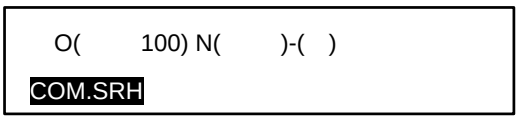
Programs to execute automatic operation can be created and registered in the memory.



(1) Set the program No. to be registered. ☞

(Example)

O(100) N( ) - ( )

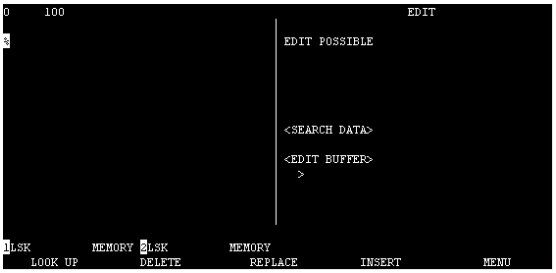


(2) Press [INS]. ☞

(a) [PROGRAM FILE] will be updated.

(b) Program creation will be enabled.

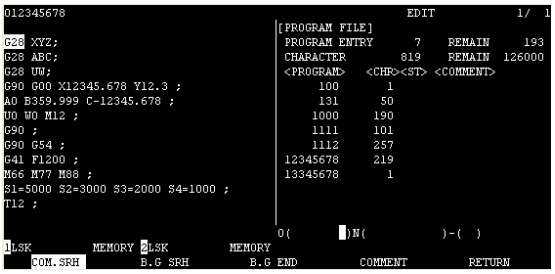
(c) Press [RETURN] to go back to the word edit screen (the first menu).



(Note 1) If the data is found in "N ( ) - ( )" when deleting the program, these values are ignored.
(Note 2) Only when [COM.SRH] is highlighted, [INS] key is valid.
(Note 3) If the set program No. is already registered, the operation search will be started.

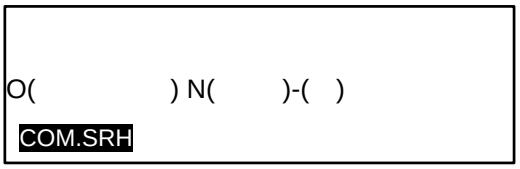
[Operation search]
Calling a program

The program Nos., sequence Nos., and block Nos. for executing automatic operation can be called from the machining programs registered in the memory.



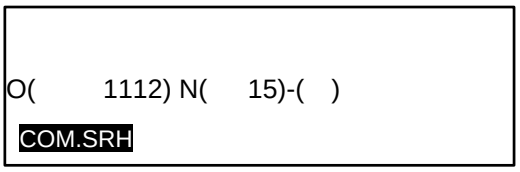
(1) Press the menu [COM.SRH].
The setting area for "COM.SRH" will be displayed.

☞



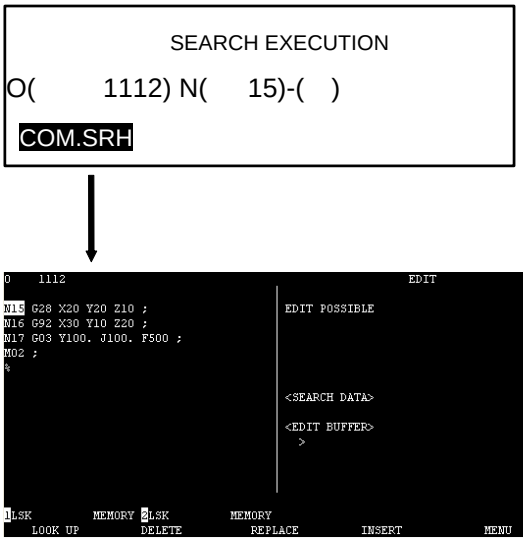
(2) Set the program No. to be called. Set the sequence No. and block No. if neccessary.
(Example)
O(1112) N(15)－( )

☞



(3) Press [INPUT].
(a) Start the search.
(b) The program No. is displayed if the designated program No., sequence No. or block No. is found, and goes back to the word edit screen (the first menu).

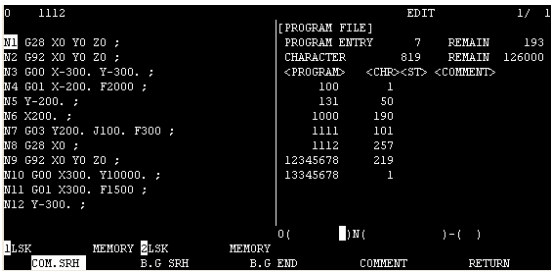
☞



(Note 1) A program will be deleted if the program No. is entered and [C.B] is pressed when the setting area is displayed.

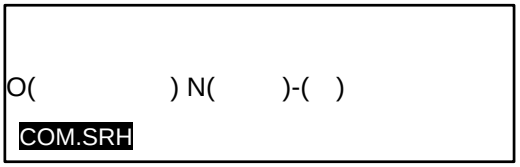
Sequence No./block No. search

Sequence No. or block No. can be searched when only sequence No. or sequence No. and block No. is entered without entering O No.

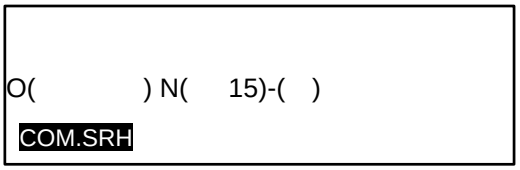


(1) Press the menu [COM.SRH].

The setting area for "COM.SRH" will be displayed.



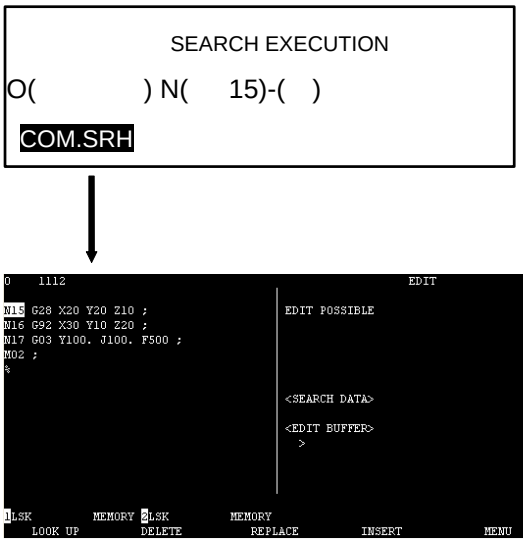
(2) Enter the sequence No. to be searched.
(Example)
O( ) N(15)－( )



(3) Press [INPUT].

(a) Start the search.

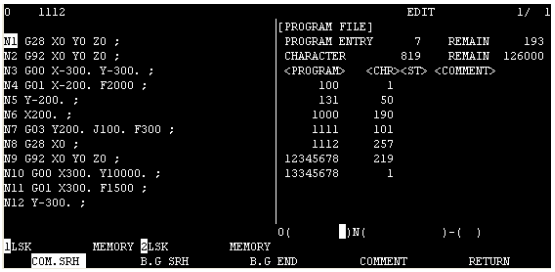
(b) The program is displayed if the sequence No. or block No. is found, and goes back to the word edit screen (the first menu).



(Note 1) If the designated sequence No. is not found at the editing cursor position or below the executed block, the message "E13 NB NOT FOUND" will be displayed.
When [INPUT] is pressed again, the designated sequence No. or block No. will be searched from the head of the program.

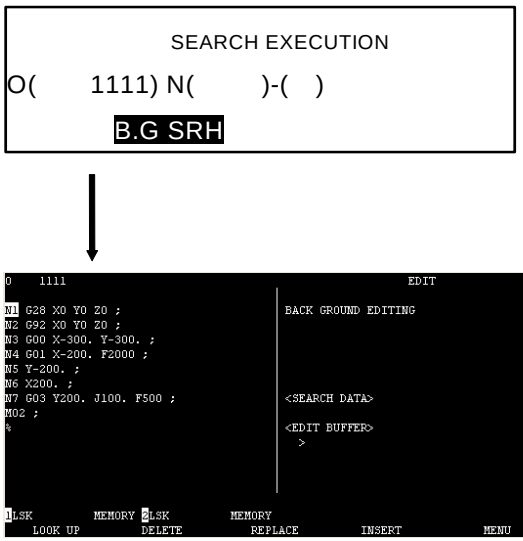
[B.G search]
Calling a program

The program No., sequence No., and block No. to be edited can be called to execute background editing. New machining program can be registered if a program No. not registered in the memory is set.



(1) Set the program No. for. the background edit.
(Example)
O(1111) N( )－( )

(3) Press [INPUT].
(a) The message "SEARCH EXECUTION" appears during the search.
(b) The designated program is displayed and goes back to the word edit screen (the first menu).
If the designated program No. does not exist, new program will be created.



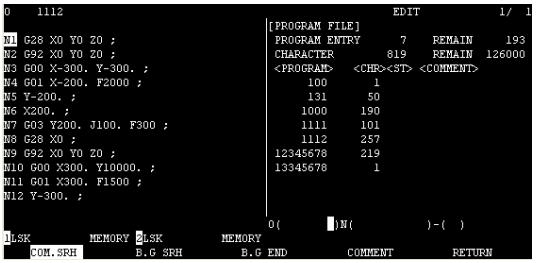
(Note 1) A program will be deleted if the program No. is entered and [C.B] is pressed when the setting area is displayed.

[B.G END]

The "B.G END" menu is used to quit editing after executing the background editing.
If a running program is displayed on the EDIT screen, changeover to the running program will not occur unless the [B.G END] menu is pressed and the background editing is canceled.
(The button does not have to be pressed even when quitting the background editing if the program in operation is not displayed on EDIT screen.)

[Comment setting]

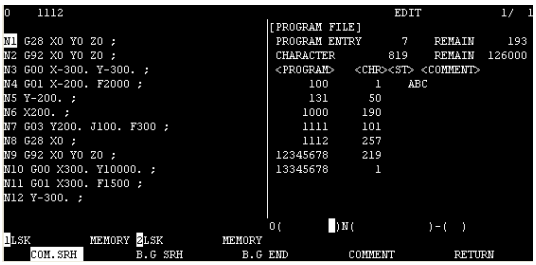
An outline of the machining program functions, specifications, applications, etc., can be set as a comment.。



(1) Press the menu [COMMENT].
The setting area for "COMMENT" will be displayed.

(2) Set the program No. that the comment to be set.
(Example)
O(100) COMMENT (ABC)

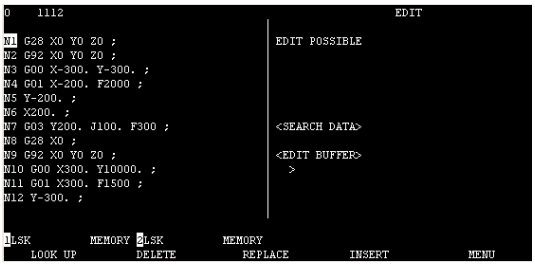
(3) Press [INPUT].
(a) Set the comment for the designated program No.
But if the designated program No. is not displayed on the screen, that program's screen is displayed. Press [INPUT] again.



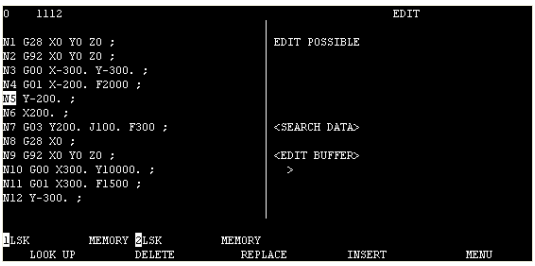
(Note 1) A program will be deleted if the program No. is entered and [C.B] is pressed when the setting area is displayed.

[Setting the program operation start position]

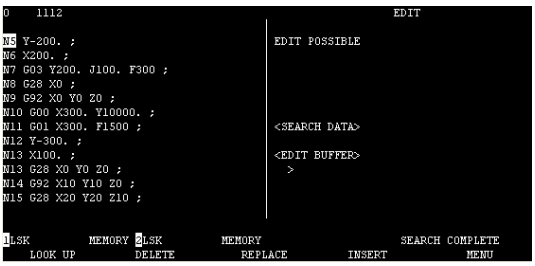
After setting a program for memory operation, the start block can be designated if you want to execute from the midway block. The operation start position in normal word editing is set at the head of the program. To change the start position, move the cursor to the head of the block to be started and press [INPUT].



(1) Move the cursor at the head of the block to be started. ☞
[↓][↑]
[BACK][NEXT]



(2) Press [INPUT]. ☞
The message "SEARCH COMPLETE" appears and the operation is enabled from the designated block.



⚠ CAUTION

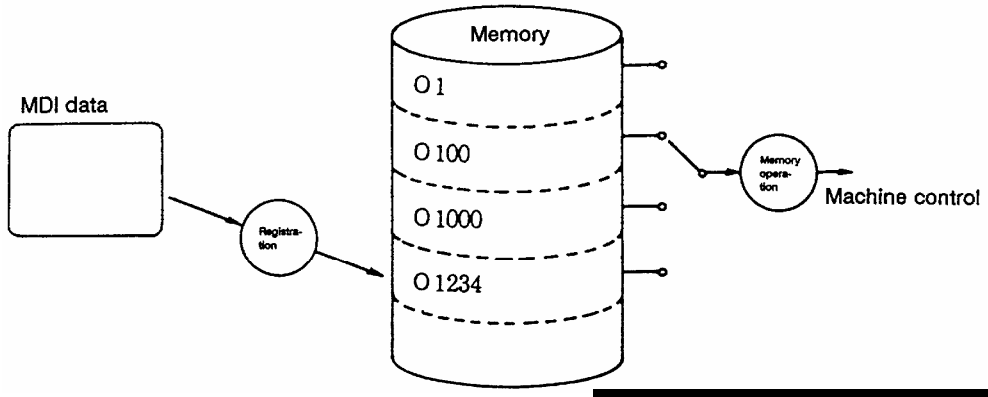
⚠ If an operation start position is set in the halfway of the blocks and the program is started, the program before the start position is set is not executed. If there is a coordinate system shift command, or M, S, T and B commands before the block set as the starting position, execute the necessary command with MDI etc. There is a danger of interference with the machine if the operation is started from the block set as a starting position without executing these operations.

⚠ The head of the program will be searched if [INPUT] is pressed when the cursor is placed at other than the head of the block. The program will be displayed from the head of the program.

2.5.5 MDI Screen Extension Operation

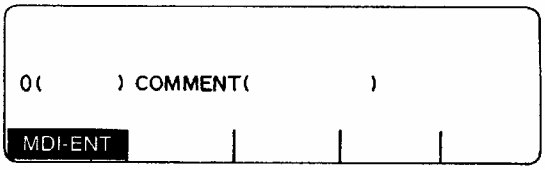
2.5.5.1 MDI Data Registration in Memory

Data set on the MDI screen can be registered in memory. Comments can be added to indicate the contents of the program to be registered.



For example, assume that MDI data is set as shown in the right. The MDI data registration procedure in memory is described below:

(1) Press the menu [MDI-ENT].

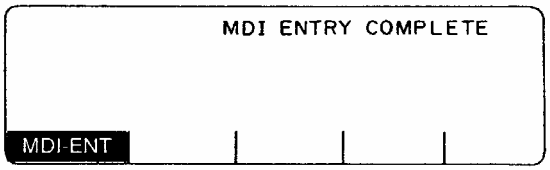


1) The setting area for MDI-ENT is displayed.

(2) Set the registered program number. A comment can also be set at the same time.

(Example)
O (1234) COMMENT ( )

(3) Press the [INPUT] key.



1) If the program has been registered normally into memory, the message "MDI ENTRY COMPLETE" is displayed. The display is cleared from the setting area; the MDI-ENT menu display returns to normal display from the reverse display.

(Note) If preparing comment, space ([SPACE]) can be written in it. But, the space is ignored after registration for efficient use of memory.

2.5.6 Edit Screen Extension Operation

2.5.6.1 Edit Data Call

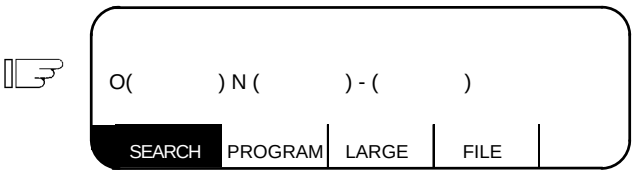
The calling method of the program or block to be edited is explained. The search function is also used to call a separate machining program from the currently running one for background edit. A search can be executed for the program head, character string, and sequence number.

(1) Search for the program head
In the setting field, specify the program number of the program to be called.

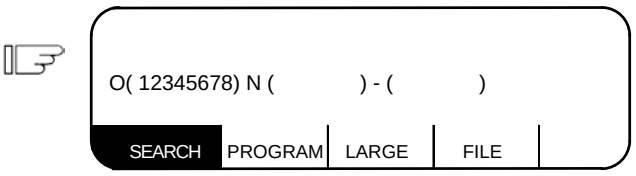
The operating procedure is as follows:

(1) Press the menu [SEARCH].

1) The setting area for SEARCH is displayed.



(2) Set the called program number.
(Example)
O (12345678) N ( ) - ( )



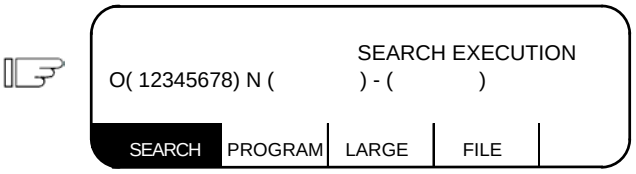
(3) Press the [INPUT] key.

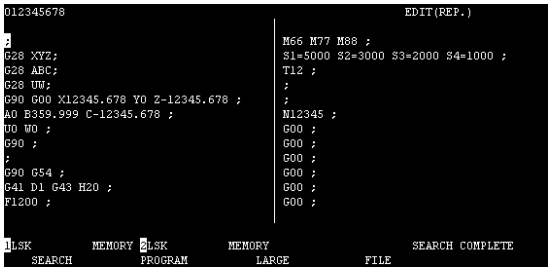
1) The message SEARCH EXECUTION is displayed during searching.

2) The specified program is displayed, beginning with top of the program.

3) The cursor is displayed at the top of the screen.

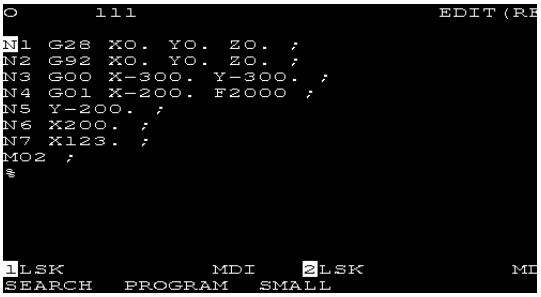
4) At normal completion of program head search, display of the setting area disappears and SEARCH menu display is restored to normal mode from reverse mode.





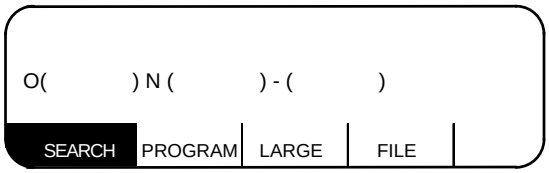
(2) Character string search

The character string search is useful particularly to search the word data to be corrected.
Specify the called program number and character string in the setting area. However, the program number need not be specified if the program already displayed on the screen is searched for a given character string.



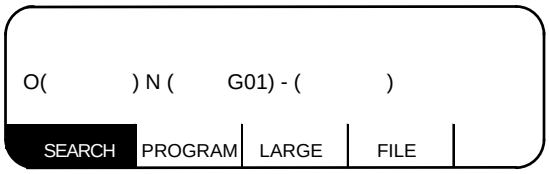
The operation procedure is described below:

(1) Press the menu [SEARCH].

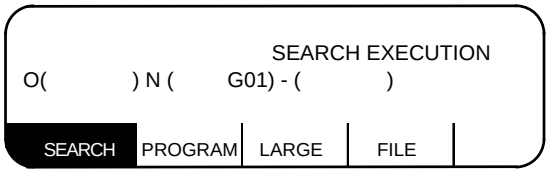


1) The setting area for SEARCH is displayed.

(2) Set the called program number and character string.
(Example)
O ( ) N (G01) - ( )



(3) Press the [INPUT] key.



1) The message SEARCH EXECUTION is displayed during searching.

2) A search for the specified character string is started at the top of the specified program. The program is displayed starting at the block containing the found character string. However, for the program already displayed on the screen, a search for the specified character string is started at the displayed portion.

3) The cursor is displayed at the top of the found character string.

4) At normal completion of character string earch, display of the setting area disappears and SEARCH menu display is restored to normal mode from reverse mode.

(Note 1) When a given character string is not found, a "NO CHARACTERS" message is displayed.
(Note 2) A string of up to 11 characters may be specified.
(Note 3) The specified character string is searched and identified in the specified number of character strings regardless of the preceding and subsequent characters. That is, for example, if G2 is to be searched, G2 of G20 to G29 and G200 and up cannot be classified and will become target character strings.

[Setup example of character string data]

N (N10) → The character string N10 is searched. (N10 and N100 are also searched.)
N (N10 X100.) → The character string N10 X100. is searched.
N (X-01234.567) → The character string X-01234.567 is searched (X-1234.567 is not searched.)
N (EOR) → The character string % (EOR code) is searched.

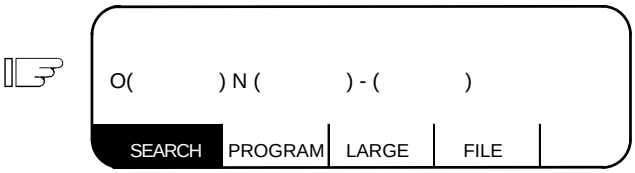
(3) Sequence number, block number search

Specify the called program number, sequence number, and block number in the setting area. If only digits are set in N ( ), a sequence number search is made. (If an alphabetic character or symbol is contained, a character string search is made.) To search the top of a program, specify only the program number. To search an already displayed program on the screen for a given sequence number, program number specification may be omitted.

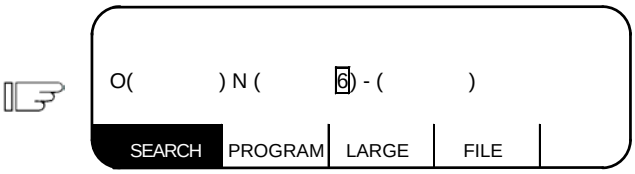
The operation procedure is described below:

(1) Press the menu [SEARCH].

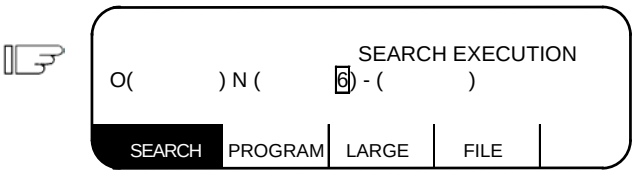
1) The setting area for SEARCH is displayed.



(2) Set the called program number, sequence number, and block number.
(Example)
O ( ) N (6) - ( )



(3) Press the [INPUT] key.



1) The message SEARCH EXECUTION is displayed during searching.

2) A search for a given N number is started at the top of the specified program. The program is displayed starting at the block containing the found N number. However, for the program already displayed on the screen, a search for the specified N number is started at the displayed portion.

3) The cursor is displayed at the top of the found block.

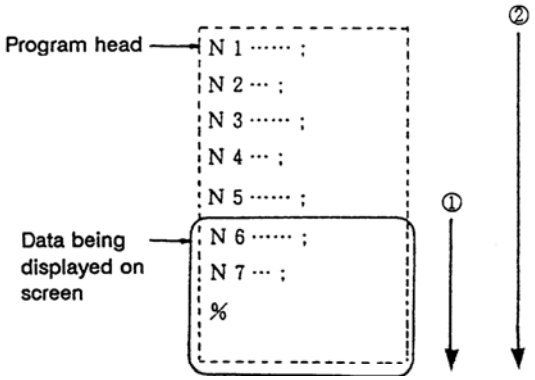
4) At normal completion of search, display of the setting area disappears and SEARCH menu display is restored to normal mode from reverse mode.

(Note 1) When a given N number is not found, an "NB NOT FOUND" message is displayed.
(Note 2) If a given program number is not found, a "PROG NOT FOUND" message is displayed.
(Note 3) The sequence number can be specified in a maximum of five digits.

(4) Action to be taken when the "NO CHARACTERS" or "NB NOT FOUND" error occurs

If a search can be executed for the currently displayed screen, the search starts with the starting block being displayed. If the specified data is not found before the program end (%), the "NO CHARACTERS" or "NB NOT FOUND" occurs. By pressing the [INPUT] key at this time, the search is retried beginning with the program head. If a search is executed for data in a block that is before the currently displayed data, the search will be accomplished by the second search.

(Example)



(Example 1) For search for N4:
(1) First search ... Error "NB NOT FOUND"
(2) Second search ... N4 can be found.

(Example 2) For search for N7:
(1) First search ... N7 can be found.

(Example 3) For search for N8:
(1) First search ... Error "NB NOT FOUND"
(2) Second search ... "NB NOT FOUND"

2.5.6.2 New Program Registration and Preparation

This function is used to prepare a new machining program.
To prepare a machining program on the EDIT screen, first press the menu [PROGRAM] and register the machining program number, then enter the program directly by using the keys.

(1) Press the menu [PROGRAM].

The setting area for PROGRAM is displayed.

(2) Set the new registered program number. A comment can also be set at the same time if necessary.
(Example)
O (1000) COMMENT (TEST)

(3) Press the [INPUT] key.

1) When the program number and comment are registered in memory, they are displayed at the screen top.

2) At the time, only one character of % is automatically registered in memory as data. Thus, the screen as shown in the right is displayed.

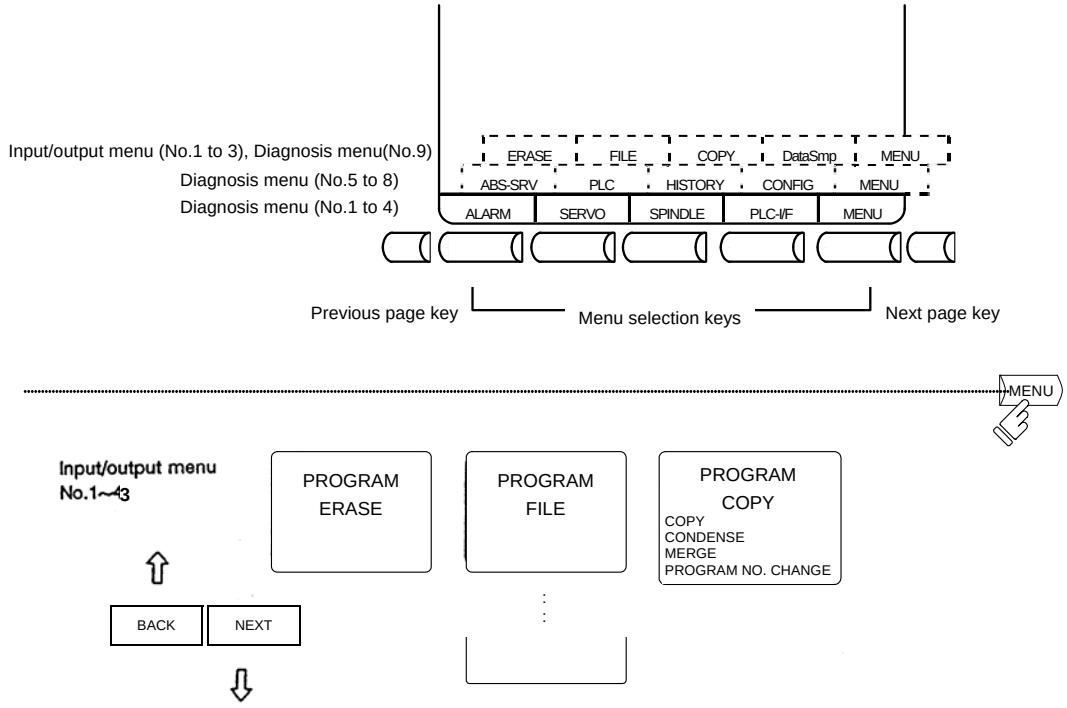
(4) Enter the work program in sequence by using the keys. Key operation is the same as normal program edit operation.

(Note 1) To later edit the work program registered in memory by using this function, also call it by pressing [SEARCH] as with other programs.

(Note 2) If preparing comment, space ([SPACE]) can be written in it. But, the space is ignored after registration for efficient use of memory.

2.6 Data In/Out

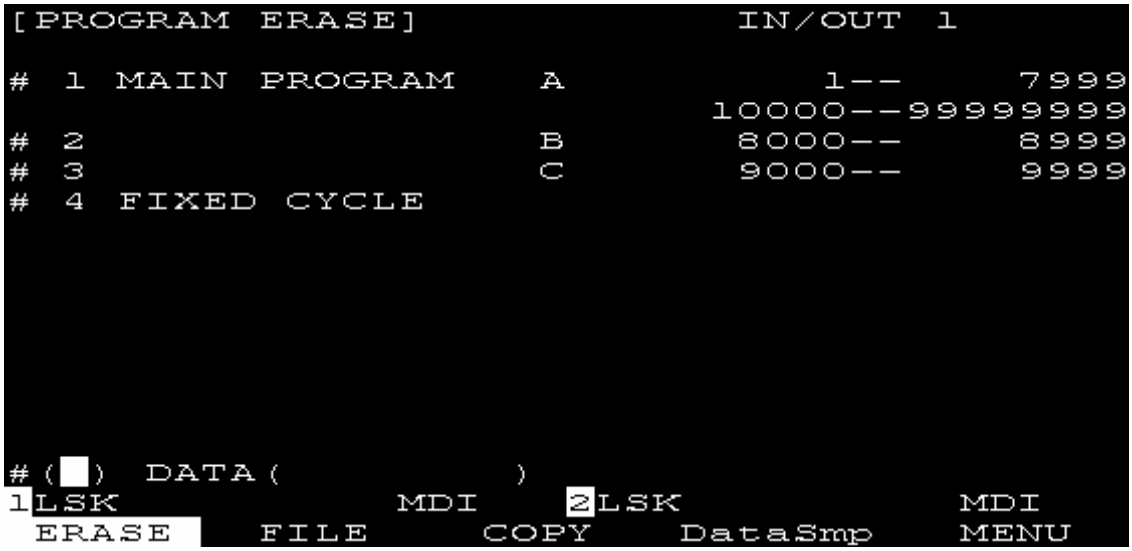
When the function selection key [DIAGN IN/OUT] is pressed, the following menu is displayed:



(Note) Screen transition is not possible while inputting or outputting the data.

2.6.1 Program Erase

When the menu [ERASE] is pressed, the PROGRAM ERASE screen is displayed.
User-prepared work programs (main program and subprogram) can be erased in any desired program number or group units on the PROGRAM ERASE screen.

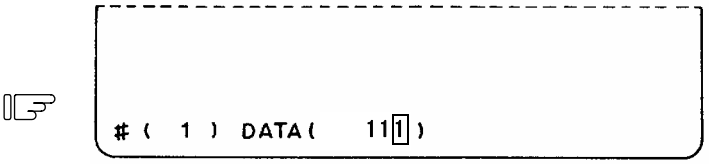


#	Item	Explanation	Data setting range		
			Program number specification	Program group erase	All program erase
1	MAIN PROGRAM <A>	This is specified to erase data in the range of work program numbers 1 to 7999 and 10000 to 99999999.	1 to 7999 and 10000 to 99999999	ALL	CLR
2	MAIN PROGRAM <B>	This is specified to erase data in the range of standard subprogram 8000 to 8999 mainly prepared by the user.	8000 to 8999	ALL	
3	MAIN PROGRAM <C>	This is specified to erase data in the range of custom programs 9000 to 9999 mainly provided by the machine manufacturer.	9000 to 9999	ALL	
4	FIXED CYCLE	Although this is specified to erase a fixed cycle program, normally it cannot be operated. To erase a fixed cycle program, set the parameter. See Appendix 2 “Registering and Editing Fixed Cycle Programs”. When NC is operated without setting the parameter, SETTING ERROR occurs.			

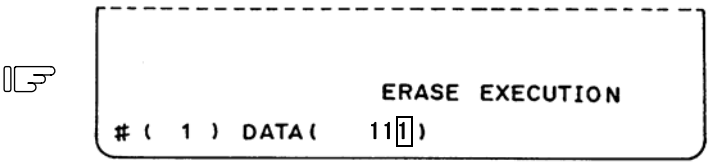
(1) To erase one machining program

To erase a specified machining program from the machining programs registered in memory, perform the following:

Set the data type and program number of the machining program to be erased.
(Example)
To erase machining program O111
(1)
DATA (1 1 1)

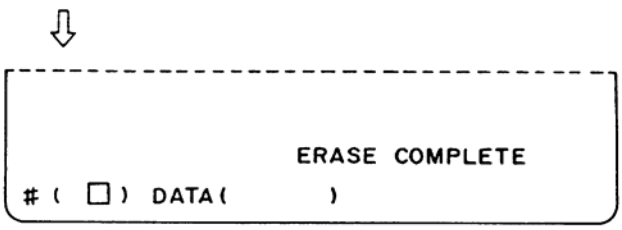


Press the [INPUT] key.

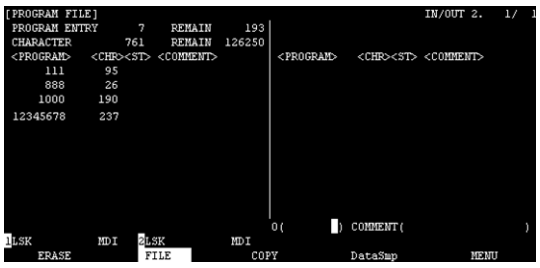


1) Program erase is started and the message ERASE EXECUTION is displayed in the message field. Usually program erasure is terminated in a moment.

2) When the erase is complete, the message ERASE COMPLETE is displayed in the message field and the data setting area becomes blank.

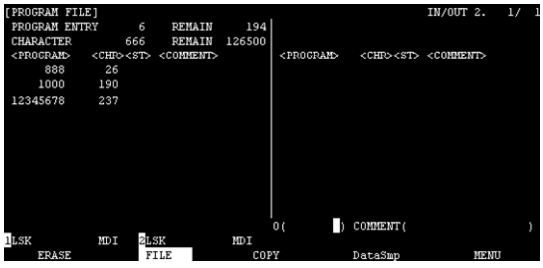


PROGRAM FILE before erase execution



Erase execution

PROGRAM FILE after erase execution



Program O1001 for which the erase function is executed disappears from the PROGRAM FILE screen. The new values are displayed in PROGRAM ENTRY, CHARACTER, and REMAIN.

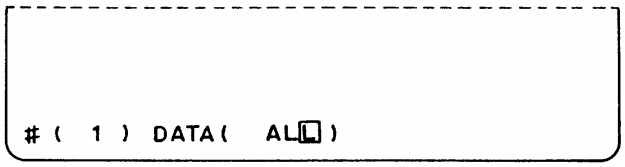
(2) To erase machining program group

To erase any of <1> 1~7999 and 10000~99999999, <2> 8000~8999, and <3> 9000~9999 of the data types of machining programs registered in memory, perform the following:

Set the data type to be erased and "ALL" in DATA ( ).
(Example)
To erase the entire machining program A group

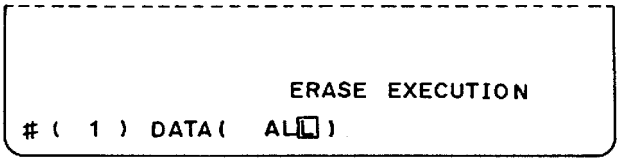
(1) DATA (A L L)

☞

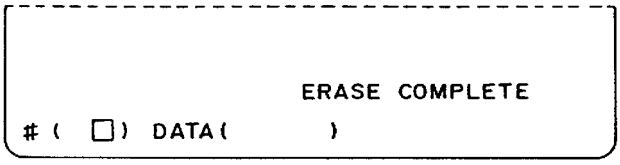


Press the [INPUT] key.

☞

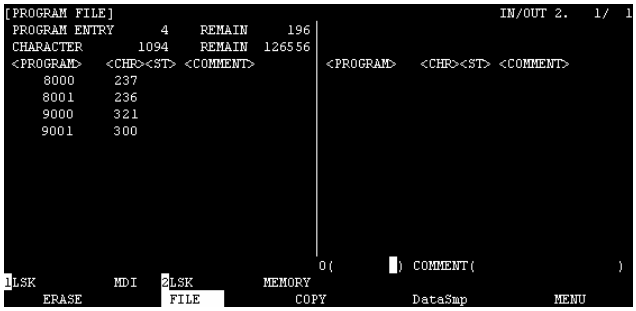


Program erase is executed and terminated in a moment. The message ERASE COMPLETE is displayed in the message field and the data setting area becomes blank.



Check REMAIN, etc. on the PROGRAM FILE screen.

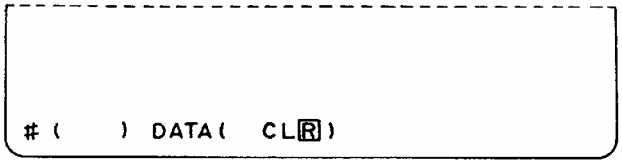
☞



(3) To erase all machining programs registered in memory

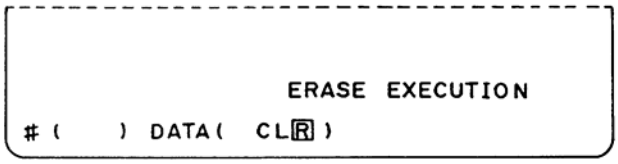
To erase all of machining programs 1~99999999 registered in memory, perform the following:

Set “CLR” in DATA ( ).
( )
DATA (C L R)

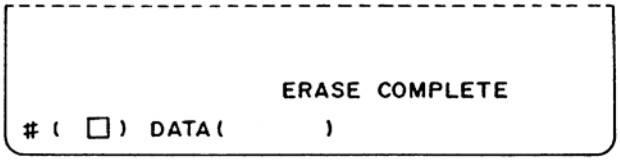


1) No value needs to be entered in # ( ).

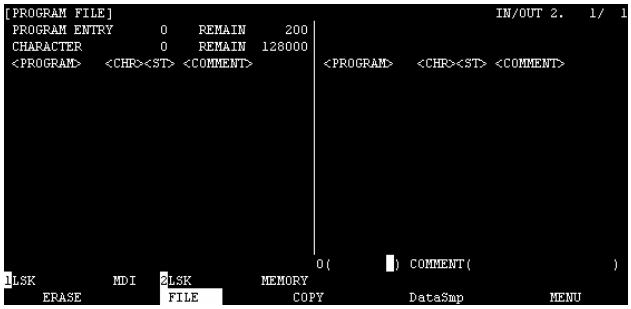
Press the [INPUT] key.



1) All machining program erase is executed and terminated almost in a moment. The message ERASE COMPLETE is displayed in the message field and the data setting area becomes blank.



Check the PROGRAM FILE screen.



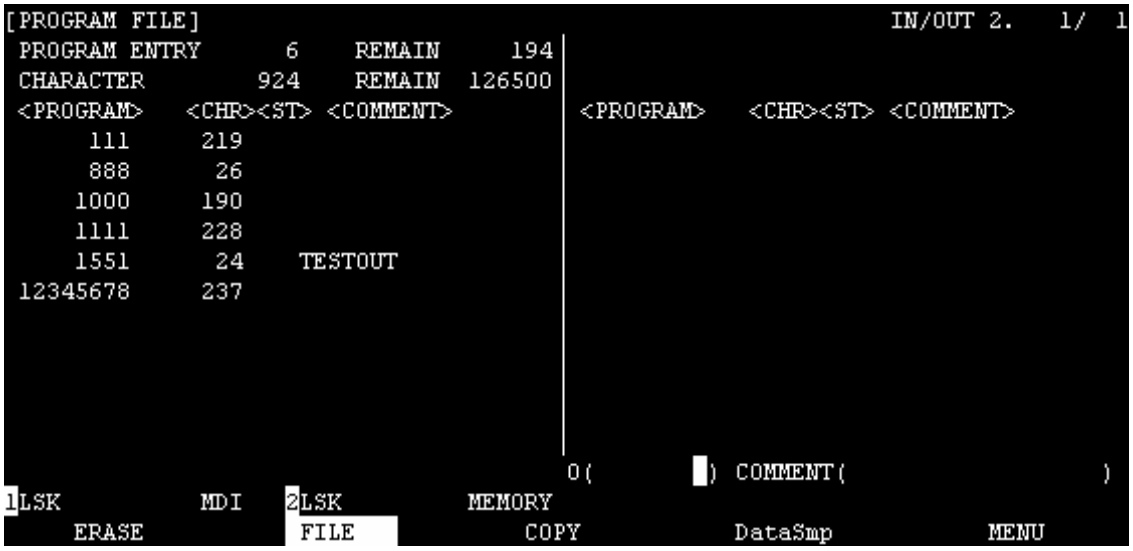
1) PROGRAM ENTRY is set to 0 and REMAIN is set to the maximum number of programs defined in the specifications.
2) CHARACTER is set to 0 and REMAIN is set to the maximum number of stored characters defined in the specifications.
   The number of programs and the number of characters are as listed below according to the specifications:

Tape storage length	Number of programs	Number of characters
40m	64	17500
80m	128	32000
160m	200	64000
320m	200	128000
600m	400	236250
1280m	1000	512000
2560m	1000	1024000
5120m	1000	2048000

3) The PROGRAM FILE screen becomes blank, indicating that no programs are registered.

2.6.2 Program File

When the menu [FILE] is pressed, the PROGRAM FILE screen is displayed.
The PROGRAM FILE screen lists the user-prepared machining programs stored in memory.



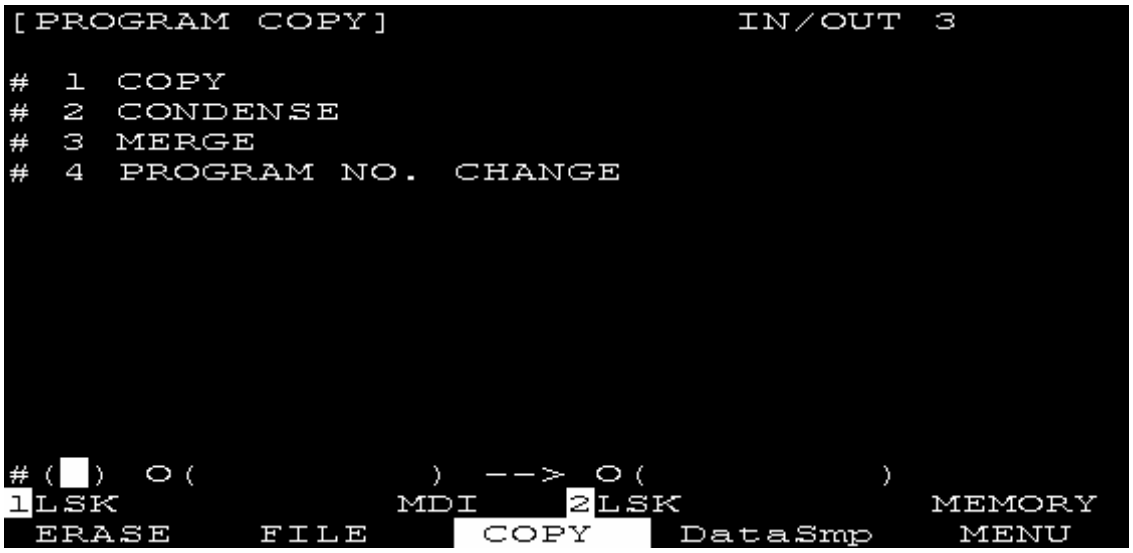
Item	Explanation
PROGRAM ENTRY and REMAIN	The number of programs already registered as user machining programs is displayed in the PROGRAM ENTRY field. The remaining number of programs that can be registered is displayed in the REMAIN field. The sum total of the PROGRAM ENTRY and REMAIN values is the maximum number of registered programs. This value is defined in the specifications.
CHARACTER and REMAIN	The number of characters already registered as user machining programs is displayed in the CHARACTER field. The remaining number of characters that can be registered is displayed in the REMAIN field. The sum total of the CHARACTER and REMAIN values is the maximum number of stored characters. This value is defined in the specifications. A value in 250-character units is displayed in REMAIN.
<PROGRAM> <CHR> <ST> <COMMENT>	<PROGRAM> The numbers of the already registered machining program are indicated in the ascending order in the range of 1 to 99999999. <CHR> The number of stored characters is indicated for each machining program number. <ST> The machining program status is indicated. (Not used) <COMMENT> A summary of the machining program functions, specifications, and applications can be displayed as a comment of up to 18 alphanumeric and symbol characters. It can be specified by data input. It can also be set on the screen.

(Note 1) If a large number of short programs are registered or edit operation is performed frequently, the memory use efficiency lowers because of the memory configuration and (CHARACTER + REMAIN) becomes less than the maximum number of available characters that are stored. If the difference between them is extremely large, the condense function can be used to improve the memory use efficiency.

(Note 2) If preparing comment, [SP] (space) can be written in it. But, the space is ignored after registration for efficient use of memory.

2.6.3 Program Copy

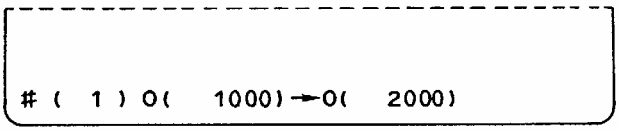
When the menu [COPY] is pressed, the PROGRAM COPY screen is displayed.
Copy, condense, merge, and number change of user-prepared machining programs (main program and subprogram) can be performed on the PROGRAM COPY screen.



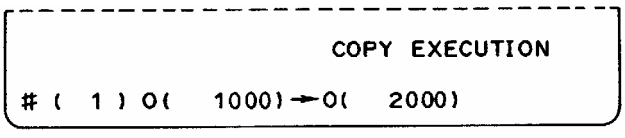
2.6.3.1 Machining Program Copy

A machining program selected among work programs registered in memory can be copied as another program having a different program number. The source program remains intact.
Set 1 in # ( ) for the copy command. Set the program number of the copy source program and the program number of new program.

(Example)
To copy program O1000 into O2000
(1)
O (1 0 0 0) → O (2 0 0 0)

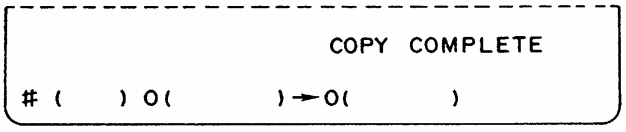


Press the [INPUT] key.



1) A copy is started and the message COPY EXECUTION is displayed in the message field. It may be terminated in a moment (when there is less copy data).

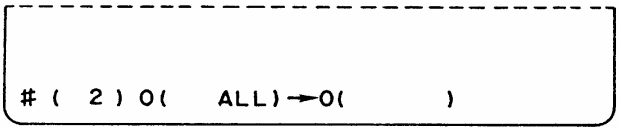
2) When the copy is complete, the message COPY COMPLETE is displayed in the message field.



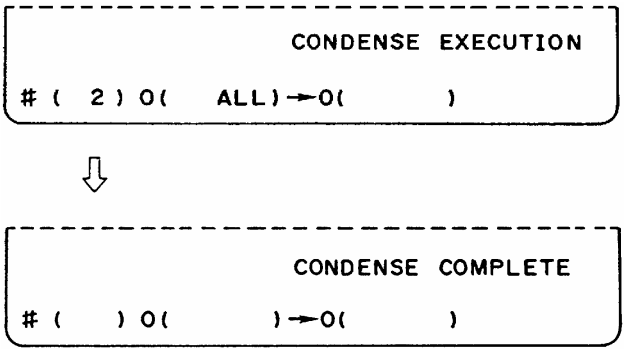
2.6.3.2 Machining Program Condense

The storage efficiency of the machining programs registered in memory may be lowered when correction such as data deletion or addition is made. When memory is used wastefully, the condense function can be used to move the intermediate blank portions forward and increase the remaining number of characters that can be registered.
Set 2 in # ( ) for the condense command. To execute the condense function for all machining programs registered in memory, set ALL in first 0 ( ).

To condense all work programs registered in memory:
(2) O (ALL) → O ( )



Press the [INPUT] key.



1) Condense is started and the message CONDENSE EXECUTION is displayed in the message field.
2) The time required for condense varies depending on the total number of the registered machining programs. Do not turn off the NC power during condense execution.

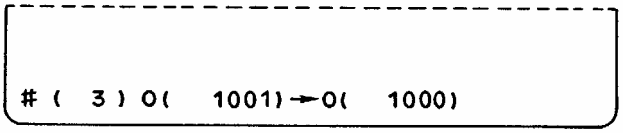
2.6.3.3 Machining Program Merge

To prepare a new machining program by using the machining program contents stored in memory, one machining program can be followed by a copy of another program. The two programs are merged into a new program.
Set 3 in # ( ) for the merge command. Set the copy source and destination program numbers in order.

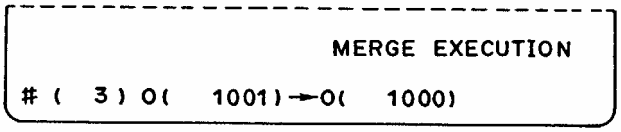
(Example 1)
To move program O1001 to the area following program O1000 for merge

([3])

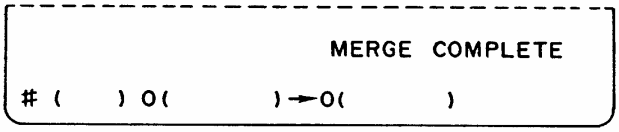
O ([1][0][0][1]) → O ([1][0][0][0])



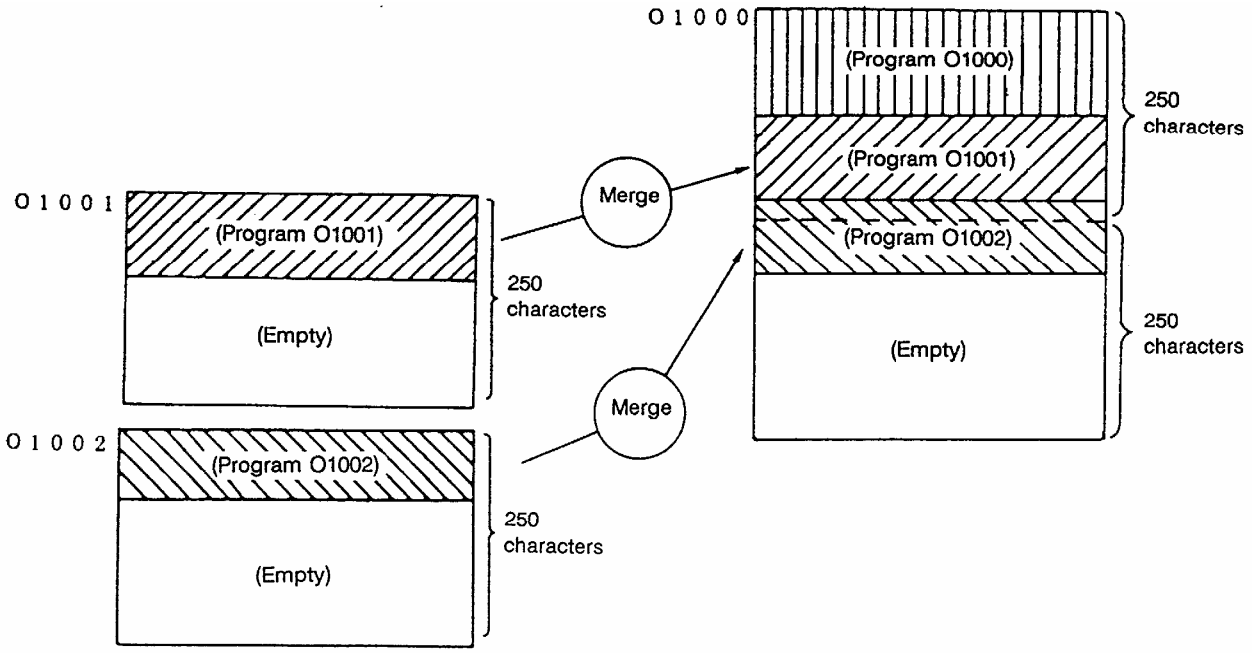
Press the [INPUT] key.



1) Merge is executed and the message MERGE EXECUTION is displayed in the message field. It may be terminated in a moment (when there is less transfer data).
2) When the merge is completed, the message MERGE COMPLETE is displayed in the message field.



(Example 2) When program O1002 is moved to the area following program O1000 for merge after operation in Example 1 (previous page), a new program is prepared as shown below. The three programs are merged into O1000. O1001 and O1002 remain unchanged.



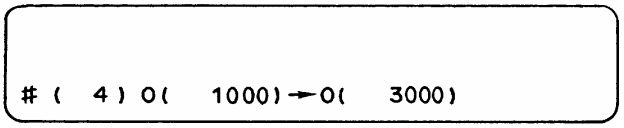
2.6.3.4 Changing the Machining Program Number

The program number of a machining program registered in memory can be changed.
To execute the number change command, set #4. Set the current and new program numbers in order.

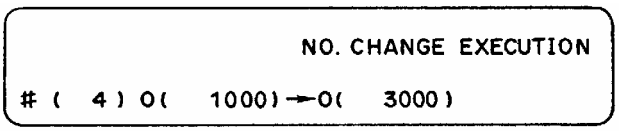
(Example)
When O1000 is changed to O3000:

([4])

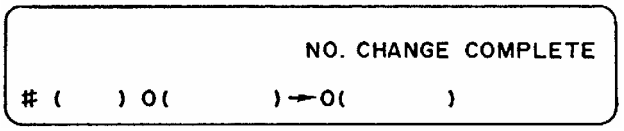
O ([1][0][0][0]) → ([3][0][0][0])



Press the [INPUT] key.

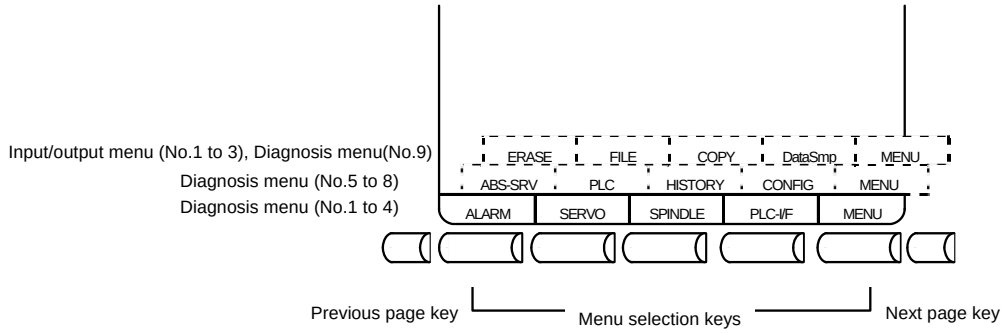


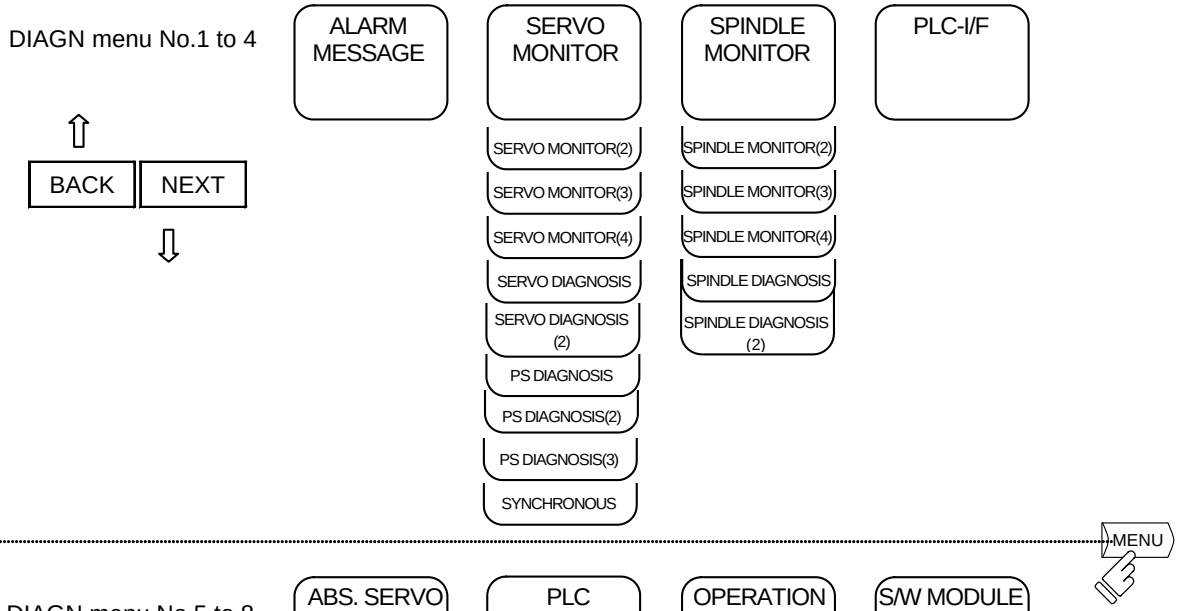
1) Number change processing is initiated with "NO. CHANGE EXECUTION" displayed in the message field. This processing is completed in a moment. Then, the message "NO. CHANGE COMPLETE" is displayed.

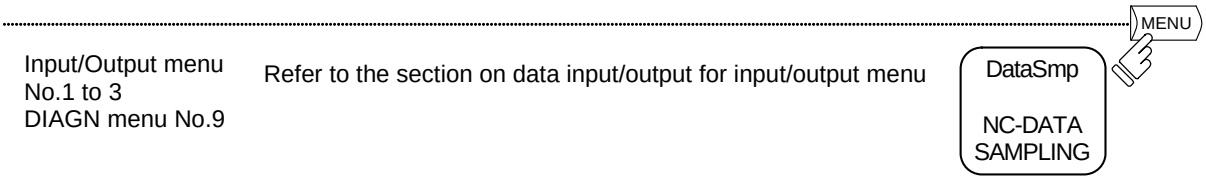


2.7 Diagnosis

When the function selection key [DIAGN IN/OUT] is pressed, the following menu is displayed.





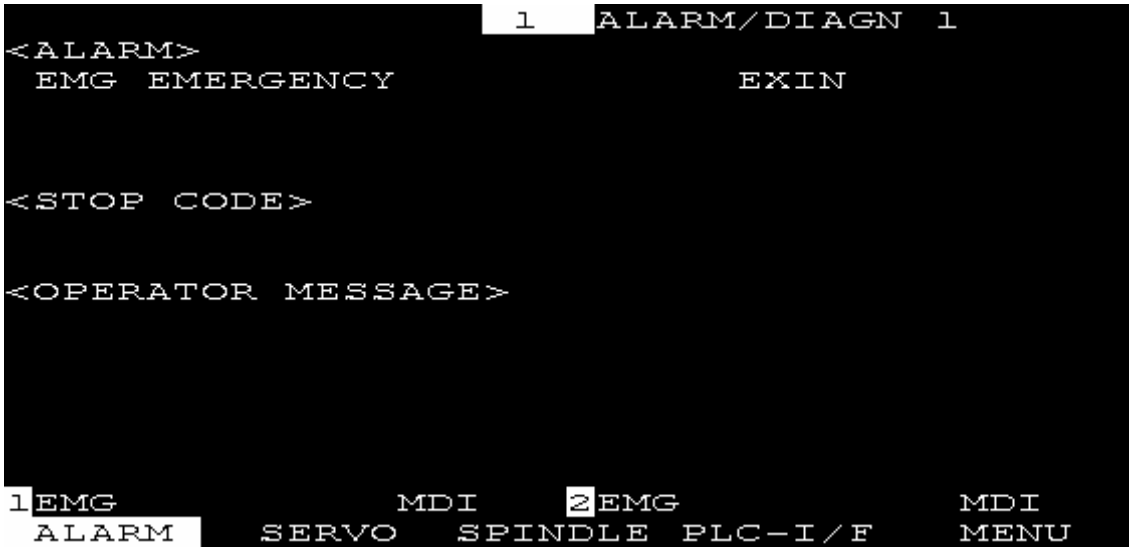


2.7.1 Alarm Message

When the menu [ALARM] is pressed, the ALARM/DIAGN screen is displayed.

(1) Alarm
The code and number or message relating to an operation alarm, program error, MCP alarm, servo alarm, or system error are displayed.

(2) Stop code
The automatic operation disable state or stop state in automatic operation mode is displayed in code and error number.



Refer to the Appendix. List of Alarms for details on the alarms.
When an alarm occurs, the class code will display on all screens.

2.7.1.1 Tracing of Alarm and Stop Codes

The alarm data will be stored if an alarm occurs. The stop code displayed at automatic operation stop, etc., is also stored. The alarm data and stop codes are stored separately. Up to 24 of each is registered, and the last 24 occurrences can be traced.

(1) Diagnosis of stored alarm and stop code data

The data in which the latest 24 alarms or stop codes are stored can be displayed on the alarm diagnosis screen and traced. The procedure is shown below.

1) Press either of the following cursor keys.
[↑] [↓]

The stored alarm data will display.

<Alarm> 1

Third to previous
Second to previous
Previous
Latest alarm

<Stop code>

Latest stop code

The digit to the right of the <Alarm> display shows the occurrence of the data.

2) The display can be changed to the previous and next alarms with the [↑] [↓] keys.

The [↑] key will show the previous data occurrence.
The [↓] key will show the next data occurrence.

3) Press the [SHIFT] [C.B CAN] keys.

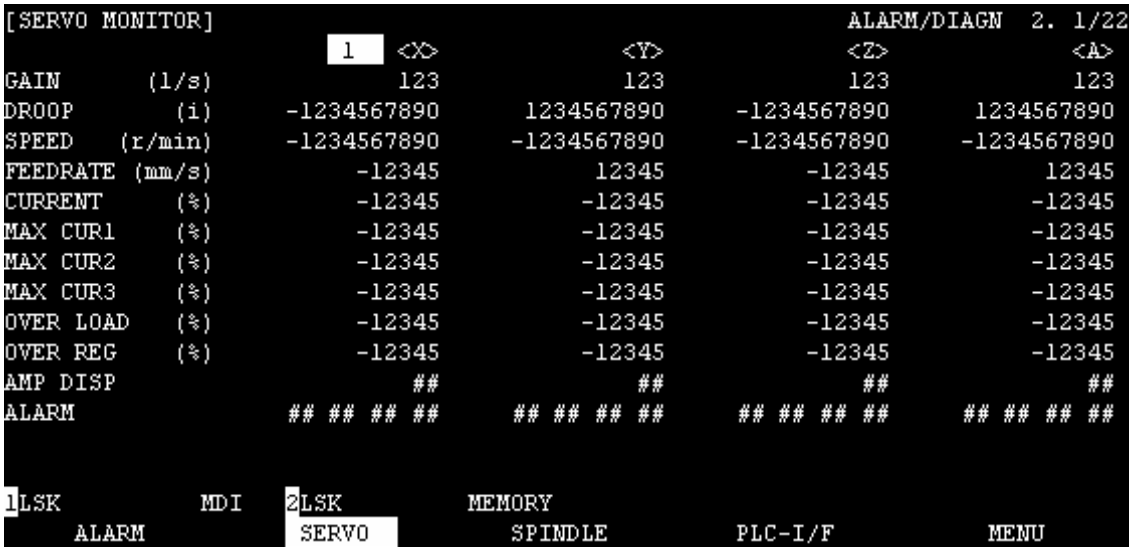
The display will return to the normal current alarm display screen. This is the same for changing over to other screens once.

(Note) The number of registered items depends on the model.

2.7.2 Servo Monitor

When the menu [SERVO] is pressed, the SERVO MONITOR screen is displayed indicating servo status during operation.

2.7.2.1 Servo Monitor

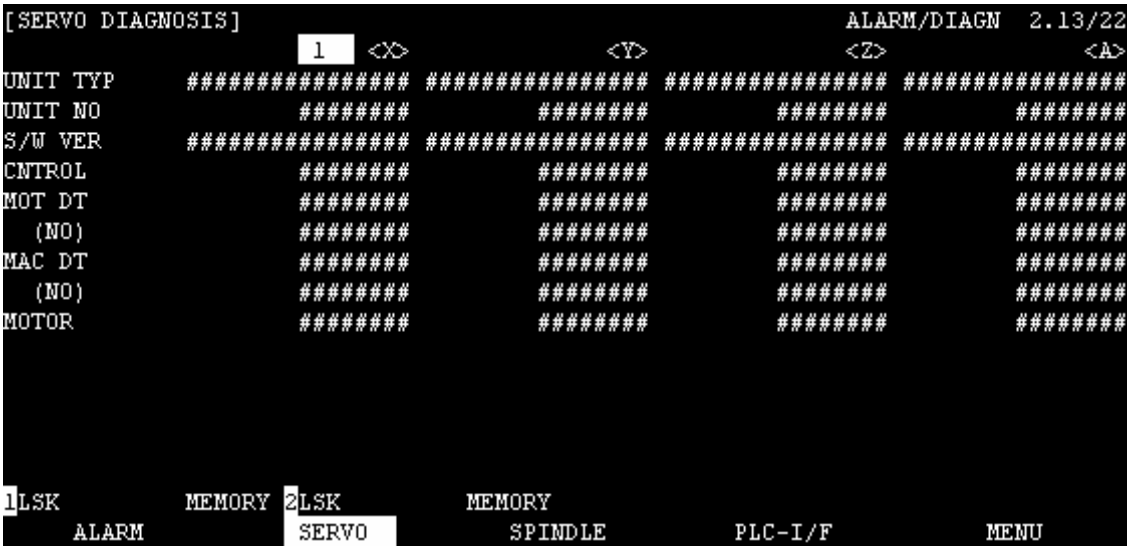


Display item		Details
GAIN	(1/s)	This displays the position loop gain. The position loop gain is: $\frac{\text{Feedrate (mm/s)}}{\text{Tracking delay error (mm)}}$
DROOP	(i)	The error of the actual machine position to the command position is called droop. This error is proportional to command speed value. The unit "i" means the input unit. A value will be displayed in μm unit when the system applies the μm unit.
SPEED	(r/min)	This displays the actual rotation speed of motor.
FEEDRATE	(mm/s)	This displays the feedrate detected by the detector mounted on the machine end.
CURRENT	(%)	This displays the FB value of the motor current in terms of continuous current during stalling.
MAX CUR 1	(%)	This displays the motor current command in terms of continuous current during stalling. An absolute peak value of the current command, which was sampled after the power ON, is displayed.
MAX CUR 2	(%)	This displays the motor current command in terms of continuous current during stalling. An absolute peak value of the current command, which was sampled within the last two seconds, is displayed.

Display item	Details
MAX CUR 3 (%)	This displays the FB value of the motor current in terms of continuous current during stalling. An absolute peak value of the current FB, which was sampled within the last two seconds, is displayed.
OVER LOAD (%)	This is the data used to monitor the overload on the motor.
OVER REG (%)	This is the data used to monitor the resistance load state when the resistance regenerative power supply is connected.
AMP DISP	This displays the 7-segment LED of the driver.
ALARM	This displays the alarms and warnings other than the LED display (displayed on drive unit).
EST DISTURB TORQ (%)	This displays the estimated disturbance torque in terms of stall rated torque when the disturbance observer is valid.
MAX DISTURB TORQ (%)	This displays the estimated disturbance torque in terms of stall rated torque when the collision detection function is adjusted. An absolute peak value of the estimated disturbance torque, which was sampled within the last two seconds, is displayed.
LOAD INERTIA R. (%)	This displays the estimated load inertia ratio when the collision detection function is adjusted.
Temperature (deg C)	This displays the thermistor temperature.
AFLT FREQUENCY (Hz)	This displays the present operation frequency of the adaptive filter.
AFLT GAIN (dB)	This displays the present filter depth of the adaptive filter.
CYC CNT (p)	This displays the position within one rotation of the encoder detector. The position is displayed as a grid point value as "0", within one rotation in the range of "0" to "RNG (movement units) ∗ 1000".
GRDSP	This displays the grid space for the reference position return. (Command unit)
GRID	This displays the distance from the dog-off point to the grid point in the dog-type reference position return. The grid mask amount is not included. (Command unit)
MAC POS	This displays the NC basic machine coordinate system position. (Command unit)
MOT POS	This displays the feedback position of the speed detector. (Command unit)
SCA POS	This displays the feedback position of the machine end position detector. (Command unit)
FB ERROR (i)	This displays the deviation between the motor end FB and the machine end FB.
DFB COMP (i)	This displays the compensation pulse amount during dual feedback control.
DIS TO GO	This displays the remaining travel distance of one block. (Command unit)
POSITION(2)	This displays the current position amount after the tool compensation amount is subtracted from the amount. (Command unit)

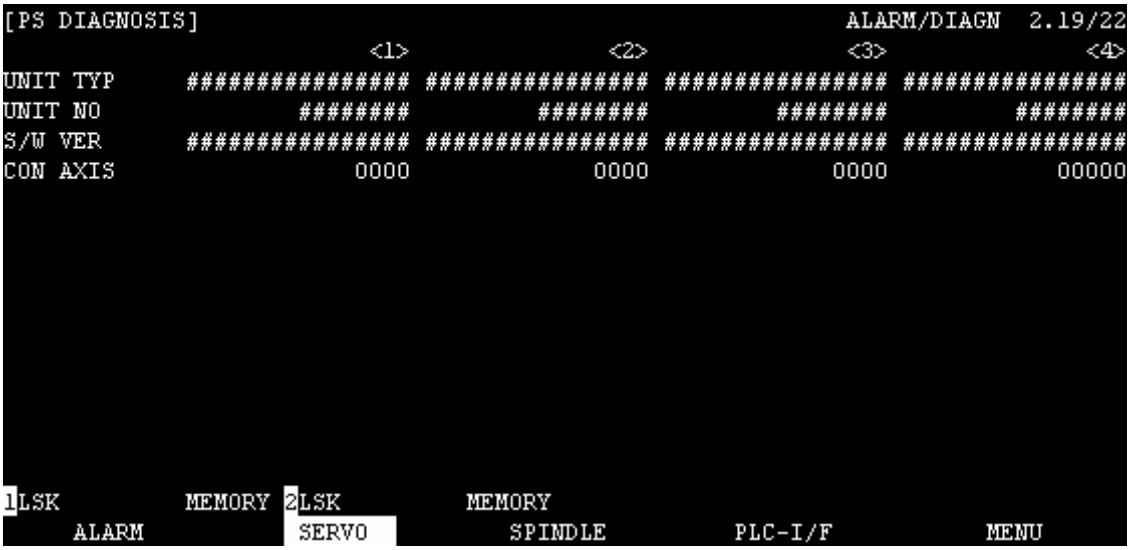
Display item	Details
MANUAL IT	This displays the amount of interrupt movement in the manual absolute OFF state. (Command unit)
CMD 1 to 6	This indicates the control signal input from NC. This is used by the system.
STS 1 to 6	This indicates the control signal output to NC. This is used by the system.

2.7.2.2 Servo Diagnosis



Display item	Details
UNIT TYP	This displays the servo driver type.
UNIT NO	This displays the servo driver serial No.
S/W VER	This displays the servo side software version.
CNTROL	SEMI : Semi-closed loop CLOSED: Closed loop DUAL : Dual feedback
MOT DT	This displays the motor end detector type.
MOT DT(NO)	This displays the motor end detector serial No.
MAC DT	This displays the machine end detector type. The type is displayed when the control method is CLOSED or DUAL. * is displayed when the method is SEMI.
MAC DT(NO)	This displays the machine end detector serial No.
MOTOR	This displays the motor type.
WORK TIME	This displays the READY ON work time. (Units: 1hr)
ALARM HIST 1 to 8	This displays servo alarms that occurred in latest order with the following formats. Time: Work time when the alarm occurred. Alarm No.: No. of the servo alarms that occurred.
MNT/SYS	Maintenance history Status

2.7.2.3 Power Supply Diagnosis

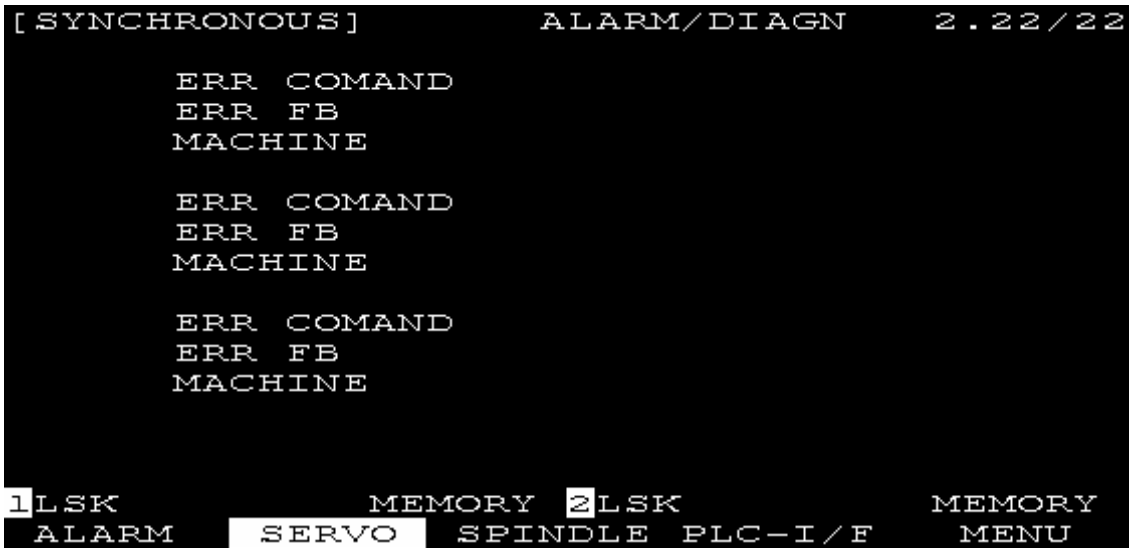


Display item	Details
UNIT TYP	This displays the power supply unit type.
UNIT NO	This displays the serial No. of the power supply unit.
S/W VER	This displays the software version.
CON AXIS	This displays the I/F channel No. (mcp_no, smcp_no) of the drive unit connected to each power supply unit.
RECOVERY ENERGY(KW)	This displays the regenerative power every two seconds. (0 to 999kW)
PW. SPLY VOLT (Vrms)	This displays the effective value of the power supply voltage. (0 to 999Vrms)
PN BUS VOLTAGE (V)	This displays PN bus voltage. (0 to 999V)
MIN PN BUS VOLT (V)	This displays the minimum PN bus voltage after the NC power ON. (0 to 999V)
MIN PN CURRENT (%)	This displays the bus current when PN bus voltage is at minimum. (driving: +、regenerative: -) (0 to 999%)
BUS CURRENT (%)	This displays the bus current. (driving: +、regenerative: -) (0 to 999%)
MAX CURRENT1 (%)	This displays the maximum driving current after the NC power ON. (0 to 999%)
MAX CURRENT2 (%)	This displays the maximum driving current in most recent 2 seconds. (0 to 999%)
MAX RGN CURRENT1(%)	This displays the maximum regenerative current after the NC power ON. (0 to 999%)
MAX RGN CURRENT2(%)	This displays the maximum regenerative current in most recent 2 seconds. (0 to 999%)
NO. OF INSTANT STOP	This displays the number of instantaneous stop exceeding 1 cycle of the power. (0 to 9999 times)
WORK TIME	This displays the READY ON cumulative time. (Units: 1hr)

Display item	Details
ALM HIST 1 to 8	This displays servo alarms that occurred in latest order with the following formats. Time : Work time when the alarm occurred Alarm No. : Number of the servo alarms that occurred
MNT/SYS	Maintenance history Status

2.7.2.4 Synchronous Error

The various data related to the synchronous error is monitored.



Display item	Explanation
ERR COMMAND	This is the deviation of the slave axis machine position in respect to the master axis. This displays the error of the command value to the servo control section before compensating the pitch error, incremental position, or backlash. If this error occurs, the parameters that should be the same for the synchronous axes are different. Check the setting value. Command error = Command "s" – command "m" – Δ Command "s": Slave axis command value Command "m": Master axis command value Δ: Command "s" – command "m" at start of synchronous control
ERR FB	This is the deviation of the slave axis feedback value in respect to the feedback value from the master axis servomotor. This displays the error of the actual machine position. The synchronous error check is carried out on this error. FB error = FBs – FBm – Δ FBs : Slave axis feedback value FBm: Master axis feedback value Δ: FBs – FBm at start of synchronous control
MACHINE	This displays the machine position of the master axis.

(Note 1) The error offset amount "Δ" is made when the operation method is changed because the PLC interface, "Synchronization control operation method" is changed.
Note that if the zero point has not established, the error offset amount "Δ" is made immediately after the zero point is established.

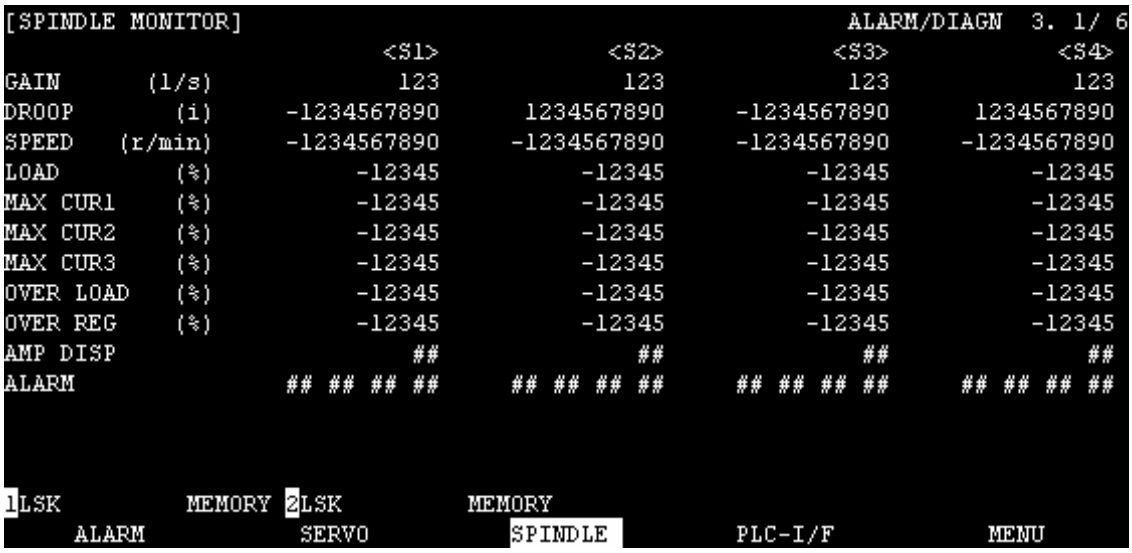
(Note 2) The following table indicates what is displayed for each item.

	Synchronous operation	Independent operation	Asynchronous operation
Axis name	Displayed	Displayed	Not displayed
Command error	Displayed	Display 0.000	Display 0.000
FB error	Displayed	Display 0.000	Display 0.000
Machine position	Displayed	Displayed	Display 0.000

2.7.3 Spindle Monitor

When the menu [SPINDLE] is selected, the state of the spindle in operation is displayed.

2.7.3.1 Spindle Monitor



Display items		Details
GAIN	(1/s)	This displays the position loop gain. The position loop gain is: $\frac{\text{Feedrate (mm/s)}}{\text{Tracking delay error (mm)}}$
DROOP	(i)	The error of the actual machine position to the command position is called droop. This error is proportional to command speed value. The unit "i" means the input unit. A value will be displayed in μm unit when the system applies the μm unit.
SPEED	(r/min)	This displays the actual rotation speed of motor.
LOAD	(%)	This displays the motor load.
MAX CUR 1	(%)	This displays the motor current command in terms of continuous current during stalling. An absolute value of the current command peak value sampled after the power ON is displayed.
MAX CUR 2	(%)	This displays the motor current command in terms of continuous current during stalling. An absolute value of the current command peak value sampled in most recent 2 seconds is displayed.
MAX CUR 3	(%)	This displays the FB value of the motor current in terms of continuous current during stalling. An absolute value of the current FB peak value sampled in most recent 2 seconds is displayed.
OVER LOAD	(%)	This is the data used to monitor the motor overload.
OVER REG	(%)	This is the data used to monitor the resistance overload state when the resistance regenerative power supply is connected.

Display items	Details
AMP DISP	This displays the 7-segment LED of the driver.
ALARM	This displays the alarms and warnings other than the LED display on drive unit.
EST DISTURB TORQ (%)	This displays the estimated disturbance torque in terms of stall rated torque when the disturbance observer is valid.
MAX DISTURB TORQ (%)	This displays the estimated disturbance torque in terms of stall rated torque when the collision detection function is adjusted. An absolute value of the estimated disturbance torque peak value sampled in most recent 2 seconds is displayed.
LOAD INERTIA R. (%)	This displays the estimated load inertia ratio when the collision detection function is adjusted.
Temperature (deg C)	This displays the spindle motor thermistor temperature.
AFLT FREQUENCY (Hz)	This displays the current operation frequency of the adaptive filter.
AFLT GAIN (dB)	This displays the current filter depth of the adaptive filter.
CYC CNT (p)	This displays the position within one rotation of the encoder detector. The position is displayed within one rotation in the range of "0" to "RNG (movement units) × 1000" using the grid point value as "0".
GRDSP	This displays the grid space for the reference position return. (Command unit)
GRID	This displays the distance from the dog-off point to the grid point when the dog-type reference position return is displayed. The grid mask amount is not included. (Command unit)
MAC POS	This displays the NC basic machine coordinate system position. (Command unit)
MOT POS	This displays the feedback position of the speed detector. (Command unit)
SCA POS	This displays the feedback position of the machine end position detector. (Command unit)
FB ERROR (i)	This displays the deviation of the motor end FB and machine end FB.
DFB COMP (i)	This displays the compensation pulse amount during dual feedback control.
ANGLE (deg)	This displays the position within one rotation of the encoder detector. The position is displayed as a grid point value as "0", within one rotation in the range "0" to "359.999".

<table>
<tr><th>Display item</th><th>Details</th></tr>
<tr><td>CMD 1</td><td>This displays the control input signals from the NC.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">1L</td><td>0</td><td>READY ON command</td></tr>
<tr><td>1</td><td>Servo ON command</td></tr>
<tr><td>2</td><td></td></tr>
<tr><td>3</td><td></td></tr>
<tr><td>4</td><td></td></tr>
<tr><td>5</td><td></td></tr>
<tr><td>6</td><td></td></tr>
<tr><td>7</td><td>Servo alarm reset command</td></tr>
<tr><td rowspan="8">1H</td><td>8</td><td>Torque limit selection command 1</td></tr>
<tr><td>9</td><td>Torque limit selection command 2</td></tr>
<tr><td>A</td><td>Torque limit selection command 3</td></tr>
<tr><td>B</td><td></td></tr>
<tr><td>C</td><td></td></tr>
<tr><td>D</td><td></td></tr>
<tr><td>E</td><td></td></tr>
<tr><td>F</td><td></td></tr>
</table></td></tr>
<tr><td>CMD 2</td><td>This displays the control input signals from the NC.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">2L</td><td>0</td><td></td></tr>
<tr><td>1</td><td></td></tr>
<tr><td>2</td><td></td></tr>
<tr><td>3</td><td></td></tr>
<tr><td>4</td><td></td></tr>
<tr><td>5</td><td></td></tr>
<tr><td>6</td><td></td></tr>
<tr><td>7</td><td></td></tr>
<tr><td rowspan="8">2H</td><td>8</td><td></td></tr>
<tr><td>9</td><td>Speed observation command valid</td></tr>
<tr><td>A</td><td>Door closed (controller)</td></tr>
<tr><td>B</td><td>Door closed (all drive units)</td></tr>
<tr><td>C</td><td></td></tr>
<tr><td>D</td><td></td></tr>
<tr><td>E</td><td></td></tr>
<tr><td>F</td><td></td></tr>
</table></td></tr>
</table>

<table>
<tr><th>Display item</th><th>Details</th></tr>
<tr><td>CMD 3</td><td>This displays the control input signals from the NC.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">3L</td><td>0</td><td></td></tr>
<tr><td>1</td><td></td></tr>
<tr><td>2</td><td></td></tr>
<tr><td>3</td><td></td></tr>
<tr><td>4</td><td></td></tr>
<tr><td>5</td><td></td></tr>
<tr><td>6</td><td></td></tr>
<tr><td>7</td><td></td></tr>
<tr><td rowspan="8">3H</td><td>8</td><td></td></tr>
<tr><td>9</td><td></td></tr>
<tr><td>A</td><td></td></tr>
<tr><td>B</td><td></td></tr>
<tr><td>C</td><td></td></tr>
<tr><td>D</td><td></td></tr>
<tr><td>E</td><td></td></tr>
<tr><td>F</td><td></td></tr>
</table></td></tr>
<tr><td>CMD 4</td><td>This displays the control input signals from the NC.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">4L</td><td>0</td><td rowspan="3">Spindle control mode selection command 1, 2, 3</td></tr>
<tr><td>1</td></tr>
<tr><td>2</td></tr>
<tr><td>3</td><td></td></tr>
<tr><td>4</td><td></td></tr>
<tr><td>5</td><td>Gear selection command 1</td></tr>
<tr><td>6</td><td>Gear selection command 2</td></tr>
<tr><td>7</td><td></td></tr>
<tr><td rowspan="8">4H</td><td>8</td><td></td></tr>
<tr><td>9</td><td></td></tr>
<tr><td>A</td><td></td></tr>
<tr><td>B</td><td></td></tr>
<tr><td>C</td><td>M-coil switch command</td></tr>
<tr><td>D</td><td>L-coil switch command</td></tr>
<tr><td>E</td><td>Sub-motor selection command</td></tr>
<tr><td>F</td><td></td></tr>
</table></td></tr>
</table>

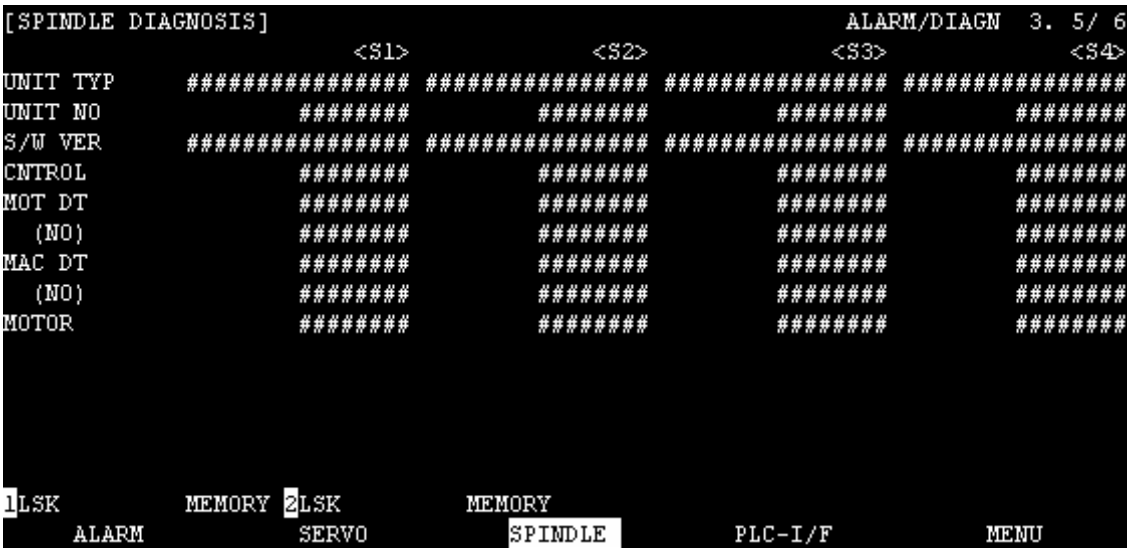
<table>
<tr><th>Display item</th><th>Details</th></tr>
<tr><td>CMD 5</td><td>This displays the control input signals from the NC.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">5L</td><td>0</td><td></td></tr>
<tr><td>1</td><td></td></tr>
<tr><td>2</td><td></td></tr>
<tr><td>3</td><td></td></tr>
<tr><td>4</td><td></td></tr>
<tr><td>5</td><td></td></tr>
<tr><td>6</td><td></td></tr>
<tr><td>7</td><td></td></tr>
<tr><td rowspan="8">5H</td><td>8</td><td></td></tr>
<tr><td>9</td><td></td></tr>
<tr><td>A</td><td></td></tr>
<tr><td>B</td><td></td></tr>
<tr><td>C</td><td></td></tr>
<tr><td>D</td><td></td></tr>
<tr><td>E</td><td>Spindle holding force up</td></tr>
<tr><td>F</td><td></td></tr>
</table></td></tr>
<tr><td>CMD 6</td><td>This displays the control input signals from the NC.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">6L</td><td>0</td><td></td></tr>
<tr><td>1</td><td></td></tr>
<tr><td>2</td><td></td></tr>
<tr><td>3</td><td></td></tr>
<tr><td>4</td><td></td></tr>
<tr><td>5</td><td></td></tr>
<tr><td>6</td><td></td></tr>
<tr><td>7</td><td></td></tr>
<tr><td rowspan="8">6H</td><td>8</td><td></td></tr>
<tr><td>9</td><td></td></tr>
<tr><td>A</td><td></td></tr>
<tr><td>B</td><td></td></tr>
<tr><td>C</td><td></td></tr>
<tr><td>D</td><td></td></tr>
<tr><td>E</td><td></td></tr>
<tr><td>F</td><td></td></tr>
</table></td></tr>
</table>

<table>
<tr><th>Display item</th><th>Details</th></tr>
<tr><td>STS 1</td><td>This displays the control output signals to the NC.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">1L</td><td>0</td><td>In READY ON</td></tr>
<tr><td>1</td><td>In Servo ON</td></tr>
<tr><td>2</td><td></td></tr>
<tr><td>3</td><td></td></tr>
<tr><td>4</td><td></td></tr>
<tr><td>5</td><td></td></tr>
<tr><td>6</td><td></td></tr>
<tr><td>7</td><td>In alarm occurrence</td></tr>
<tr><td rowspan="8">1H</td><td>8</td><td>In torque limit selection 1</td></tr>
<tr><td>9</td><td>In torque limit selection 2</td></tr>
<tr><td>A</td><td>In torque limit selection 3</td></tr>
<tr><td>B</td><td></td></tr>
<tr><td>C</td><td>In-position</td></tr>
<tr><td>D</td><td>In torque limit</td></tr>
<tr><td>E</td><td></td></tr>
<tr><td>F</td><td>In warning occurrence</td></tr>
</table></td></tr>
<tr><td>STS 2</td><td>This displays the control output signals to the NC.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">2L</td><td>0</td><td>Z-phase passed</td></tr>
<tr><td>1</td><td></td></tr>
<tr><td>2</td><td></td></tr>
<tr><td>3</td><td>In zero speed</td></tr>
<tr><td>4</td><td></td></tr>
<tr><td>5</td><td></td></tr>
<tr><td>6</td><td></td></tr>
<tr><td>7</td><td>In external emergency stop</td></tr>
<tr><td rowspan="8">2H</td><td>8</td><td></td></tr>
<tr><td>9</td><td>In speed observation</td></tr>
<tr><td>A</td><td>Door closed (controller)</td></tr>
<tr><td>B</td><td>Door closed (Local drive units)</td></tr>
<tr><td>C</td><td></td></tr>
<tr><td>D</td><td></td></tr>
<tr><td>E</td><td></td></tr>
<tr><td>F</td><td></td></tr>
</table></td></tr>
</table>

<table>
<tr><th>Display item</th><th>Details</th></tr>
<tr><td>STS 3</td><td>This displays the control output signals to the NC.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">3L</td><td>0</td><td></td></tr>
<tr><td>1</td><td></td></tr>
<tr><td>2</td><td></td></tr>
<tr><td>3</td><td></td></tr>
<tr><td>4</td><td></td></tr>
<tr><td>5</td><td></td></tr>
<tr><td>6</td><td></td></tr>
<tr><td>7</td><td></td></tr>
<tr><td rowspan="8">3H</td><td>8</td><td></td></tr>
<tr><td>9</td><td></td></tr>
<tr><td>A</td><td></td></tr>
<tr><td>B</td><td></td></tr>
<tr><td>C</td><td></td></tr>
<tr><td>D</td><td></td></tr>
<tr><td>E</td><td></td></tr>
<tr><td>F</td><td></td></tr>
</table></td></tr>
<tr><td>STS 4</td><td>This displays the control output signals to the NC.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">4L</td><td>0</td><td rowspan="3">Spindle control mode selected 1, 2, 3</td></tr>
<tr><td>1</td></tr>
<tr><td>2</td></tr>
<tr><td>3</td><td></td></tr>
<tr><td>4</td><td></td></tr>
<tr><td>5</td><td>Gear selected 1</td></tr>
<tr><td>6</td><td>Gear selected 2</td></tr>
<tr><td>7</td><td></td></tr>
<tr><td rowspan="8">4H</td><td>8</td><td></td></tr>
<tr><td>9</td><td></td></tr>
<tr><td>A</td><td></td></tr>
<tr><td>B</td><td></td></tr>
<tr><td>C</td><td>M-coil switched</td></tr>
<tr><td>D</td><td>L- coil switched</td></tr>
<tr><td>E</td><td>Sub-motor selected</td></tr>
<tr><td>F</td><td></td></tr>
</table></td></tr>
</table>

<table>
<tr><th>Display item</th><th>Details</th></tr>
<tr><td>STS 5</td><td>This displays the control output signals to the NC.
<table>
<tr><td></td><td>Bit</td><td>Details</td></tr>
<tr><td rowspan="8">5L</td><td>0</td><td>Current detection</td></tr>
<tr><td>1</td><td>Speed detection</td></tr>
<tr><td>2</td><td></td></tr>
<tr><td>3</td><td></td></tr>
<tr><td>4</td><td></td></tr>
<tr><td>5</td><td></td></tr>
<tr><td>6</td><td>In coil changeover</td></tr>
<tr><td>7</td><td></td></tr>
<tr><td rowspan="8">5H</td><td>8</td><td>1-amplifire 2-motor switching</td></tr>
<tr><td>9</td><td>2nd speed detection</td></tr>
<tr><td>A</td><td></td></tr>
<tr><td>B</td><td></td></tr>
<tr><td>C</td><td></td></tr>
<tr><td>D</td><td></td></tr>
<tr><td>E</td><td>In spindle holding force up</td></tr>
<tr><td>F</td><td>2nd in-position</td></tr>
</table></td></tr>
<tr><td>STS 6</td><td>This displays the control output signals to the NC.
<table>
<tr><td></td><td>Bit</td><td>Details</td></tr>
<tr><td rowspan="8">6L</td><td>0</td><td></td></tr>
<tr><td>1</td><td></td></tr>
<tr><td>2</td><td></td></tr>
<tr><td>3</td><td></td></tr>
<tr><td>4</td><td></td></tr>
<tr><td>5</td><td></td></tr>
<tr><td>6</td><td></td></tr>
<tr><td>7</td><td></td></tr>
<tr><td rowspan="8">6H</td><td>8</td><td></td></tr>
<tr><td>9</td><td></td></tr>
<tr><td>A</td><td></td></tr>
<tr><td>B</td><td></td></tr>
<tr><td>C</td><td></td></tr>
<tr><td>D</td><td></td></tr>
<tr><td>E</td><td></td></tr>
<tr><td>F</td><td></td></tr>
</table></td></tr>
</table>

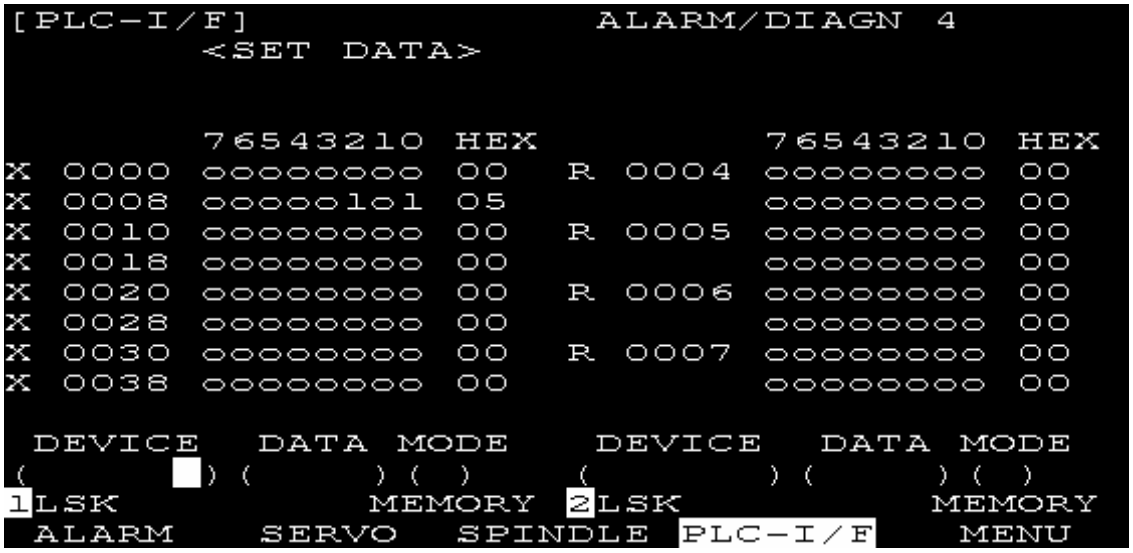
2.7.3.2 Spindle Diagnosis



Display item	Details
UNIT TYP	This displays the spindle type.
UNIT NO	This displays the spindle serial No.
S/W VER	This displays the software No. and version on the spindle side.
CNTROL	SEMI : Semi-closed loop CLOSED: Closed loop DUAL : Dual feedback
MOT DT	This displays the motor end detector type.
MOT DT(NO)	This displays the motor end detector serial No.
MAC DT	This displays the machine end detector type. The type is displayed when the control method is CLOSED or DUAL. * is displayed when the method is SEMI.
MAC DT(NO)	This displays the machine end detector serial No.
MOTOR	This displays the motor type.
WORK TIME	This displays the READY ON cumulative time. (Units: 1hr)
ALARM HIST 1 to 8	This displays servo alarms that occurred in latest order with the following formats. Time: Work time when the alarm occurred. Alarm No.: No. of the servo alarms that occurred.
MNT/SYS	Maintenance history Status

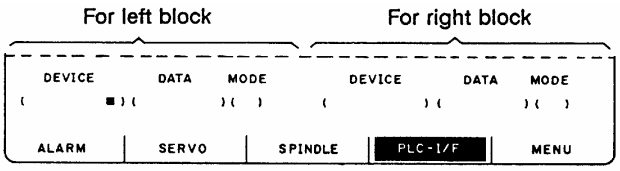
2.7.4 PLC Interface Diagnosis of CNC CPU

When the menu [PLC-I/F] is pressed, the CNC CPU's PLC-I/F screen is displayed.
The PLC-I/F screen enables you to set and display input/output signals for PLC (Programmable Logic Control Unit) control.
It can be used to check machine sequence operation during PLC development, check input/output data between control unit and PLC in operation trouble, and make forcible definition.



2.7.4.1 PLC-I/F Setting and Display

(1) Data setting area



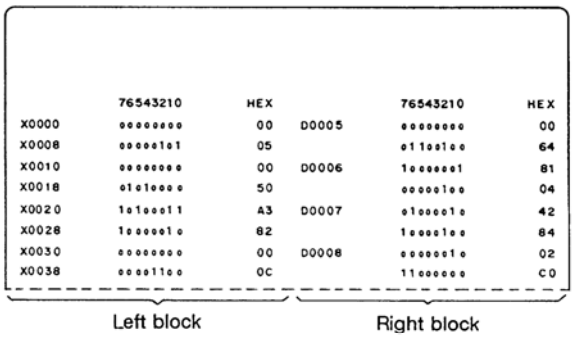
DEVICE ( )
Set the device number used with PLC (input X○○○, output Y○○○, and timer T○○).

DATA ( )
To forcibly define PLC data, set data corresponding to the setup device number. Set 1 or 0 for bit data. Set hexadecimal (HEX) data for byte data.

MODE ( )
Specify the type of forcible definition. 1: Single-shot type
2: Modal type

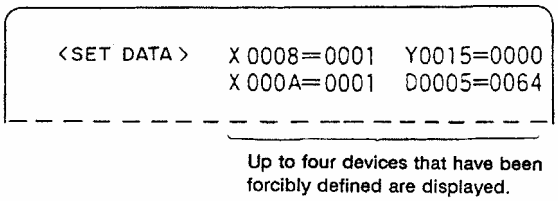
(2) Device data display area

Data corresponding to the device numbers specified in the setting area is displayed.
Data is displayed in both binary notation and hexadecimal notation.
The device numbers can be displayed in the left and right blocks separately.



(3) Forcible definition data display (modal type)

When forcible definition data and mode are set, the device numbers and data are displayed and setup data is forcibly defined.



(Note) To use this function during machine running, pay extreme attention to sequence operation.

(4) PLC display device names

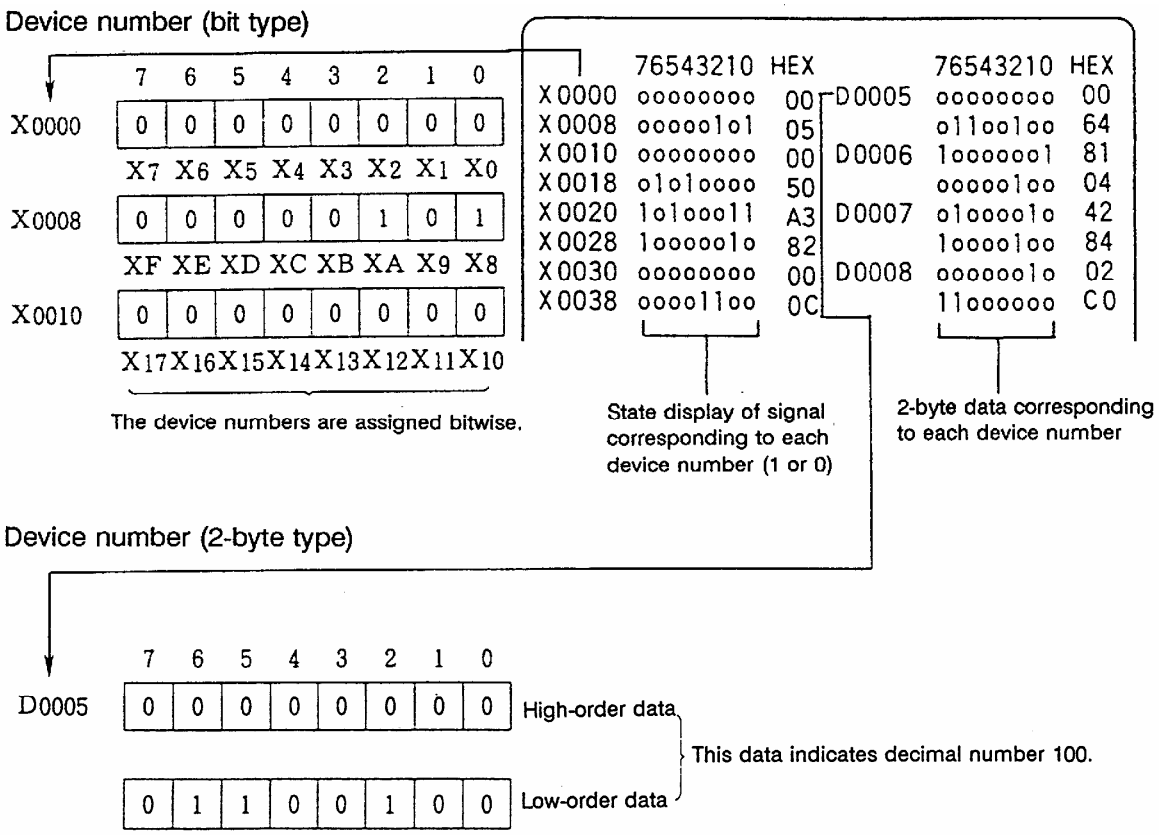
The devices used with the PLC are listed below.
The device number for devices X, Y, B, W and H are expressed with a hexadecimal. All other device numbers are expressed as decimals.

Device	Device range		Units	Details
X*	X0 to XAFF	2816 points	1-bit	Input signals to the PLC. Machine input, etc.
Y*	Y0 to YE7F	3584 points	1-bit	Output signals from the PLC. Machine output, etc.
M	MO to M15 M1000 to M8191	16 points 7192 points	1-bit	Safety ladder I/F signal For temporary memory
L	L0 to L255	256 points	1-bit	Latch relay (Backup memory)
F	F0 to F127	128 points	1-bit	For temporary memory. Alarm message interface
SM*	SM0 to SM127	128 points	1-bit	Special relay
SW	SW0 to SW1FF	512 points	16-bit	Special register for link
SD	SD0 to SD127	128 points	16-bit	Special register
T	T0 to T15	16 points	1-bit/16-bit	10ms unit timer
	T16 to T95	80 points	1-bit/16-bit	100ms unit timer
	T96 to T103	8 points	1-bit/16-bit	100ms incremented timer
	T240 to T255	16 points	1-bit/16-bit	100ms incremented timer (Fixed timers)
C	C0 to C23	24 points	1-bit/16-bit	Counter
	C24 to C127	104 points	1-bit/16-bit	Counter (Fixed counters)
D	D200 to D8191 (Note 3)	7992 points	16-bit/32-bit	Data register
R*	R0 to R9215	9216 points	16-bit/32-bit	File register. CNC word I/F
W	W0 to W1FFF	8192 points	16-bit/32-bit	Link register
Z	Z0 to Z1	2 points	16-bit	Address index
N	N0 to N7			Master control's nesting level
P*	P0 to P249 P370 to P378 P252, P255			Conditional jump, label for sub-routine
K	K-32768 to K32767			Decimal constant for 16-bit command
	K-2147483647 to K2147483647			Decimal constant for 32-bit command
H	H0 to HFFFF			Hexadecimal constant for 16-bit command
	H0 to HFFFFFFFF			Hexadecimal constant for 32-bit command

(Note 1) Devices with an asterisk in the device field have sections with predetermined applications. Do not use these devices for other applications.

(Note 2) Some of the fixed timers and fixed counters cannot be changed by setting the value. Note that this does not apply when the D, R devices are designated.

(5) How to read device numbers and display data



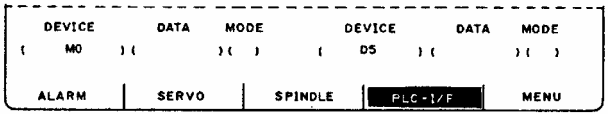
2.7.4.2 PLC Device Data Display

Monitor display of state signals and register data used with PLC can be made.
When the PLC-I/F screen is first selected, 8-byte input/output data starting at device X0000 is displayed in the left block; 8-byte input/output data starting at device Y0000 is displayed in the right block.
The screen always monitor-displays the PLC signal state. When PLC signal changes, the displayed state also changes with the PLC signal change.
However, a lag occurs between PLC signal change and signal display, thus signal display may be delayed or a response to a very short signal change may not be made.

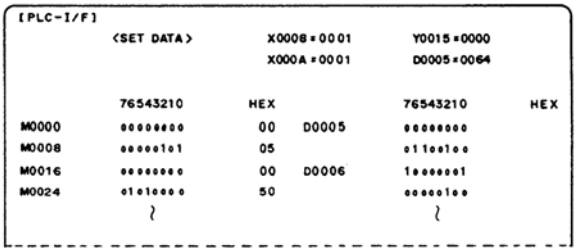
(1) Display device number setting
Set the device number in DEVICE ( ). If a different device number or device address is set in the right DEVICE ( ), the specified device numbers are displayed in the left and right halves of the screen from the display area top to bottom.

(Example 1)

Set M0 in left DEVICE ( ) and D5 in right DEVICE ( ).

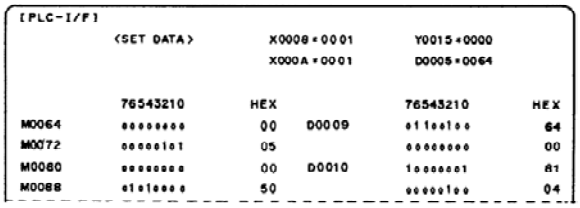


Press the [INPUT] key.



(2) Device number display change in 8-byte units
The current device number display at the top can be changed in 8-byte units by using the [BACK] or [NEXT] key.

When the display screen shown in Example 1 appears, press the [NEXT] key once.



(3) When the last device number is exceeded by feeding pages, the data display screen of the last device number remains.
(4) If a number exceeding the preset numbers in specifications or an invalid address is set, a setting error results.

2.7.4.3 PLC Interface Signal Forcible Definition (Single-shot Type)

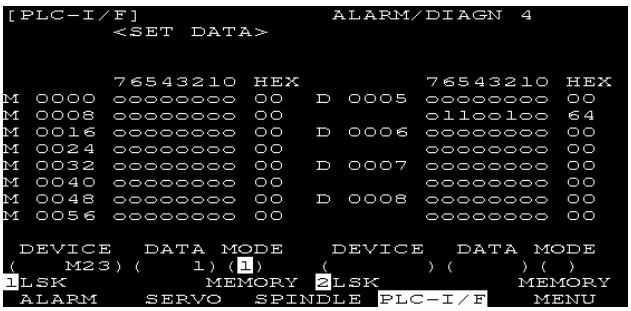
This function is used to forcibly define signals to check the PLC functions.

(1) Mode setting

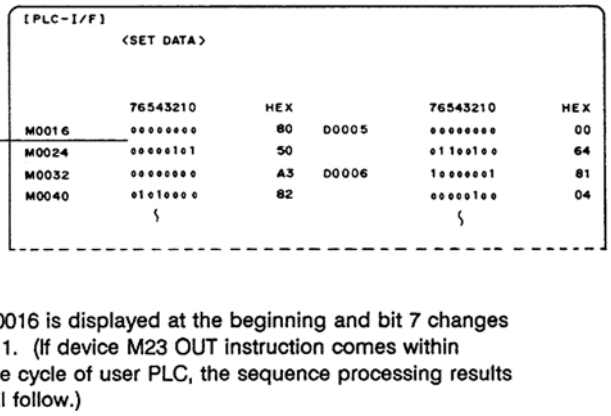
Set the device number to be forcibly set in DEVICE ( ), definition data in DATA ( ), and 1 in MODE ( ), then press the [INPUT] key. The setup data is processed and forcibly defined at the top of one cycle of user PLC.

(Example 2)

Set “M23” in DEVICE ( ),
“1” in DATA ( ), and
“1” in MODE ( ).



Press the [INPUT] key.



M0016 is displayed at the beginning and bit 7 changes to 1. (If device M23 OUT instruction comes within one cycle of user PLC, the sequence processing results will follow.)

Data is not displayed in the SET DATA field at the screen top.

CAUTION

When forcibly setting (forcibly outputting) data on the I/F diagnosis screen during machine operation, pay careful attention to the sequence operation.

(Note) An input signal to the PLC is updated at the beginning of each cycle of the PLC. Therefore, the signal, once forcibly defined in single-shot type mode, is restored after one cycle.

2.7.4.4 PLC Interface Signal Forcible Definition (Modal Type)

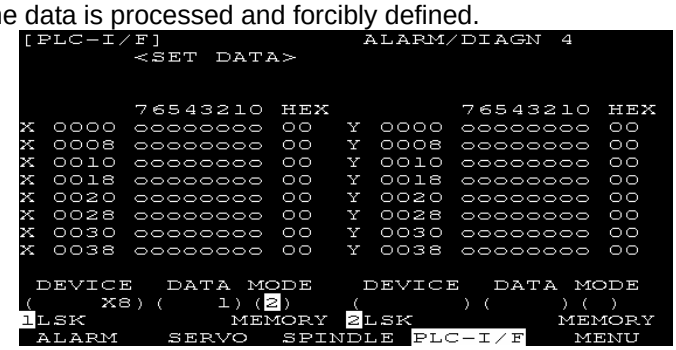
This is PLC interface signal forcible definition of modal type. Once it is set, it is held until canceled.

(1) Mode setting

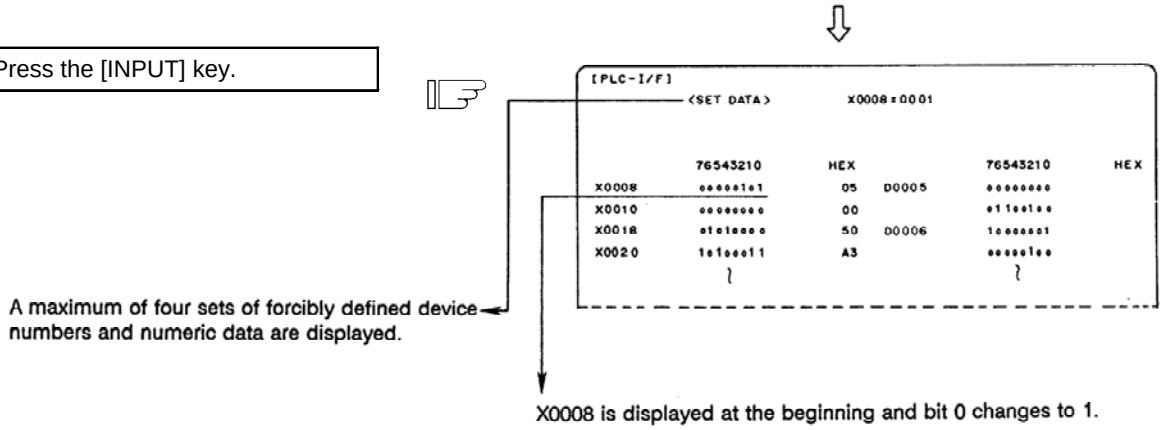
Set the device number to be forcibly defined in DEVICE ( ), definition data in DATA ( ), and 2 in MODE ( ), then press the [INPUT] key. The data is processed and forcibly defined.

(Example 3)

Set X8 in DEVICE ( ),
1 in DATA ( ), and
2 in MODE ( ).



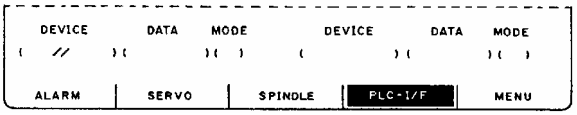
Press the [INPUT] key.



A maximum of four sets can be forcibly defined in modal type. If four sets have been defined and additional setting is made, forward feed is made and the latest four sets become effective.

(2) Cancel of forcibly defined data

Set two slashes // in DEVICE ( ).



Press the [INPUT] key.

The SET DATA field display becomes all blank and forcibly defined data is canceled.
(Setting of data 0 is not a cancel.
Data of 0 is forcibly defined.)

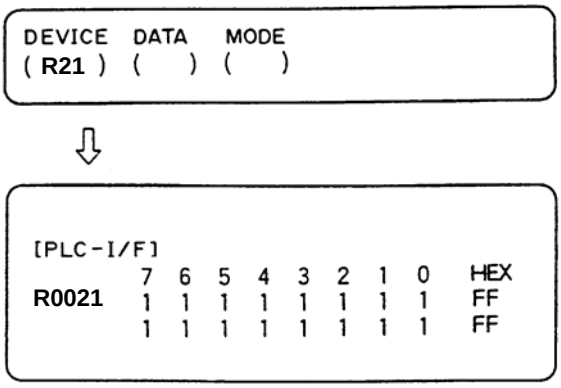
Forcibly defined data is also canceled by turning on/off power.

2.7.4.5 Diagnosis Executed When an Emergency Stop Status Occurs

When the NC is put in an emergency stop status, the message "EMG" (emergency stop) is displayed in the running status display area on the screen. At this time, the cause can be known.
Refer to "Appendix Messages during Emergency Stop (EMG)" for details of message.

(1) Check using the DIAGN screen

Set R21 in the DEVICE ( ) and then press [INPUT] key.



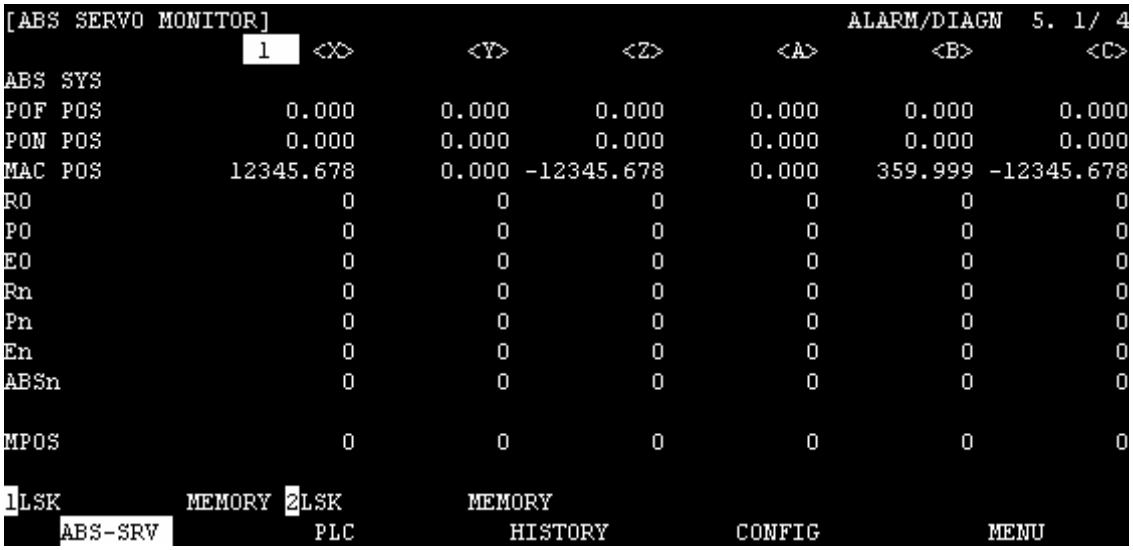
R0021 (The cause of the emergency stop)

File register (R)	F	E	D	C	B	A	9	8	7	6	5	4	3	2	1	0	← Bit
	Servo drive unit emergency stop output	Spindle drive unit emergency stop output	Door interlock, dog/OT arbitrary allocation device illegal	File area error	Built-in PLC program execution error	Q and Qr bus alarm arisen emergency stop	Power down handling error emergency stop	Contactor shutoff test	Emergency stop in the safety circuit	Built-in PLC program S/W emergency stop output is 1	Power supply external emergency stop	Control unit EMG connector Emergency stop state	Network error emergency stop	APLC emergency stop cause state	Target MELSEC stop state	Built-in PLC stop state	

2.7.5 Absolute Position Monitor

2.7.5.1 ABS Servo Monitor

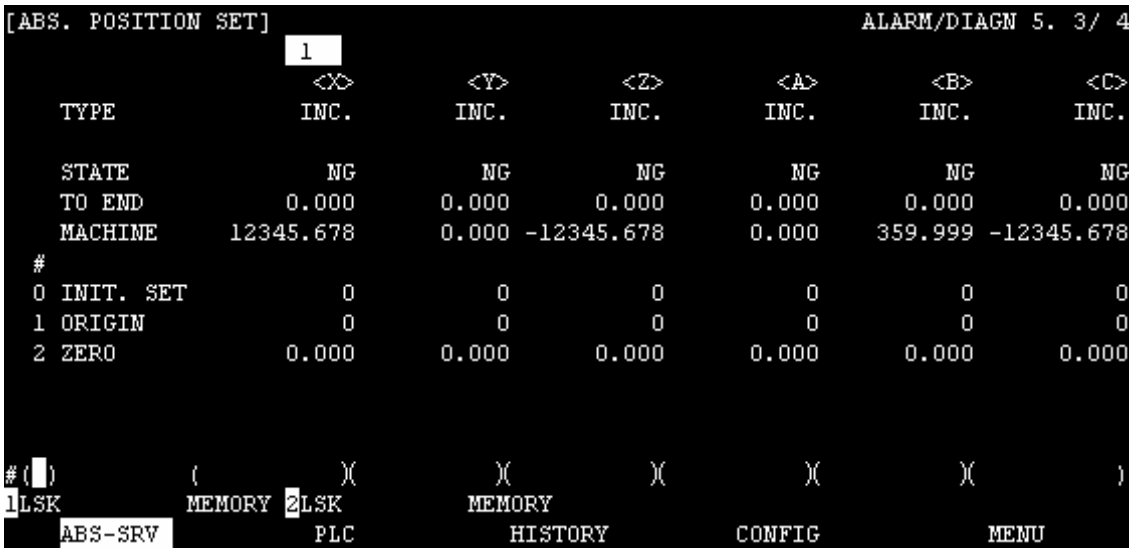
The servo state in the absolute position detection system is displayed. This can be used to confirm each detector data for the current machine value.

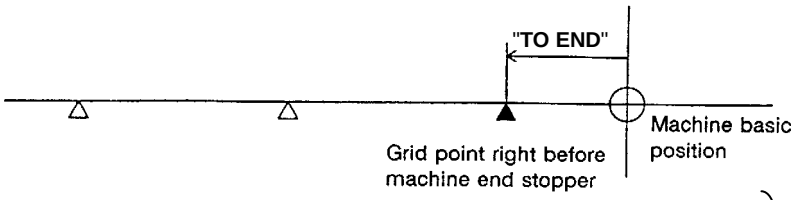
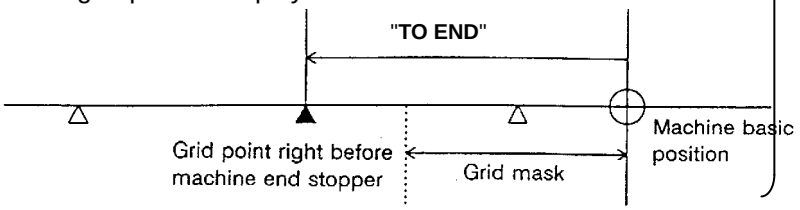


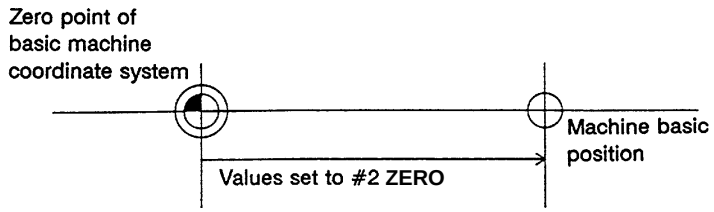
Display item	Display unit	Explanation
ABS SYS		The status of the absolute position detection system on the servo side is displayed. ES : Semi-closed encoder ESS : Semi-closed high-speed serial encoder INC : Incremental
POF POS	Command unit	The absolute position when the power is turned off is displayed.
PON POS	Command unit	The absolute position when the power is turned on is displayed.
MAC POS	Command unit	The coordinate value in the basic machine coordinate system is displayed.
R0		The multi-rotation counter value of the detector, saved when the basic point was set, is displayed.
P0	Output unit	The position in one rotation of the detector, saved when the basic point was set, is displayed.
E0		The absolute position error, saved when the basic point was set, is displayed.
Rn		The motor accumulated speed is displayed.
Pn		The position in one rotation is displayed. One rotation is divided into 4096, 8192 or 32768. The No. of divisions differs according to the detector.
En		The absolute position error when the power is turned off is displayed.
ABSn		The current absolute position is displayed.
MPOS		The offset amount of the MP scale at the power ON is displayed.

2.7.5.2 Absolute Position Initialization

Pressing the menu [ABS-SRV] displays the ABS SERVO MONITOR screen. When the [NEXT] key is pressed on that screen, the ABS POSITION SET screen will display. This screen is used to set the parameter data used to establish the zero point and to display the initialization.



Display item	Explanation
TYPE	The position detection system selected by the absolute position parameter is displayed. INC. : Incremental system DOG : Dog type absolute position detection STOPPER : Dogless absolute position detection machine end stopper method NO STOPPER : Dogless absolute position detection origin point alignment method
STATE	The progress of initialization is displayed, such as "OK" if the absolute position has been established or "NG" if the absolute position is lost.
TO END	The distance between the machine basic position and the first grid point is displayed.  If the first grid point is covered by the grid mask, the distance to the next grid point is displayed. 
MACHINE	"NOT PASS" is displayed until the machine passes a grid point after power on. After the machine passes the grid, the current machine position is displayed.

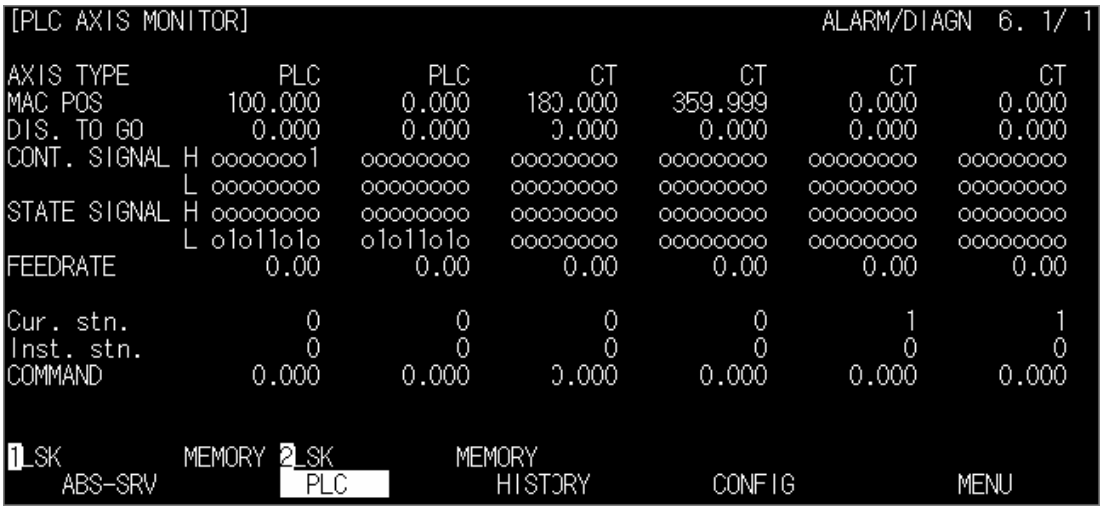
#	Parameter	Explanation	Setting range (unit)
0	INIT. SET	Select the axis for which absolute position initialization is to be performed: 0: Do not perform initialization. 1: Perform initialization. **(Note)** "0" is set when the power is turned on again.	0/1
1	ORIGIN	This parameter is valid when the origin point alignment method is used. Specify 1 when the machine has been positioned to the origin point. **(Note)** This parameter is invalid for other methods.	0/1
2	ZERO	This parameter is used to determine the basic machine coordinate system. Specify the coordinates of the machine basic position seen from the zero point of basic machine coordinate system or absolute position basic point. (Whether the machine basic position or absolute position basic point is used is determined by data specified for #2059 "zerbas" of the setup parameter.)  	±99999.999 (mm)

This section explains only the ABS POSITION SET screen. For details of the actual absolute position initialization and display contents, see the Setup Manual.

2.7.6 PLC Axis Monitor

PLC axis monitor screen displays the state of miscellaneous axis as well as PLC axis information.

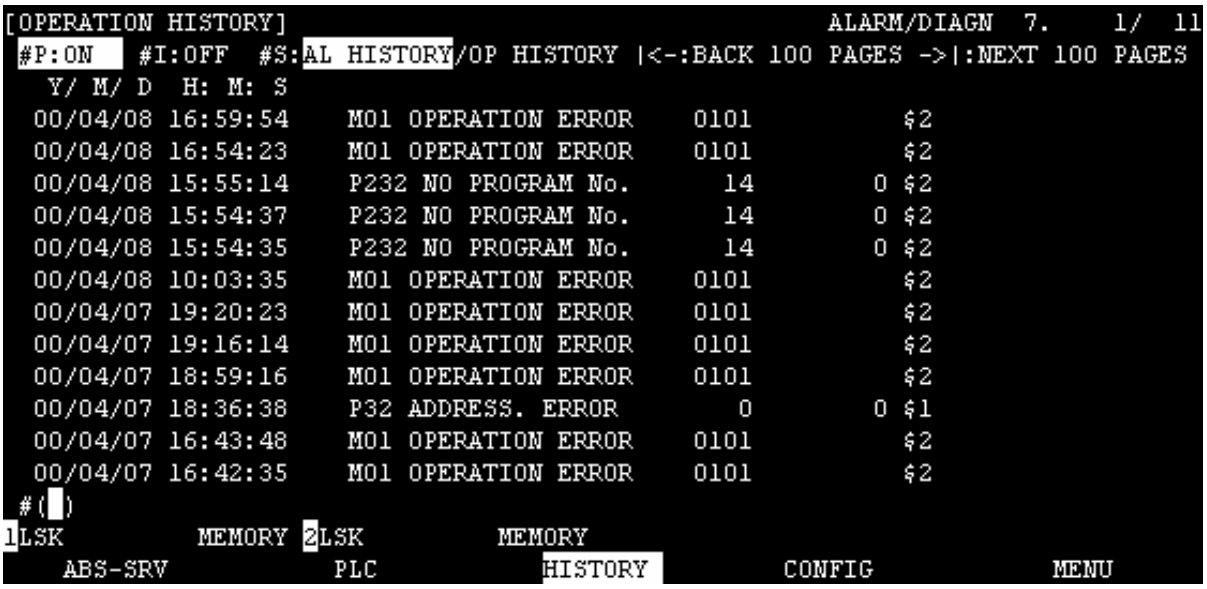
For NC axis, the position counter's inch/mm display can be set with "#1041 I_inch", and it can be set with "#1042 pcinch" for PLC axis.



Display item	Contents
AXIS TYPE	Displays axis type. (PLC: PLC axis, CT: Indexing axis
MAC POS	Displays machine position.
DIS. TO GO	Displays remaining distance. (Not displayed with JOG feed)
CONT. SIGNAL H, L	Displays the state of PLC axis command.
STATE SIGNAL H, L	Displays the state of PLC axis status.
FEEDRATE	Displays the feedrate.
Cur. stn.	Displays the station No. currently stopped.
Inst. stn.	Displays the automatic operation's command station No. or the nearest station No. where it is stopped with manual operation.
COMMAND	Displays the coordinate position corresponding with the target station No.

2.7.7 Operation History

Pressing the menu [HISTORY] displays the OPERATION HISTORY screen.
"Operation history with date and time" allows to store the date and time of occurrence and NC operation history when the NC alarm, key input or NC input/output signal status has been changed. The stored data is backed up after the power is turned OFF. This function is useful at troubleshooting.



The display items are as follows.

Display items		Details
#P: ON, #I: OFF		This displays the operation history mode (#P: ON, #I: OFF). The currently selected mode is highlighted. The operation history mode is retained after the power is turned OFF.
#S: AL HISTORY/ OP HISTORY		This shows the history display mode (#S: AL HISTORY/OP HISTORY). The currently selected mode is highlighted. The alarm history mode only displays the alarm history. The operation history mode displays the changes of alarm history, key history and input/output signal status. The history display mode is not retained after the power is turned OFF: when the power is turned ON again, alarm history mode is selected.
Operation history data		In the operation history data, time (yy/mm/dd hh:mm:ss) and messages are displayed.
	Message	This displays the changes of key history, alarm history and input/output signal status. The alarm history shows the data for each part system. "$1" and "$2", for example, indicate the 1st and 2nd part system respectively.

(Note) When a single part system is used, the alarm history does not include the part system display

2.7.7.1 Diagnosis of Operation History and the Occurrence Time of History (History Data)

The history data, which consists of the latest operation history and the date and time of occurrence, is displayed in the "OPERATION HISTORY" screen.

The history data is displayed in reverse chronological order. The page number is displayed on the right of the "ALARM/DIAGN" (the right top of the screen).Newer history is displayed in the earlier page. The number of history data pages depends on the number of history data.

Maximum number of histories

Number of alarm histories	168
Number of key histories	400
Number of input/output signal change histories	15132
Total number of operation histories	15700
Number of histories displayed per page	12
Number of history data pages (in alarm history mode)	14
Number of history data pages (in operation history mode)	1309

The history data page can be changed as follows.

Changing the pages

Key	Operation
NEXT	As the key is pressed, the page displays the older history. When the NEXT key is pressed at the last page, the first page will be displayed.
BACK	As the key is pressed, one page is turned at a time to display the newer history. When the BACK key is pressed at the first page, the last page will be displayed.
→|(right tab)	As the key is pressed, 100 pages are turned at a time to display the older history. When there is no page left by turning 100 pages, the last page will be displayed. As the key is held down, pages are turned continuously.
|←(left tab)	As the key is pressed, it goes back 100 pages from the current page at a time to display the newer history. When there is no page left by going back 100 pages, the first page is displayed. As the key is held down, pages are turned continuously.

2.7.7.2 Correspondence of Operation Keys and Key history

Correspondence of Operation key and Key history

Key name	Operation key	Key history
Function switches	MONITOR	MON
	TOOL PARAM	T/P
	EDIT MDI	E/M
	DIAGN IN/OUT	D/I
	SFG	SFG
	F0	F0
Menus	Menu 1	ME1
	Menu 2	ME2
	Menu 3	ME3
	Menu 4	ME4
	Menu 5	ME5
BACK key	BACK	BAK
NEXT key	NEXT	NXT
Input key	INPUT	INP
Key codes which are not registered		???
Data correction keys	DEL	DEL
	INS	INS
	CAN C.B	C.B
	CAN(Cancel)	CAN
Cursor keys	↑	CRU
	↓	CRD
	←	CRL
	→	CRR
	|←(left tab)	TBL
	→|(right tab)	TBR
Alphabet keys	Alphabets (A to Z)	A to Z
Number keys	Numerics (0 to 9)	0～9
Symbol keys	Symbols	
	(+、-、*、/、.)	+、-、*、/、.
	(, 、(、)、[、])	, 、(、)、[、]
	(EOB, =, #, Space, $, !)	;, =, #, SP, $, !

Correspondence of input/output signal change status and the change history

Input/output signal change status	Input/output signal change history
Input signal change (ON to OFF)	Y310
Input signal change (OFF to ON)	!Y310
Output signal change (ON to OFF)	X310
Output signal change (OFF to ON)	!X310
Simultaneous input	Y310+
Simultaneous output	X310+
Power supply ON	P.ON
Power supply OFF	P.OFF
Instantaneous power failure	ACDL
Emergency stop signal change (ON->OFF)	Emergency stop PLC
Emergency stop signal change (OFF->ON)	! Emergency stop PLC

When there are more than two signal changes at a time, changes are displayed with "+" except one of them.

(Example)
X002
X001+
X001+

It shows three signals were changed from ON to OFF simultaneously.

2.7.7.3 Input/Output Signal History Covering Range

Input/output signal history archives the changes of the signals shown in the table below.

Signal history covering range

Signal type	Signal history covering range
Input signal	X310 to XADF
Output signal	Y310 to YE7F
Machine contact Input signal	X000 to X2FF
Machine contact Output signal	Y000 to Y2FF
Emergency stop factor	R21 , R22

(Note 1) Changing the signals of R21/R22 indicates the emergency stop. A message, such as "EMG EMERGENCY PLC", will appear on Operation History screen.

(Note 2) Signals shown below are out of coverage.

Name of the signals which are out of coverage	Device No.
NC axis up-to-speed	(1st axis) X409 to (16th axis) X5E9

2.7.7.4 Suspending the Operation History Function

History data is continually updated by NC alarm occurrence, pressing the keys or NC input/output signal change. Enter "I" in #( ) on the screen to suspend the update of history data and save the current history data. Enter "I" on "#( )" in the screen to suspend the update of history data and save the current history data.

Enter "P" in "#( )" on the screen to update the history data again.

Suspending the operation history function

(1) Enter "I" in "#( )".

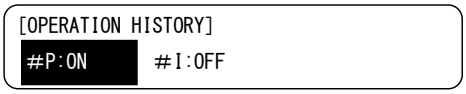
(2) Press the [INPUT] key. ☞

"#I:OFF" is reverse displayed and the operation history function will be suspended.

Executing the operation history function

(1) Enter "P" in "#( )".

(2) Press the [INPUT] key. ☞



"#P:ON" is reverse displayed and the operation history function will be continually updated.

2.7.7.5 Saving the Operation History Data

Operation history data can be saved in the save area for in case an error occurs. The save area, which is in the SRAM area and has the same capacity as the operation history data, retains the data after the power OFF.

Saving the operation history data

(1) Enter "B" in "#( )".	☞	Current operation history data is saved in the save area.

Besides the screen operation, the operation history data can also be saved under any of the following conditions. Once the data is saved under the conditions, another operation history data will not be saved unless the power is turned ON again.

- When an alarm has occurred in the drive unit (or in the CV unit)
- When history data save signal is ON (history data save signal: Y322)

2.7.7.6 Switching the History Display

Display of the history data can be switched as follows.

History display mode	Contents of history data displayed
Alarm history	Only alarm history is displayed.
Operation history	Alarm history, key history, and the input/output signal change history are displayed.

Alarm history mode is set when the power is turned ON.

When switching the history display, enter the data as follows.

Displaying the operation history (Switching from alarm history display to operation history display)

(1) Enter "S" in "#( )".

(2) Press the [INPUT] key. ☞

#S:AL HISTORY/OP HISTORY

"OP HISTORY" will be highlighted and the operation history will be displayed.

Displaying the alarm history (Switching from operation history display to alarm history display)

(1) Enter "S" in "#( )".

(2) Press the [INPUT] key. ☞

#S:AL HISTORY/OP HISTORY

“AL HISTORY” will be highlighted and the alarm history will be displayed.

2.7.7.7 Clearing the Operation History and the Occurrence Time of History (History Data)

Operation history and the history data including the date and time of occurrence are all cleared by pressing the "SHIFT", "CAN C.B", and then the "INPUT" key, which means entering the cancel, in the "OPERATION HISTORY" screen. After that, the operation history and the date and time of occurrence will show blank. The history page number will show "1/1".

2.7.7.8 Setting and Displaying the Time and Date of Occurrence of Operation History Data

In operation history, 100 years from 1970 can be displayed. The last two digits of the year is displayed. Date and time are set in the "TIME" screen ("COMMAND" in the "MONITOR" screen on page 3/3).

Be sure to clear the operation history and the history data including the time and date of occurrence after changing the date or time setting. Otherwise, the history data might not be displayed in reverse chronological order.

2.7.7.9 Outputting the Storage Data for Operation History

CNC data input/output function of GOT enables to output the operation history that occurred in the NC operation.

Refer to "CNC Data Input/Output" for CNC data input/output.

Output data

Output data	Alarm history Input/output signal change history Key history
File name	TRACE.TRC
Output device	Standard CF card/ Expansion memory card

(Note 1) Neither inputting nor deleting the operation history file is allowed.

(Note 2) Output data is not affected by the history display mode change. Operation history data will still be output as shown above in the alarm history display mode.

(1) Outputting

Output a operation history file to a memory card as follows.

(a) Display the "Data in/out" screen.

(b) Select "PROGRAM COPY".

(c) Select the device name, directory, and file name in the area A as follows.

Device: CNC
Directory: Maintenance data
File name: TRACE.TRC

(d) Select "built-in CF card" or "extend memory card" for a device name in the area B.

(e) Press the "Exec" button.

(2) Output data format

Output data will be written to a memory card as follows.

(year, month, day) [SP] (hour, minute, second) [SP] (Key code) [CR] [LF]
(or error message)
(or signal name ON/OFF)

[SP] ... space

[CR] ... Carridge return

[LF] ... line feed

(Note) Output data format is ASCII or SHIFT_JIS code. (Example)

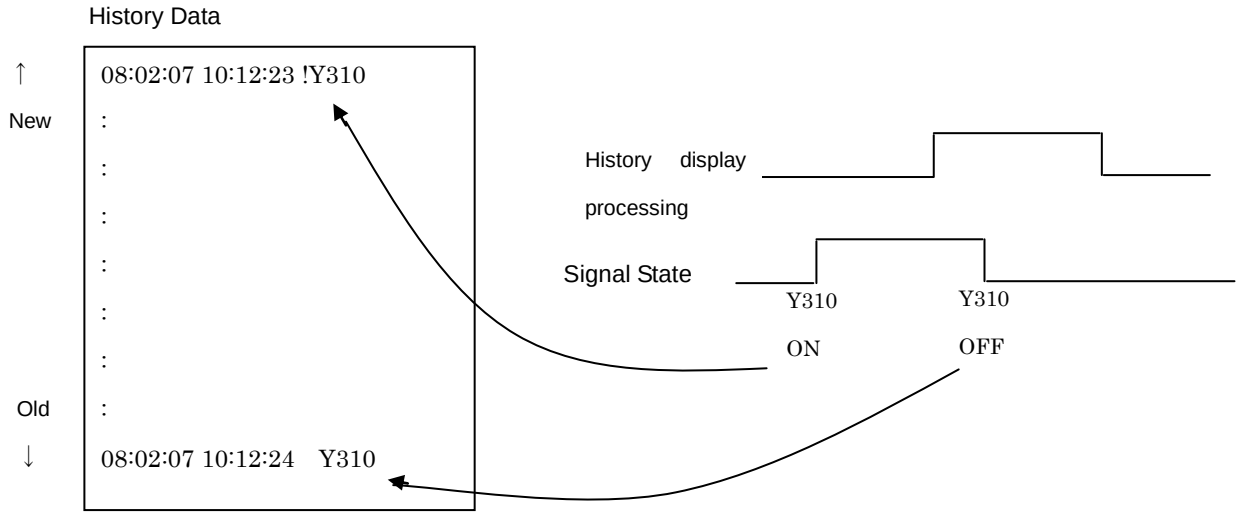
```
TRACE [CR] [LF]
00/01/01 [SP] 13:59:58 [SP] CAN [CR] [LF]
00/01/01 [SP] 13:59:56 [SP] Y220 [CR] [LF]
00/01/01 [SP] 13:59:55 [SP] !Y220 [CR] [LF]
00/01/01 [SP] 13:59:02 [SP] P153 I.F ERROR $1 [CR] [LF]
                    .
                    .
00/01/01 [SP] 13*00:56 [SP] Y218 [CR] [LF]
%
```

2.7.7.10 Precautions

- If a new history occurs while the OPERATION HISTORY screen is selected, the history will be saved but will not be displayed. The history will be displayed by pressing the "HISTORY" menu key to refresh the "OPERATION HISTORY" screen or opening another screen and then returning to the "OPERATION HISTORY" screen.
- When operation histories occur at the same time, they will be displayed in the following order (older to newer): P.OFF, input/output signal change history, key history, alarm history and P.ON. The operation history occurrence time is updated every one second.
- Stop code is not saved in the operation history.
- The following NC alarms are not saved in the operation history.

No.	Details
M01 0004	External interlock axis found
M01 0005	Internal interlock axis found
M01 0109	Block start interlock
M01 0110	Cutting block start interlock

- The operation history data can be input/output without setting the operation history mode to "#I: OFF". However, when operating the remote monitor while outputting the operation history in the CNC data input/output screen on GOT, set the operation history mode to "#I: OFF" beforehand to prevent the output history data from being rewritten.
- Displaying the maximum or nearly maximum number of histories will take some time. Some histories that occurred in the history display process might be shown at the last of the history data. When this happens, suspend the history function and display the history screen again.

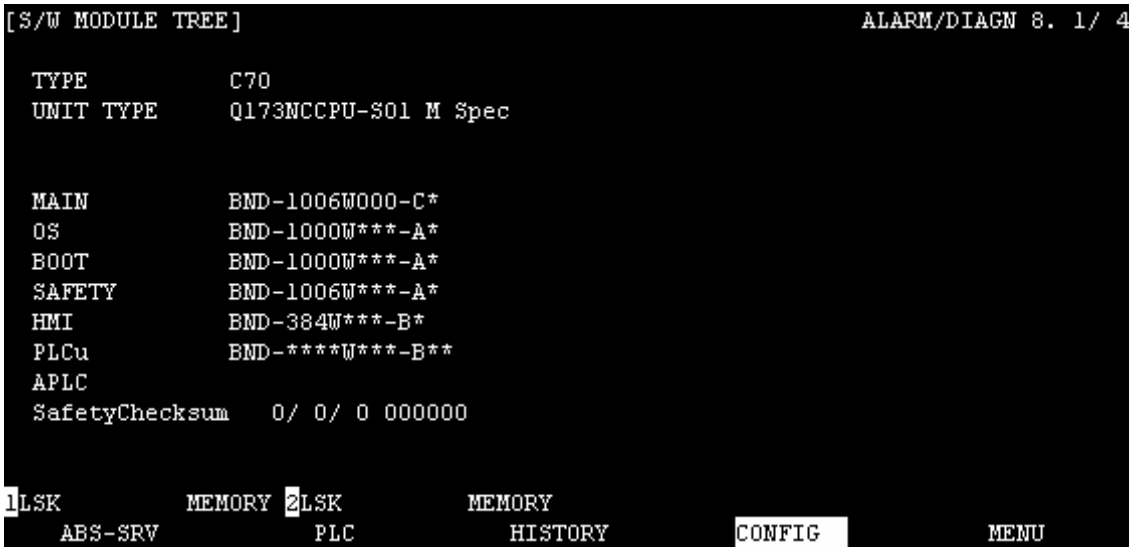


2.7.8 System Configuration

The software control number and hardware mounting state are displayed on this screen. Refer to this screen when contacting the service department in case of trouble, etc.

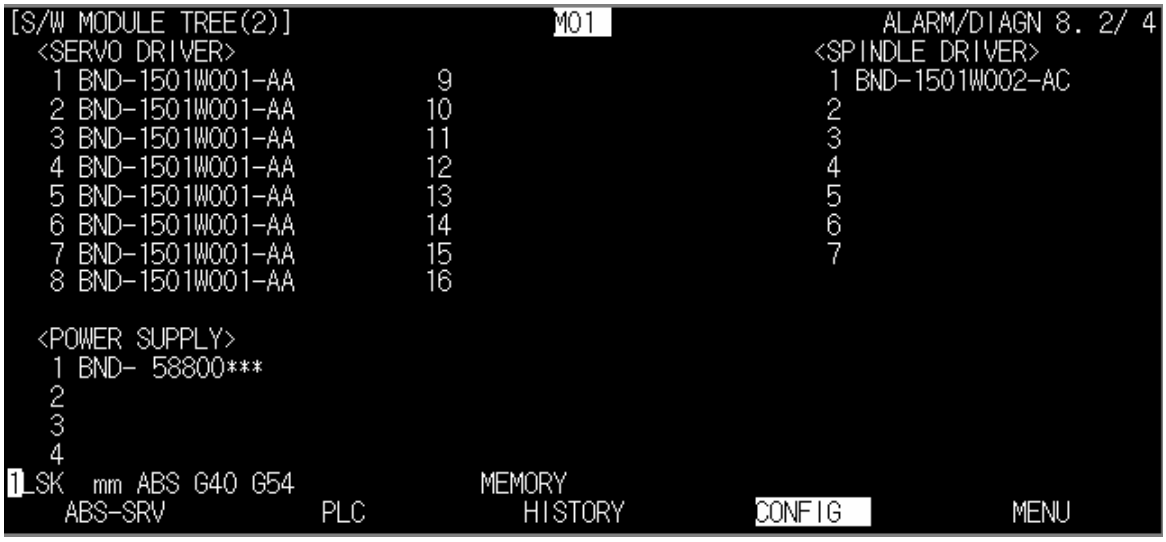
2.7.8.1 S/W Module Tree (NC System)

Pressing the menu [CONFIG] displays the S/W MODULE TREE screen.



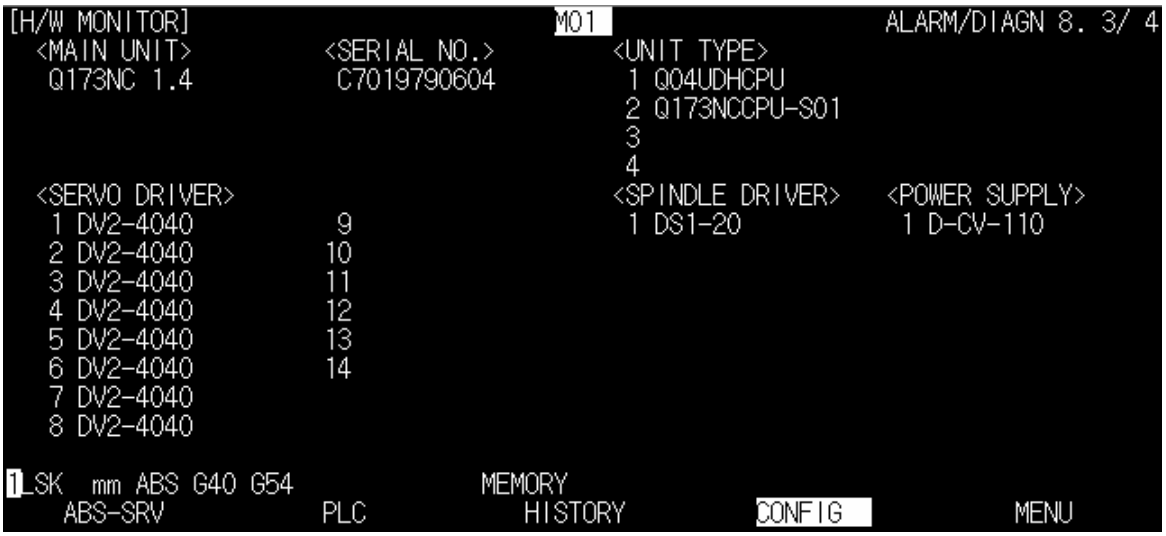
2.7.8.2 S/W Module Tree(2) (Drive Unit)

Pressing the [NEXT] key on the S/W MODULE TREE screen displays the S/W MODULE TREE(2) screen.



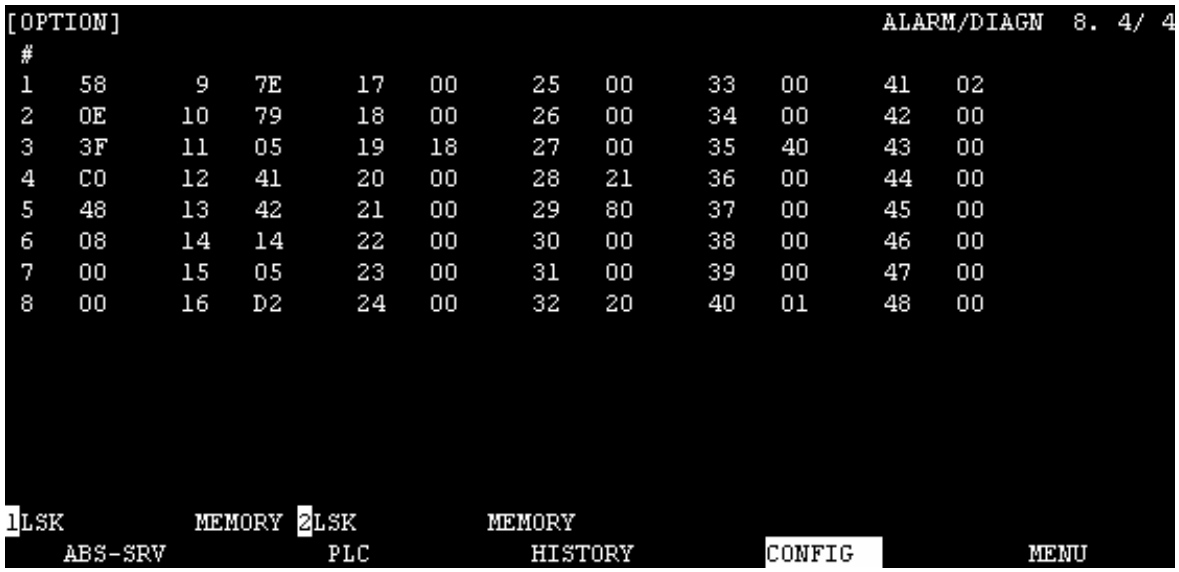
2.7.8.3 H/W Monitor

Pressing the[NEXT] key on the S/W MODULE TREE(2) screen displays the H/W MONITOR screen.



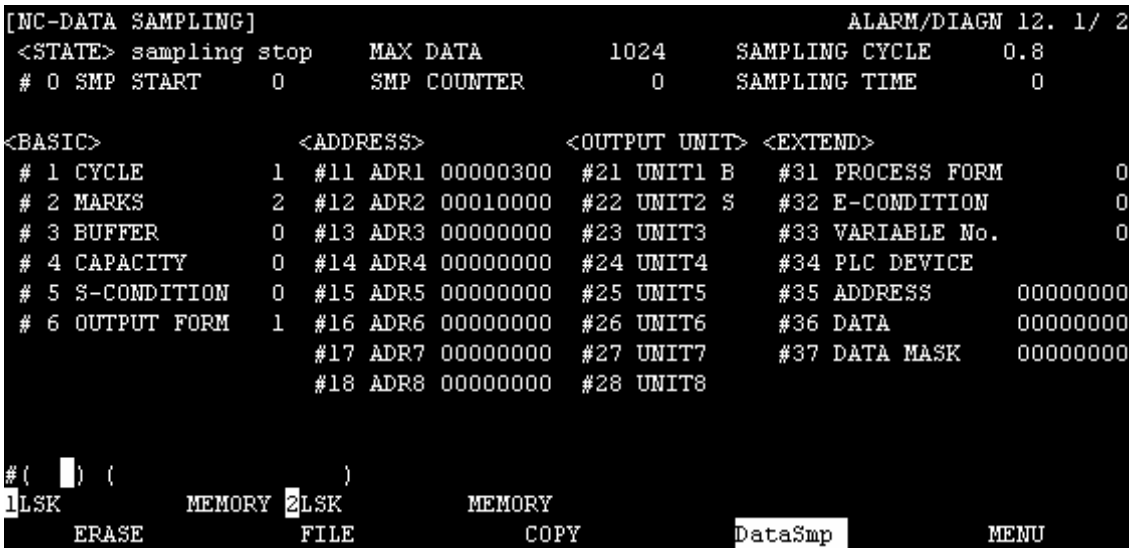
2.7.8.4 OPTION

Pressing the [NEXT] key on the H/W MONITOR screen displays the OPTION screen. You can see the current settings for the option parameters.



2.7.9 NC Data Sampling

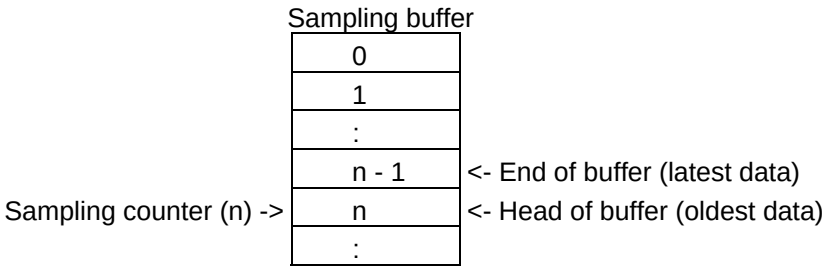
Sampling start/stop, sampling state display, and the sampling parameters necessary for sampling are set in the data sampling screen. The NC internal data (speed output data from NC to the drive unit, or feedback data from the drive unit, etc.) can be sampled.
The sampled data can be output.



Specifications for sampling

Item	Specifications
Sampling cycle	0.8ms ∗ set value (1 to 255)
Number of sampled axes	Servo axes: 1 to 16 Spindle: 1 to 7
Number of sampling channels	1 to 8 points
Number of sampled data items	Max. 1,310,720 points **(Note)** This is a whole number of data items. The more sampling channels are provided, the less data items will be handled by a channel.

2.7.9.1 Display Items

<table>
<tr><th>Item</th><th>Details</th></tr>
<tr><td>MAX DATA</td><td>Displays the maximum number of data to be sampled.
The number of data is usually calculated from the set value of "#4 CAPACITY".
Note that the number is at most 50% of DRAM available memory.</td></tr>
<tr><td>SMP COUNTER</td><td>Displays the position in the sampling buffer during the sampling process.
When the ring buffer is valid, the head of the buffer is shown in the counter at the end of sampling.
 
</td></tr>
<tr><td><STATE></td><td>Displays the current sampling status.
"sampling" : Sampling is being executed.
"sampling stop" : Sampling is not being executed or has been completed.
"trigger wait" : This indicates a state from when "#0 SMP START" is set to "1" to when a sampling start trigger is detected and the sampling starts, without the condition that "0" (manual start) is set for "#5 S-CONDITION".
"ready to stop" : This indicates a state from when an end trigger is confirmed to when the buffer reaches full, under the condition that "one-shot" or "repetition" is selected in "#31 PROCESS FORM".
(The sampling stop state will be entered after the buffer has reached full.)</td></tr>
<tr><td>SAMPLING CYCLE</td><td>Display the sampling cycle.
It will be determined with "#1 CYCLE".
(It will be displayed to the first decimal point.)</td></tr>
<tr><td>SAMPLING TIME</td><td>Display the amount of buffer currently set and the amount of time (sec) to become buffer full which depends on the number of channels.
(It will be rounded off to a decimal.)</td></tr>
</table>

2.7.9.2 Parameters

Parameters for NC data sampling are set on this screen.

(Note 1) Parameter output is not executed for the data set on this screen.

(Note 2) "#0 SMP START" will return to "0" (sampling stop) at the power ON, while the data #1 to #37 will be retained. However, When "#100 P.ON START" is set to "1", "#0 SMP START" will be set to "0" when the power is turned ON and the sampling starts.
The parameters, which are rewritten from MS Configurator via API, will not be kept. The data before adjustment/measurement will return when the power is turned ON again.

(Note 3) Setting the parameters #31 to #37 is basically not necessary. These parameters are for expanding functions.

(Note 4) The parameters cannot be set when "#1224 aux08/bit0" is set to "0", "#1164 ATS" is set to "1" on the base specifications parameter screen.
The error message "DATA PROTECTING" will occur when the setting is attempted.

(Note 5) The parameters except #0 cannot be set unless <STATE> shows "sampling stop".
The error "SETTING NOT POSSIBLE" will occur when the setting is attempted.

(Note 6) The parameters except #0 cannot be changed while a machining program is running.
The error message "PROGRAM RUNNING" will occur when the change is attempted.

<table>
<tr><th>#</th><th>Item</th><th>Details</th><th>Setting range</th></tr>
<tr><td>0</td><td>SMP START</td><td>Set "1" and then press "INPUT" to start sampling. The value will automatically become "0" when the sampling is completed.
0: Sampling stop:
Sampling has not been executed or has been completed.
(This is usually not to be set by a user.)
1: Sampling start:
Sampling is being executed or in the "trigger wait" state.
9: Sampling forced end:
The sampling will stop immediately when this value is set during the sampling process. Then the value will become "0".

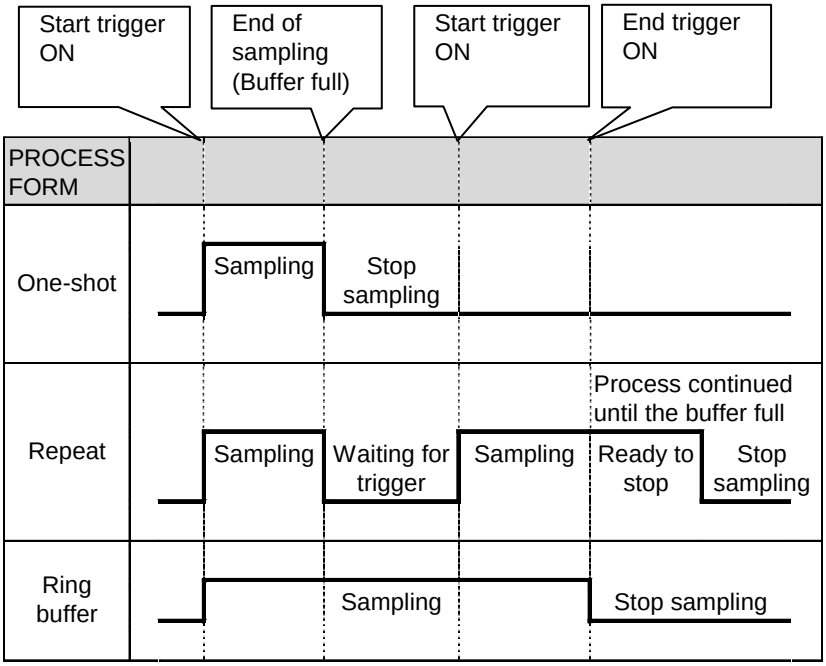
(Note 1) If a user sets "0", the sampling will stop under the condition that the "trigger wait" state or "ring buffer" type is selected.
Without the condition, the sampling state will not be changed.
(Note 2) The operation when "1" is set differs according to the "#5 S-CONDITION" setting.
<table>
<tr><th>"#5 S-CONDITION"</th><th>Operation
when "1" is set for "#0 SMP START"</th></tr>
<tr><td>"0" (manual start)</td><td>Sampling starts immediately.</td></tr>
<tr><td>Others</td><td>System waits for a trigger.
Sampling starts when the trigger is detected.</td></tr>
</table>
</td><td>0,1,9</td></tr>
<tr><td>1</td><td>CYCLE</td><td>Set the sampling cycle.
Cycle = 0.8ms ∗ setting value
(Example) When set to "1": 0.8ms cycle, When set to "2": 1.7ms cycle

(Note 1) When there are many part systems and axes, the cycle is "1.7ms * set value".</td><td>1 to 255</td></tr>
<tr><td>2</td><td>MARKS</td><td>Set the number of channels for sampling.</td><td>1 to 8</td></tr>
<tr><td>3</td><td>BUFFER</td><td>Not used.</td><td>0</td></tr>
<tr><td>4</td><td>CAPACITY</td><td>Set the capacity (maximum number of data) of the buffer to use.
Maximum number of data = (Set value + 1) ∗ 1024 points</td><td>0 to 1279</td></tr>
</table>

#	Item	Details	Setting range
5	S-CONDITION	Select the condition to start sampling. 0: Manual start Sampling starts when "#0 SMP START" is set to "1". 1: Variable No. Sampling starts when "#33 VARIABLE No." is set to a non-"0" (nor null) value in a machining program. 2: PLC device start Sampling starts when the signal set in "#34 PLC DEVICE" turns ON. **(Note)** When "1" is set • Not the variable value itself but "setting a non-"0" (nor null) value for the variable in a program" works as a trigger. Therefore, setting a variable value on COMMON VARIABLE screen or with the PLC window does not work as a start trigger. • "Setting the variable to non-0 (nor null) value in a program" is canceled (the start trigger is turned OFF) by turning ON the reset (RST1/RST2/RRW) or SA signal. • If a calculation is used for setting a common variable, which is a floating point data, the result "0" may not be recognized due to the calculation error. • In multi-part system, a start trigger will be turned ON when any one of the part systems meet the conditions.	0 to 4
6	OUTPUT FORM	Select the data format when outputting the sampled data in text style. 0: Outputs in decimal figure (64 bits, with sign). 1: Outputs in hexadecimal (32 bits, in 8 digits). **(Note 1)** When "1" is set, lower 32 bits of the 64-bit sampled data are output. **(Note 2)** Refer to "2.7.9.4 Data Output Format" for the details of the output format of sampled data.	0,1
11 to 18	ADR1 to ADR8	Set the addresses for sampling. (Note) Refer to "2.7.9.3 Address Designation" for details of the setting range.	(Note)
21 to 28	UNIT1 to UNIT8	Not used. Set to "0" (blank).	-

#	Item	Details	Setting range
31	PROCESS FORM	Set the sampling process type. 0: One-shot (The sampling does not stop until the buffer gets full.) 1: Repeat valid (After the process ends (the buffer gets full), the "trigger wait" state will be entered again.) 2: Ring buffer valid **(Note 1)** When a value except "0" is set for this item, "#32 E-CONDITON" must also be set. Otherwise, the sampling cannot be stopped until forcibly ended. **(Note 2)** The output range of the sampled data differs according to the process type. [When "0" or "1" is set] Output data is from the head to the end of the sampling buffer. When the sampling has been forcibly ended, however, the data sampled until then is output. [When "2", ring buffer valid is set] Output data is from the oldest sampling data to the data at the sampling end. (Ex) "N" stands for a number of sampled data in one cycle of buffer. "n" stands for a sampling counter at the end of sampling. • When the data have not been sampled for one cycle of buffer The sampled data 0 to n-1 are output. • When the data have been sampled for one or more cycles of buffer The sampled data n to N-1, and then 0 to n-1 are output.	0 to 2

#	Item	Details	Setting range
32	E-CONDITION	Select the condition for ending the sampling process. 0: Sampling completed Sampling ends upon buffer full when "one-shot" is selected in "#31 PROCESS FORM". 1: Variable No. Sampling ends when "#33 VARIABLE No." is set to "0" (or null) value in a machining program. 2: PLC device start Sampling ends when the signal set in "#34 PLC DEVICE" is turned OFF. **(Note 1)** When "1" is set • Not the variable value itself but "setting the variable to 0 (or null) value in a program" works as a trigger. Therefore, setting a variable value on COMMON VARIABLE screen or with the PLC window does not work as an end trigger. • "Setting the variable to non-0 (nor null) value in a program" is canceled (the end trigger is turned ON) by turning ON the reset (RST2/RRW) which initialize the modal or SA signal. • If a calculation is used for setting a common variable, which is a floating point data, the result "0" may not be recognized due to the calculation error. • In multi-part system, an end trigger will not be turned ON until all part systems meet the conditions. **(Note 2)** Setting "0" for the end condition is invalid unless one-shot is selected for "#31 PROCESS FORM". (The sampling does not end at buffer full). (Continued to the next page)	0 to 4

#	Item	Details	Setting range
32	E-CONDITION	(Continued from the previous page) **(Note 3)** Even if the ending conditions are satisfied, the subsequent flow will differ according to the value set in "#31 PROCESS FORM". Refer to the following chart for details. (The chart is also applied when the sampling is manually ended.)  Flow of sampling process	0 to 4
33	VARIABLE No.	Set the No. of a variable that triggers the start/end. 0: System variable (#1299) Others: Designated common variable (#100 or later, #500 or later) **(Note 1)** Setting #5 S-CONDITION" or "#32 E-CONDITION" to "1" (variable No.) makes the variable set in this parameter trigger the start/end. **(Note 2)** "E02 DATA OVER" will occur when the entered No. does not exist.	0 to 999
34	PLC DEVICE	Set the PLC device that triggers the start/end. Add "∗" at the head of its No. to designate a normally closed contact device. 0(blank): Sampling start/stop signal (Y321) Others: Designated device **(Note 1)** Setting "#5 S-CONDITION" or "#32 E-CONDITION" to "2" (PLC device) makes the device set in this parameter trigger the start/end. **(Note 2)** "E01 SETTING ERROR" will occur when the entered device No. does not exist. "E02 DATA OVER" will occur when the No. is out of the setting range.	(*)X0 to XAFF (*)Y0 to YE7F (*)B0 to 1FFF

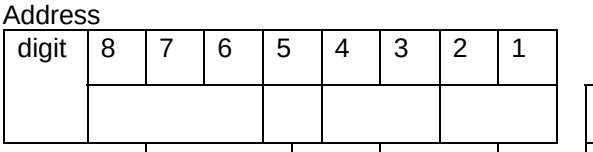
#	Item	Details	Setting range
35	ADDRESS	Not used.	-
36	DATA	Not used.	-
37	DATA MASK	Not used.	-
100	P.ON START	Sampling starts automatically immediately after NC's power is turned ON. **(Note)** When "1" is set, sampling starts every time the power is turned ON until "0" is set again.	0,1
111 to 118	SIZE	Not used.	-
121 to 128	SHIFT	Designate the number of right-shifts for each channel, which will be executed on a sampling data when outputting the sampling data. **(Note 1)** Shift is executed for the value after converting the output unit. **(Note 2)** Sign extension is executed when shifting.	0 to 63
131 to 138	MASK	Designate a mask applied to the sampling data (execute AND) for each channel when outputting the sampling data. However, the mask is not applied if 0(0x0000000000000000) is set. **(Note 1)** Set "1" to the BIT to remain in the mask. **(Note 2)** Mask is applied to the data after shifting. **(Note 3)** On the screen, the data is displayed separately in higher (H) 32 bit and lower (L) 32bit although the 64 bit ( ) are input continuously to the setting area to apply the mask.	0x0000000000000000 to 0xFFFFFFFFFFFFFFFF
141 to 148	TYPE	Designate the data type (with/without sign, notation system of base n) for each channel when outputting the sampling data. S: Decimal number with sign U: Decimal number without sign H: Hexadecimal number 0 (Blank): Handled in the same way as "S" is set. **(Note 1)** This is invalid if "#6 OUTPUT FORM" is set to "1" (exclusive type for synchronous tapping tool).	S,U,H,0

2.7.9.3 Address Designation

The following three methods are available for setting the address: designating the prepared data with its index No.; setting the actual address; searching for the head address of the symbol and set it automatically. The applied method is automatically recognized.
A different designation method can be used for each address.
(Note) If the number of addresses exceeds the "#2 MARKS" value, the set address will be ignored.

(1) Index No. designation
The lower 6 digits of the 8-digit hexadecimal address are designated as index No.
The index No. is fixed regardless of the axis configuration.
The following shows how to set the lower 6 digits of address.
(No setting for the higher 2 digits is interpreted as "00".)
Allocation of digits is as follows.

Address



digit	Application	Setting range
2,1	Target No.	Refer to "Target No. List".
4,3	Servo axis No.	01 to 10 (in hexadecimal)
5	Spindle No.	1 to 7
8,7,6	Not used	"0" when the setting is omitted

(Note1) In the following cases, the designated index No. will be treated as an illegal value and the sampling will not be executed. (The sampled data will be "0".)
- When designated spindle No., servo axis No. or target No. was out of the setting range.
- When both spindle No. and servo axis No. were set.
- When a value except "0" was set in the unused part.

(Note 2) If the number of addresses set is more than the setting value for "#2 MARKS", it will be ignored.

Target No. List (hexadecimal)

Target No.	Servo axis, PLC axis	Spindle
00	Position FB (Note 1)	Position FB (Note 1)
01	Position command (Note 1)	Position command (Note 1)
02	Number of motor rotations (Note 2)	Number of motor rotations (Note 2)
04	Load current (Note 3)	Load current (Note 4)
06	Droop (Note 1)	Droop (Note 1)
40 to 45	Control input 1 to 6	Control input 1 to 6
48 to 4D	Control output 1 to 6	Control output 1 to 6
7E	—	Spindle position one rotation data
7F	—	Spindle position FB one rotation data
80	—	Cycle counter
81	Machine position	—
82	F Δ T	—
84	—	Synchronous tapping error width
85	—	Synchronous tapping error angle
86	Disturbance load torque	Disturbance load torque

(Note 1) Position FB, position command, and droop are output as an interpolation unit.
EX) When "#1003 iunit" is B (1μμm) and output result is 2000:
20000 * 0.5 (interpolation unit) = 1000μm = 1mm

(Note 2) Output unit for the number of motor rotations is 0.01 r/min.

(Note 3) The same value as the load current displayed in the servo monitor screen is sampled for the load current value of the servo and PLC axes. Output unit is %.

(Note 4) The same value as the load displayed in the spindle monitor screen is sampled for the load current value of the spindle. Output unit is %.
Droop's sampling result is twice the value of a droop displayed in the servo/spindle monitor screen of CNC monitor.

Setting example (of index No.)

Servo axis	1st axis	2nd axis	···	15th axis	16th axis
Feedback position	000100	000200	···	000F00	001000
Commanded position	000101	000201	···	000F01	001001

Spindle	1st axis	2nd axis	···	6th axis	7th axis
Feedback position	010000	020000	···	060000	070000
Commanded position	010001	020001	···	060001	070001

Setting examples for each purpose are shown below.
• For synchronized tapping: 3rd servo axis feedback (000300) - 1st spindle feedback (010000)
• For high-accuracy (roundness): 1st servo axis feedback (000100) - 2nd servo axis feedback (000200)
• For spindle synchronization: 1st spindle feedback (010000) - 2nd spindle feedback (020000)

2.7.9.4 Data Output Format

The following shows the format when the sampled data is output in text style.

(1) Output in decimal figure
Sampled data is output in decimal figure when "#6 OUTPUT FORM" is set to "0".
The data of all channels are output to one line, with each data delimited by comma (","). CR+LF is output to the end of the line.

(Example) When the number of channels is "3" , the output unit for all channels is "B(1 μ m)" and the form for all channels is "S (decimal with sign)".
Stops when 1ch is 100mm, 2ch is -100mm and 3ch is 50mm

	1CH	2CH	3CH
1st data	200000,	-200000,	100000
2nd data	200000,	-200000,	100000
3rd data	200000,	-200000,	100000
4th data	200000,	-200000,	100000
5th data	200000,	-200000,	100000
6th data	200000,	-200000,	100000
7th data	200000,	-200000,	100000
	・・・	・・・	・・・

(2) Output in hexadecimal figure (in 8 digits)
Sampled data is output in 8-digits hexadecimal figure when "#6 OUTPUT FORM" is set to "1".
The output data is the sampling buffer dumped in the length of the long type data (32bits).
The data is output to one line per channel. CR+LF is output to the end of the line.

(Example) When the number of channels is "3" and the output unit for all channels is "B(1 μ m)"
Stops when 1ch is 100mm, 2ch is -100mm and 3ch is 50mm

1CH 1st data	00030D40
2CH 1st data	FFFCF2C0
3CH 1st data	000186A0
1CH 2nd data	00030D40
2CH 2nd data	FFFCF2C0
3CH 2nd data	000186A0
1CH 3rd data	00030D40
	・・・

2.7.9.5 Outputting the Data

Use either of the following screens to output the sampled data.
• CNC Data In/Out screen (Refer to "3. CNC Data Input/Output" in "III. MAINTENANCE")

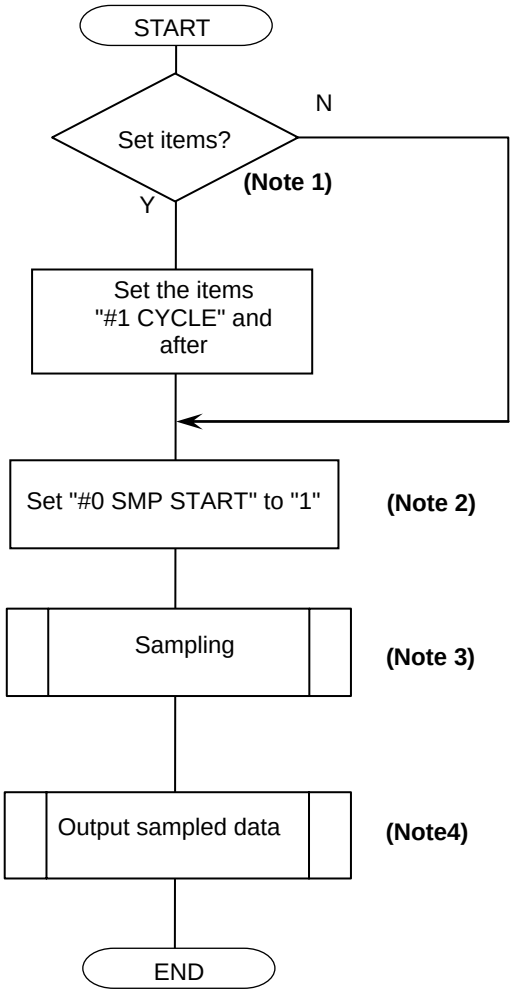
(Example 1) Outputting data with CNC data input/output screen
(1) Select CNC for "Device" on top of the screen.
(2) Select "Mainte data" for "Directory".
(3) Select "NCSAMP.CSV".
(4) Press "Exec" to transfer the file.

(Example 2) Outputting data with the remote monitor tool
(1) Select [File] -> [Transfer] -> [Maintenance] from the menu bar to display the dialog box for sending/receiving files.
(2) Designate "M01:\LOG\" for the directory.
(3) Designate the "NCSAMP.CSV" file.
(4) Press the "Receive" button. Then select the save destination.
(5) Pressing the "Save" button starts the file transfer.

(Note 1) Data cannot be output during the data sampling (when the <STATE> display is other than "sampling stop" on the NC data sampling screen).
(Note 2) Changing the value of "#2 MARKS", "#4 CAPACITY" or "#31 PROCESS FORM", the output data of which will be changed after the sampling has been completed, resets the sampling counter to "0". The sampled data, if output after the change, will have the size 0.

2.7.9.6 Flow of the Operation

Refer to "2.7.6.2 Parameters" for the details of the setting items.



(Note 1)
Go to N for using previously set values.

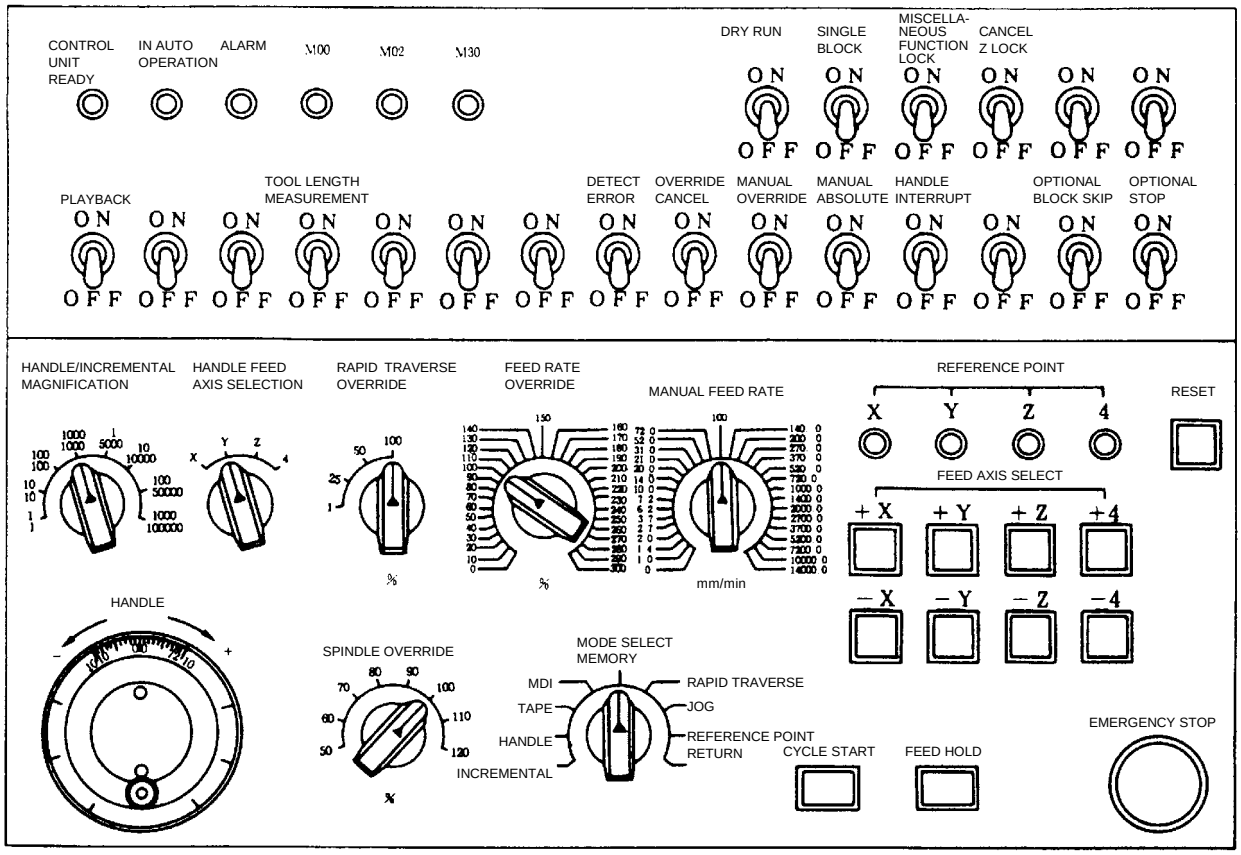
(Note 2)
In the manual start mode
... The sampling is immediately executed.
In other mode
... The trigger standby state is entered.
The sampling starts after the start trigger is detected.

(Note 3)
Refer to "#32 E-CONDITON" in "2.7.6.2 Parameters".

(Note 4)
Refer to "2.7.6.5 Outputting the Data".

II. MACHINE OPERATIONS

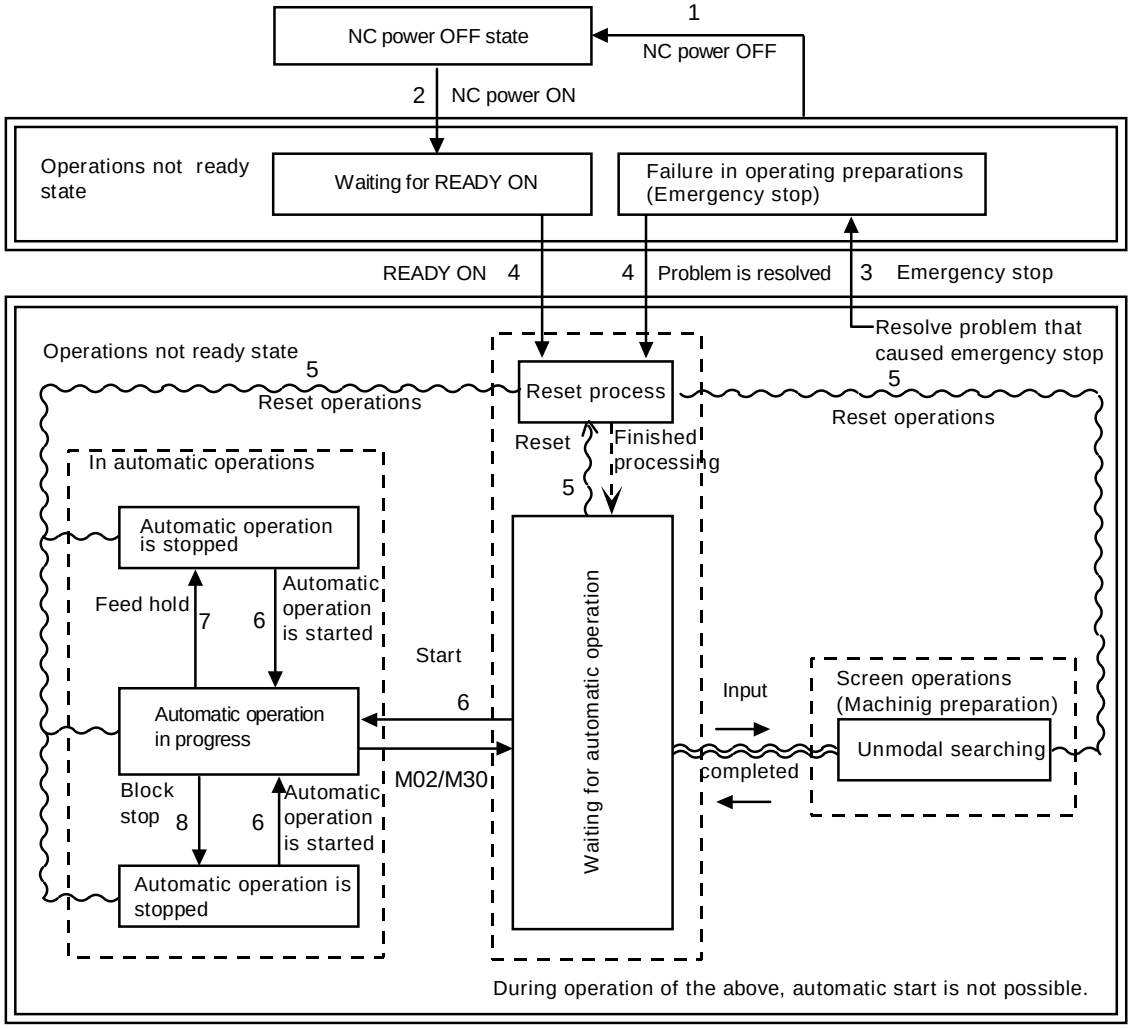
This chapter explains the functions and operation method of the machine operation switches for operation (automatic operation and manual operation) by using the illustration of the machine operation panel.
The actual machine operation and motion vary from one minute to another. Refer to the operation manual issued by the machine tool builder. Use this chapter for reference.



1. Operation State

1.1 Operation State Transition Diagram

The NC unit operation state changes momentarily according to the program contents or signals from the operation panel or machine. The controller roughly classifies the operation state into power OFF, not ready, and ready.
NC operation is enabled only in the operating preparation ready state. The operating preparation ready state is furthermore classified as shown below.
Manual mode operation is enabled in the operation complete state.



1.2 Power OFF

The power OFF state means that no power is supplied to the control circuit.

(1) From any other state to power OFF (transition 1.)

- When the POWER OFF switch of the setting display unit is pressed.
- When POWER OFF signal is input from the machine.
- When power supplied from the machine to NC unit is turned OFF.

1.3 Not Ready

The operation preparation not ready state means that the system is not ready to run because of the NC unit itself or because of the machine, even though power is supplied to the NC unit control circuit. The READY lamp on the setting display unit is OFF.

(1) From power OFF to not ready (transition 2.)

- When the POWER ON switch of the setting display unit is pressed.
- When POWER ON signal is input from the machine.

(2) From ready to not ready (transition 3.)

- When EMG (emergency stop) is displayed on the setting display unit screen.
- When any of the following alarms is displayed on the setting display unit screen;
  Servo alarm, spindle alarm, MCP alarm, system alarm

1.4 Ready

The operating preparation ready state means that power is supplied to the NC unit control circuit and the system is ready to run. The READY lamp on the setting display unit is ON.
The state is furthermore classified into the following four states.

1.4.1 Reset

The reset state means that the NC unit is reset.

(1) From not ready to reset (transition 4.)
This state is also called initial state.

(2) From another ready to reset (transition 5.)
- When the RESET key on the setting display unit is turned ON.
- When external reset signal is input from the machine.
- When M02 or M30 is executed (depending on the machine specifications).

1.4.2 Automatic Operation Start

The automatic operation start state means starting in automatic mode. The AUTO START BUSY lamp on the machine operation panel is ON.

(1) From another ready to automatic operation start (transition 6.)
When the CYCLE START switch on the machine operation panel is pressed in automatic mode.

CAUTION
- Stay out of the movable range of the machine during automatic operation. During rotation, keep hands, feet and face away from the spindle.
- Carry out the dry run before the actual machining to confirm the machining program, tool offset amount and workpiece coordinate system offset amount.

1.4.3 Automatic Operation Pause

The automatic operation pause state means that operation or motion temporarily pauses during execution of one block during the automatic operation start. The AUTO PAUSE lamp on the machine operation panel is ON and the AUTO START lamp is OFF.

(1) From automatic operation start to automatic operation pause (transition 7.)
- When the FEED HOLD switch on the machine operation panel is pressed.
- When automatic mode input is out.

1.4.4 Automatic Operation Stop

The automatic operation stop state means that execution of one block is completed and stopped during automatic operation start. Both the AUTO START and AUTO PAUSE lamps on the machine operation panel are OFF.

(1) From automatic operation start to automatic operation stop (transition 8.)
- When the SINGLE BLOCK switch on the machine operation panel is turned ON and execution of the block is completed.
- When the automatic mode input changes to another automatic mode input.

2. Indicator Lamps

2.1 NC Unit Ready

The NC UNIT READY lamp indicates that the NC unit is ready to run. This will light approx. one second after the NC power is turned ON. The lamp is turned OFF at emergency stop or when an alarm occurs in the drive or operation block.

2.2 Automatic Operation Busy

The IN AUTO OPERATION lamp is ON from CYCLE START switch turning on in the automatic operation mode (Memory, or MDI) to the program end after M02 or M30 execution, reset, or emergency stop.

2.3 Automatic Operation Start Busy

The AUTO START lamp indicates that the NC unit is executing control in the automatic operation mode. It is ON from the automatic operation start state entered when the CYCLE START switch is pressed in the automatic operation mode (Memory, or MDI) to the automatic operation start end such as the automatic operation pause busy state entered when the FEED HOLD switch is pressed or block completion stop (block stop).

2.4 Automatic Operation Pause Busy

The AUTO PAUSE lamp is ON from FEED HOLD switch turning on to CYCLE START switch turning on or when the MODE SELECT switch is changed from the automatic to manual mode during the automatic operation.

2.5 Return to Reference Point

Output is executed when the controlled axis arrives at the reference point during manual or automatic reference point return.

2.6 NC Alarm

The NC ALARM lamp is turned ON when an alarm occurs during NC operation.

2.7 M00

If M00 given in a program is executed during automatic operation, automatic operation stop is performed after execution of the M00 block is completed. The M00 lamp is turned ON. (This depends on PLC processing.)

2.8 M02/M30

When M02 or M30 is executed during automatic operation, the NC unit reaches the program end and the M02 or M30 lamp is turned ON. (This depends on PLC processing.)

3. Reset Switch and Emergency Stop Button

3.1 Reset Switch

The NC unit is reset by turning ON the RESET switch on the machine operation panel or the RESET key on the setting display unit. When the RESET switch or key is turned ON while the controller is running, the unit is placed in one of the following states.

1) If a movement command is being executed, movement stops with deceleration and the remaining distance in the executing block is cleared.
2) If miscellaneous function such as M, S, or T is being executed, execution of the miscellaneous function is interrupted.
3) The active and buffer memory contents and display are cleared.
4) If a program error occurs and remains, the program error state is cleared and the NC ALARM lamp is turned OFF.
5) If reset is input while using the input/output device, the input/output is interrupted.

3.2 Emergency Stop Button

The EMERGENCY STOP button is a red mushroom-shape pushbutton. The not ready state is set by pressing the EMERGENCY STOP button.
During emergency stop, the READY lamp is turned OFF and automatic operation and manual operation do not work. The NC unit is reset.
If the EMERGENCY STOP button is pressed when a movement command is executed, the moving axis stops and all other machine motions also stop. When the EMERGENCY STOP button is released, the READY lamp is turned ON in about one second and operation enable state (READY state) is entered.

When parameter is emergency stop hold type, if the EMERGENCY STOP button is released, the emergency stop state is held. To release the emergency stop state, turn ON the RESET switch.
If the EMERGENCY STOP LIMIT switch of each axis works, the same state as if the EMERGENCY STOP button were pressed may be entered depending on the machine specifications.

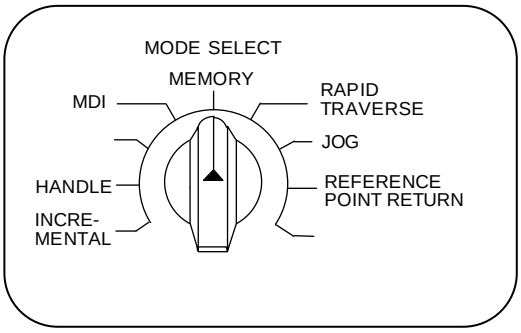
⚠ CAUTION

If the axis overruns or makes an abnormal noise, press the EMERGENCY STOP button immediately and stop the axis.

4. Operation Mode

4.1 Mode Selection Switch

The MODE SELECT switch is used to determine the controller operation mode.



Jog feed mode:
Select the mode to move the controlled axis consecutively at manual feedrate.
Rapid traverse feed mode:
Select the mode to move the controlled axis consecutively at rapid traverse feedrate.
Return to reference position mode:
Select the mode to position the controlled axis at the machine reference position manually.
Incremental mode:
Select the mode to move the controlled axis at a given distance.
Handle feed mode:
Select the mode to move the controlled axis by using the manual handle.
Memory Mode:
Select the mode for memory operation.
MDI mode:
Select the mode for MDI operation.

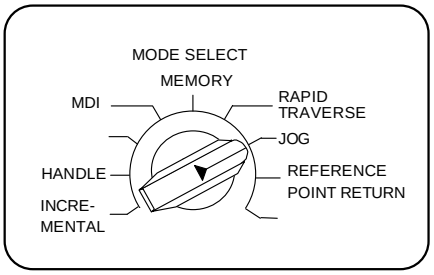
(Note 1) See 1.4 for the running state when a change is made to another mode during automatic operation.

4.2 Jog Feed Mode

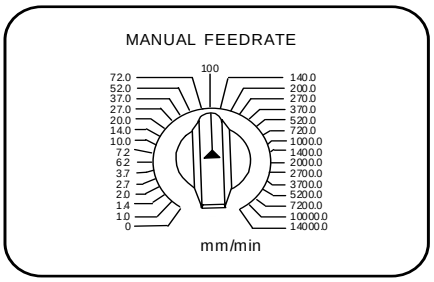
The jog feed mode enables the machine to be moved consecutively at the feedrate set by using the MANUAL FEED RATE switch manually. The jog feed mode is started by using the FEED AXIS SELECT switch.
See Section 5 for the MANUAL FEED RATE switch.

Operation procedure

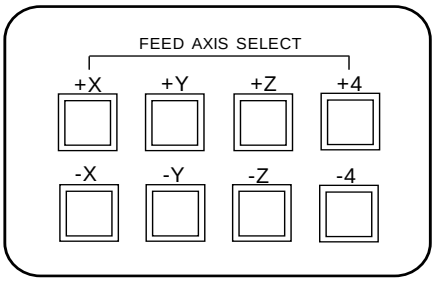
Using the MODE SELECT switch, select the jog mode.



Using the MANUAL FEED RATE switch, set the feedrate.
The feedrate unit is the travel distance (mm) per minute.



To move the controlled axis, turn on the FEED AXIS SELECT switch. The controlled axis is moved while the switch is turned on.
When the switch is turned off, the controlled axis stops with deceleration.



(Note 1) When the MANUAL OVERRIDE switch on the operation panel is turned on, the override value set by using the FEED RATE OVERRIDE switch takes precedence over the feedrate set by using the MANUAL FEED RATE switch.

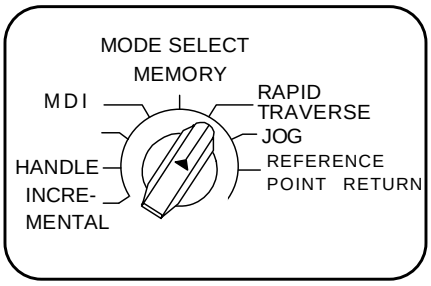
4.3 Rapid Traverse Feed Mode

The rapid traverse feed mode enables the machine to be moved consecutively at rapid traverse feedrate manually.
The rapid traverse feedrate can be changed in four steps by using the RAPID TRAVERSE OVERRIDE switch. The rapid traverse feed mode is started by using the FEED AXIS SELECT switch.

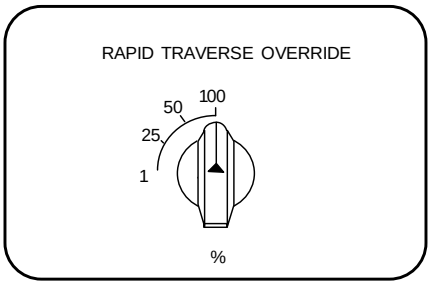
(Note 1) Refer to the manual issued by the machine manufacturer for the rapid traverse feedrate.
(Note 2) See Section 5 for the RAPID TRAVERSE OVERRIDE switch.

Operation procedure

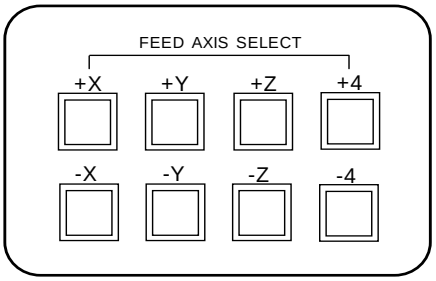
Using the MODE SELECT switch, select the rapid traverse feed mode.



Using the RAPID TRAVERSE OVERRIDE switch, set any desired override value.



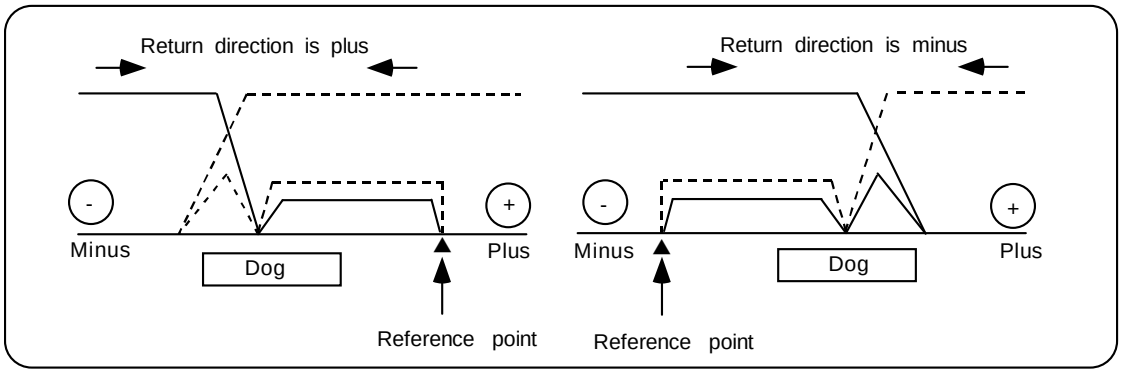
To move the controlled axis, turn on the FEED AXIS SELECT switch. The controlled axis is moved while the switch is turned on.
When the switch is turned off, the controlled axis stops with deceleration.



(Note 1) The override value set by using the CUTTING FEED OVERRIDE switch is not effective for the rapid traverse feedrate; when the override value is 0%, the controlled axis does not move.

4.4 Return to Reference Position Mode

This mode enables a given controlled axis to be returned to the defined position unique to the machine (reference position) manually.
The first return to reference position after the NC power is turned on becomes the dog mode. In the second or later return to reference position, the dog mode or high speed return can be selected by setting a given parameter.
Patterns of return to reference position are shown below.



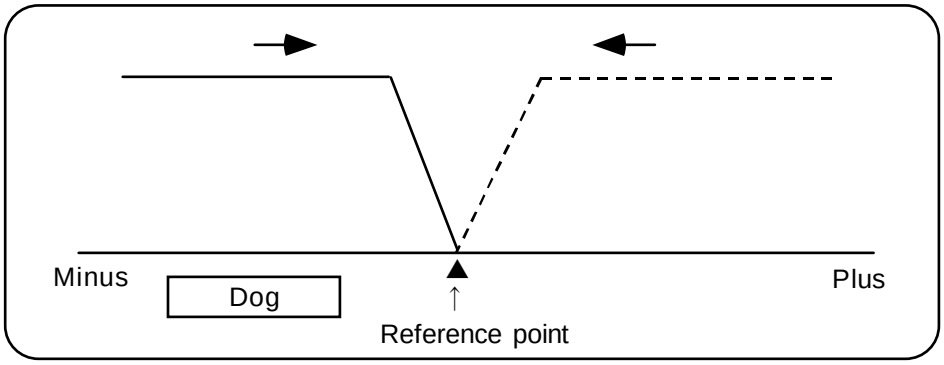
Dog mode return to reference position

The steps below describe what happens to the controlled axis when it returns to the reference point for the first time with the power on and with the machine in an "operations not ready state" (emergency stop is engaged or the servo alarm is on) or when the parameters are selected in the dog mode.

(1) The controlled axis is moved in the direction where the near point detection limit switch and dog approach each other in the return to reference position mode.
(2) When the limit switch kicks the dog, the controlled axis once stops with deceleration.
(3) Next, the controlled axis moves to the reference position at the approach rate set in the parameter.
(4) When it arrives at the reference position, the reference position arrival signal is output.

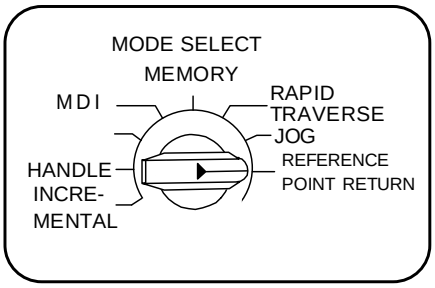
High speed return to reference position

If high speed return is set in a given parameter after dog mode return to reference position is executed, then high speed return to reference position will be made.
If the return direction is erroneous in high speed return to reference position, an alarm occurs.
A return is made to the reference position at the rapid traverse feedrate.

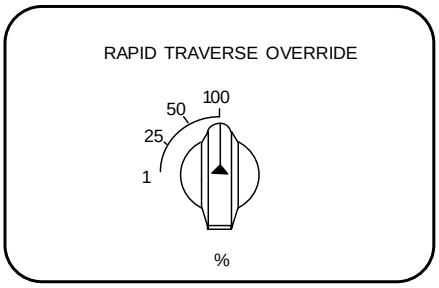


Operation procedure

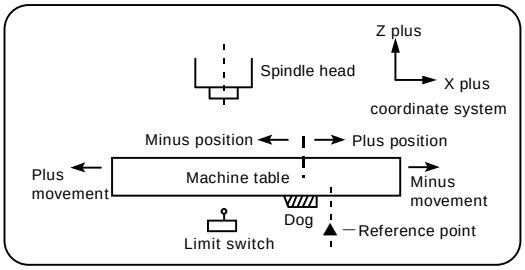
Using the MODE SELECT switch, select the return to reference position.



Using the RAPID TRAVERSE OVERRIDE switch, set any desired override value.

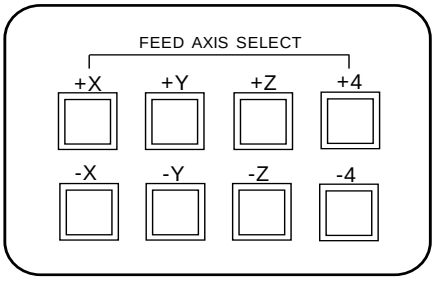


Check the current machine position.



The machine position depends on whether the near point detection limit switch is the plus or minus side with the near point dog on the machine table as illustrated above as the reference.
When the limit switch exists on the dog, move to either plus or minus.

Using the FEED AXIS SELECT switch, move the machine.
If the limit switch exists in the minus direction as illustrated above, turn on a plus FEED AXIS SELECT switch.



For dog mode return to reference position, turn on the FEED AXIS SELECT switch (+ or –) in the direction where the dog and limit switch approach each other.

For high-speed reference point return, turn on the FEED AXIS SELECT switch (+ or –) in the direction that the spindle head approaches the reference point.

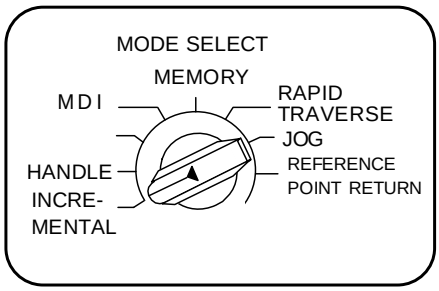
Hold the FEED AXIS SELECT switch on during return to reference position until the machine passes by the dog (dog mode) or the REFERENCE POSITION ARRIVAL lamp goes on (high speed return).

4.5 Incremental Feed Mode

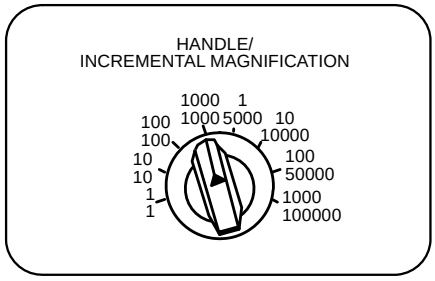
The incremental feed mode enables the controlled axis to be moved at a given distance selected by using the HANDLE/INCREMENTAL MAGNIFICATION switch at the manual feedrate when the FEED AXIS SELECT switch is on.

Operation procedure

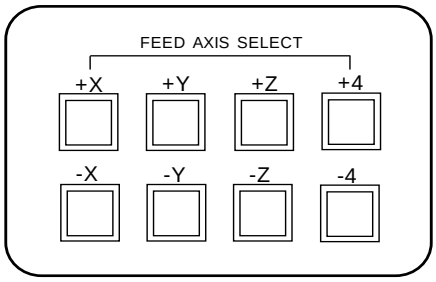
Using the MODE SELECT switch, select the incremental feed mode.



Using the HANDLE/INCREMENTAL MAGNIFICATION switch set a travel distance.



The controlled axis selected by turning on the FEED AXIS SELECT switch once is moved at a given distance.

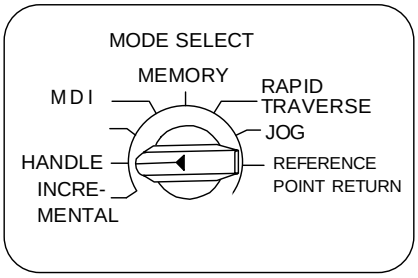


4.6 Handle Feed Mode

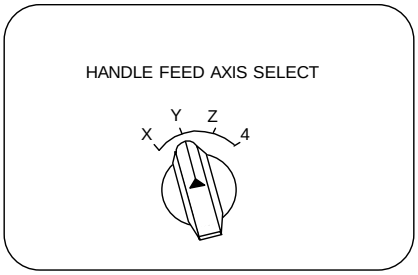
The controlled axis can be moved by turning the manual handle.
The travel distance per graduation of the handle depends on how the HANDLE/INCREMENTAL MAGNIFICATION switch is set.
The axis that can be moved by using the manual handle is determined by setting the HANDLE FEED AXIS SELECT switch.

Operation procedure

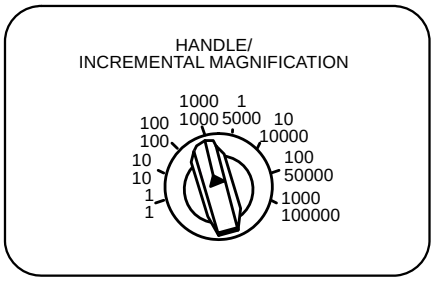
Using the MODE SELECT switch, select the handle feed mode.



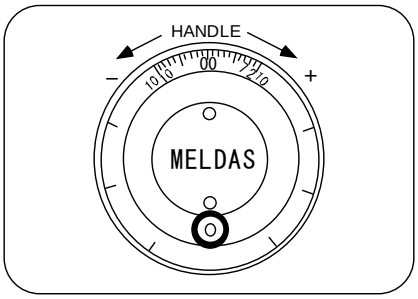
Using the HANDLE FEED AXIS SELECT switch, select the controlled axis to be moved.



Using the HANDLE/INCREMENTAL MAGNIFICATION switch, set the travel distance per graduation of the handle.



Move the axis by turning the HANDLE in any desired direction.



(Note 1) If the handle is turned quickly when a high handle magnify- cation is set, the feed will be clamped at the corresponding axis' rapid traverse rate for safety purposes. The amount exceeding the clamp will be discharged resulting in a fraction irrelevant to the handle magnification.

(Note 2) As for the time constant in handle feed, the G01 time constant or step (time constant 0) can be selected using the base specifications parameter (#1194 H_acdc).

4.7 Memory Mode

In the memory mode, a work program registered in memory is called and automatic operation is executed.

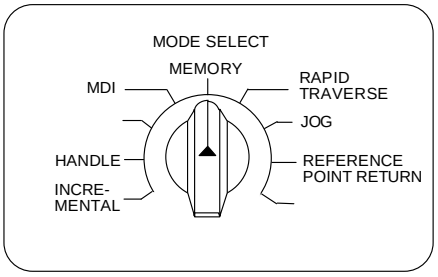
Operation procedure

Call the work program for memory operation by using the setting and display unit.

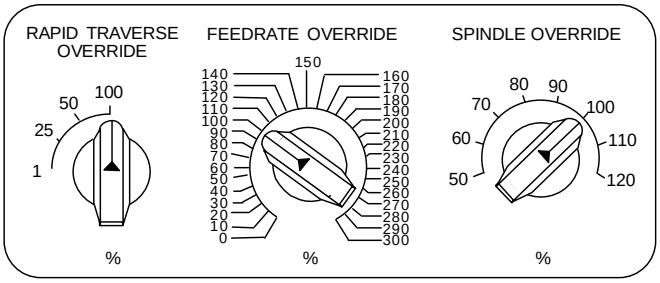
(Refer to the “Operation Search” section in the Operation Manual.)

Check whether or not the work program is called normally.

Using the MODE SELECT switch, select the memory mode.

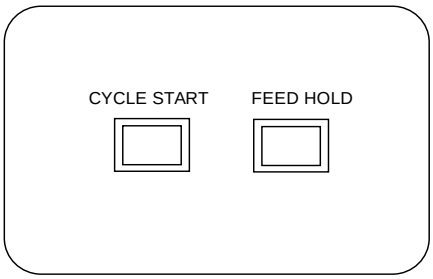


Set any desired override value by using the switch RAPID TRAVERSE OVERRIDE, FEED RATE OVERRIDE, SPINDLE OVERRIDE. Normally, set the value to 100%.



Automatic operation is started by turning on the CYCLE START switch.

The CYCLE START switch becomes effective when it is once turned on, then off.



To temporarily stop machine motion, turn on the FEED HOLD switch. The controlled axes being moved stop with deceleration.

When machine motion is stopped by using the FEED HOLD switch, automatic operation will be restarted by turning on the CYCLE START switch.
Memory operation terminates when M02 or M30 in the program is executed. The M02 or M30 lamp on the machine operator panel is turned on.
To repeat execution of a single program, input the rewind signal by reset & rewind at user PLC.
To forcibly terminate automatic operation, turn on the RESET switch.

CAUTION

Carry out dry operation before actually machining, and confirm the machining program, tool offset and workpiece coordinate system offset.

4.8 MDI Operation Mode

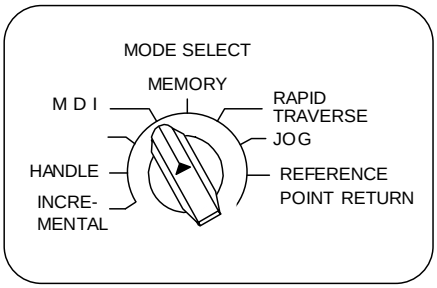
In the MDI operation mode, automatic operation is executed by using a program set on the CRT setting and display unit MDI screen.

Operation procedure

The MDI operation follows the memory operation.

Set data on the CRT setting and display unit MDI screen.

Using the MODE SELECT switch, select the MDI mode.



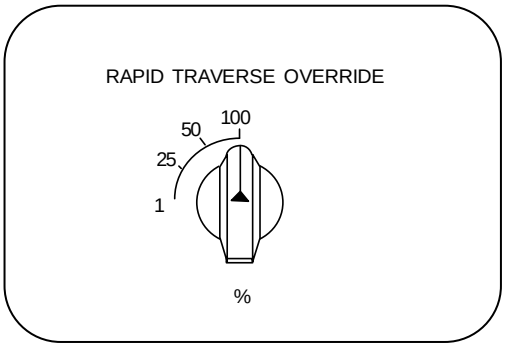
The following steps are the same as the memory operation steps.

5. Operation Panel Switches in Operation Mode

5.1 Rapid Traverse Override

Use the RAPID TRAVERSE OVERRIDE switch to override the rapid traverse feedrate in automatic or manual operation.
RAPID TRAVERSE OVERRIDE is applicable to the following:

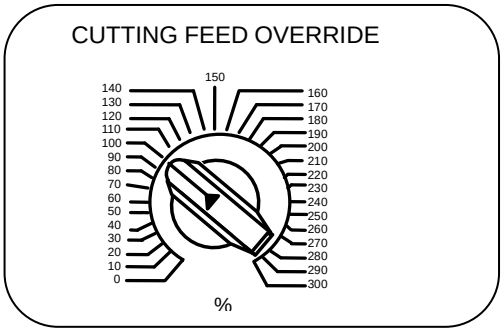
Automatic operation: G00, G27, G28, G29, G30
Manual operation: Rapid traverse, return to reference position, incremental feed



(Note) Set the "CUTTING FEED OVERRIDE" switch to 0 to set the raid traverse override value to 0%.

5.2 Cutting Feed Override

Use the CUTTING FEED OVERRIDE switch to override the feedrate in automatic operation (G01, G02, or G03F command) or the manual feedrate of jog feed in manual operation in 10% units in the range of 0% to 300%. CUTTING FEED OVERRIDE is also applicable to the dry run rate in automatic operation.

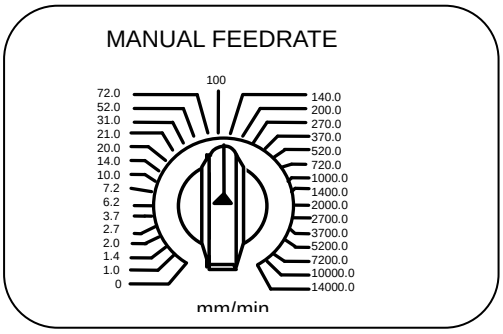


(Note 1) The dry run rate is the movement rate set by using the MANUAL FEEDRATE switch by overriding the programmed feedrate in automatic operation.
(Note 2) See Section 6.6 for feedrate override applied to manual feedrate.

5.3 Manual Feedrate

Use the MANUAL FEEDRATE switch to set the feedrate in jog feed mode during manual operation. The feedrate can be selected among 31 steps from 0 to 14000.0 mm/min.
When the MANUAL OVERRIDE switch (interrupt switch) is turned on, the override value set by using the CUTTING FEED OVERRIDE switch takes precedence over the value set by using the MANUAL FEEDRATE switch.

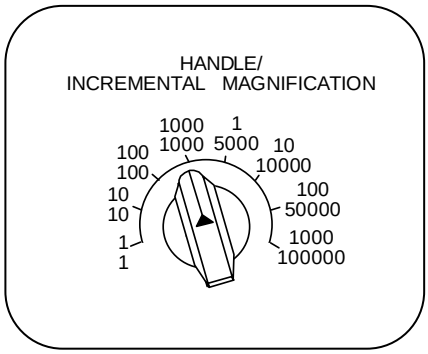
Manual feedrate (mm/min)				
0.	7.2	72	720	7200
1.0	10.0	100	1000	10000
1.4	14.0	140	1400	14000
2.0	20.0	200	2000	
2.7	27.0	270	2700	
3.7	37.0	370	3700	
5.2	52.0	520	5200	



5.4 Handle/Incremental Feed Magnification Factor

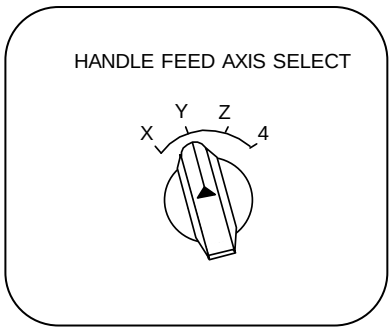
Use the HANDLE/INCREMENTAL MAGNIFICATION switch to set the travel distance specified when manual handle feed or incremental feed is made.
The travel distances for each axis are listed below:
(Up to 1000 can be set for handle feed.)

Handle	Incremental
1	1
10	10
100	100
1000	1000
1	5000
10	10000
100	50000
1000	100000



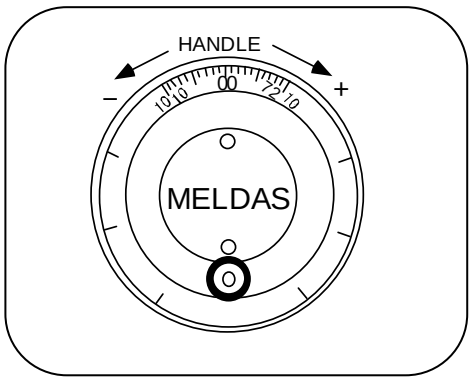
5.5 Handle Feed Axis Selection

Use the HANDLE FEED AXIS SELECT switch to select the axis moved by handle operation when the handle mode is selected.



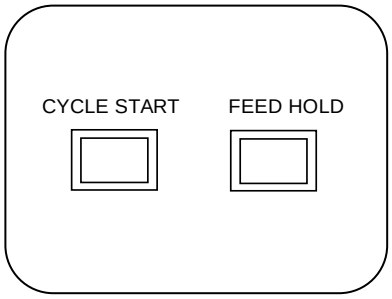
5.6 Manual Pulse Generator

In the manual handle mode, fine feed of the machine can be made by turning the manual pulse generator.
The manual pulse generator has 100 graduations per revolution and outputs one pulse per graduation. The travel distance per pulse is set by using the HANDLE/INCREMENTAL MAGNIFICATION switch.



5.7 Cycle Start and Feed Hold

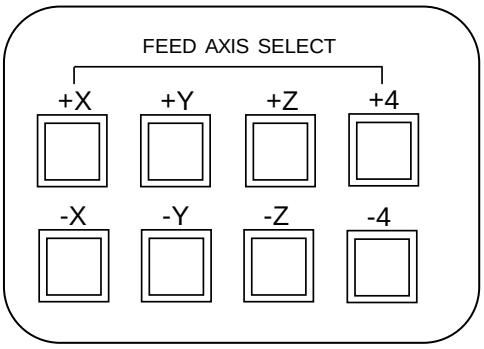
Use the CYCLE START switch to start automatic operation (memory, tape, or MDI). Automatic operation is executed by turning on the switch. Use also the switch for restart from stop by the FEED HOLD switch or the automatic operation stop state.



The CYCLE START switch becomes effective when the switch is turned on, then off.
Use the FEED HOLD switch to temporarily stop automatic operation (for example, deceleration stop of the control axis during automatic operation). To restart operation, use the CYCLE START switch.

5.8 Feed Axis Selection

Use the FEED AXIS SELECT switch to start the controlled axis during manual operation. While the FEED AXIS SELECT switch is held on, the selected controlled axis is moved. When the switch is turned off, the controlled axis move stops.



6. Operation Panel Switch Functions

6.1 All Axes Machine Lock

(1) When the "ALL AXES MACHINE LOCK" switch is turned ON, the NC commands for the manual operation or automatic operation movement can be executed without moving the machine.
The current position display on the setting and display unit will be counted.
(2) Feedrate during the machine lock is the command speed.
(3) If the "ALL AXES MACHINE LOCK" switch is changed during automatic operation, the automatic operation will stop after the block currently being executed is completed, and then the setting will be validated.
(4) During reference point return (G28, G30), the movement will be controlled with the machine lock status up to the middle point. The machine lock status will be ignored from the middle point to the reference point.
(5) If the "MACHINE LOCK" switch is changed during manual operation, the setting will be validated after the feed is stopped once.
(6) The M, S, T and B commands are executed according to the program.
(7) After the axis is moved in the "MACHINE LOCK" ON state, the current position display when the "MACHINE LOCK" is turned OFF and the machine position will not match.
If AUTO START is pressed in this state, the difference between the current position and the machine position will be added to the movement amount.
If RESET is pressed, the current position display will change to match the machine position. Thus, after turning "MACHINE LOCK" OFF, press RESET before starting operation.

(Note) Each coordinate value updated with the machine lock will be preset to the value created by reset (including M02 and M30) based on the machine value.

6.2 Chamfering (L system)

Chamfering can be validated/invalidated in the thread cutting cycle using an external switch.

6.3 Miscellaneous Function Lock

(1) M, S, T, or B function execution can be ignored by turning on the MISCELLANEOUS FUNCTION LOCK switch.
(2) M, S, T, B function BCD output is made, but the start signal is not output.
(3) If the MISCELLANEOUS FUNCTION LOCK switch is changed during command execution, automatic operation stops after the block being executed is terminated. Then, it becomes effective.

6.4 Single Block

(1) When the SINGLE BLOCK switch is turned on, automatic operation stops after the block being executed is terminated. That is, automatic operation stops after one program block is executed.
(2) The single block stop point in the fixed cycle mode is fixed according to the fixed cycle.

6.5 Dry Run

(1) When the DRY RUN switch is turned on, the feedrate set by using the MANUAL FEED RATE switch takes precedence over the programmed feedrate (F).

6.6 Manual Override

(1) When the MANUAL OVERRIDE switch is turned on, the override value set by using the FEED OVERRIDE switch takes precedence over the value set by using the MANUAL FEED RATE switch.

(2) The override value also takes precedence over the dry run during automatic operation.

(3) Manual override becomes effective immediately when the switch is turned on.

6.7 Override Cancel

(1) When the OVERRIDE CANCEL switch is turned on, the programmed F command value takes precedence over the override value set by using the FEED RATE OVERRIDE switch.

(2) It is not effective for manual override.

6.8 Optional Stop

(1) If M01 is programmed, the machine automatically stops by turning on the OPTIONAL STOP switch. When the switch is off, M01 is ignored and the machine does not stop.
(Note) Some processes within the user-PLC are required.

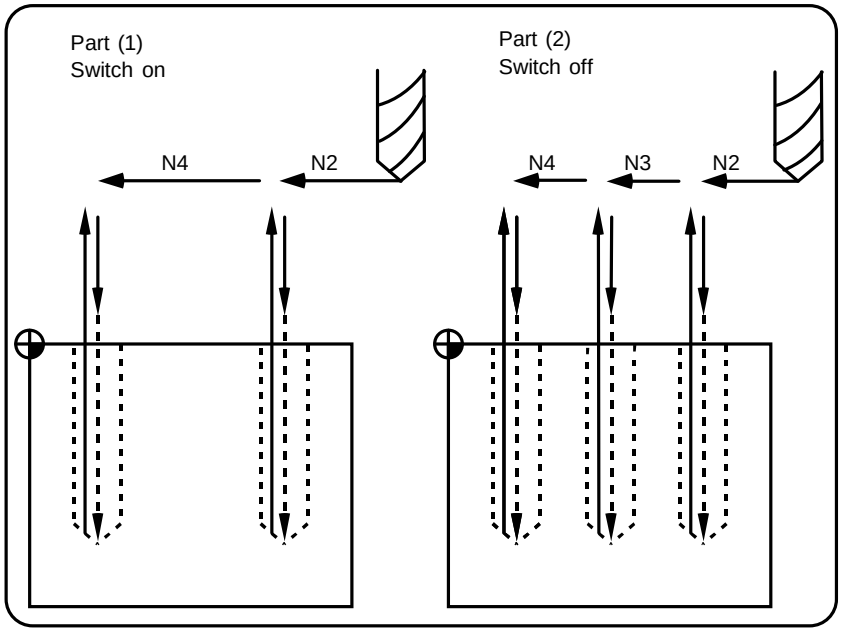
(2) The machine stops after the M01 block is executed.

6.9 Optional Block Skip

When the OPTIONAL BLOCK SKIP switch is turned on, a block which begins with a slash (/) is skipped; when the switch is off, the block is executed. This enables the operator to specify whether or not a block beginning with a slash (/) code is executed.

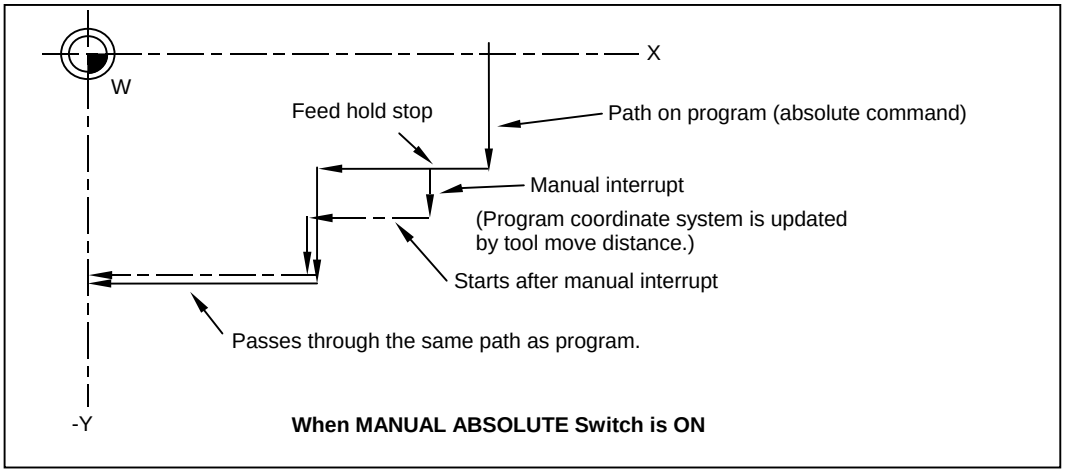
(Example) To work two parts as illustrated below, if the following program is prepared and work is made by turning on the OPTIONAL BLOCK SKIP switch, part (1) is provided; if work is made by turning off the switch, part (2) is provided:

```
Program  N1G54 ;
         N2G90G81X50. Z-20. R3. F100 ;
         /N3X30. ;
         N4X10. ;
         N5G80 ;
         M02 ;
```

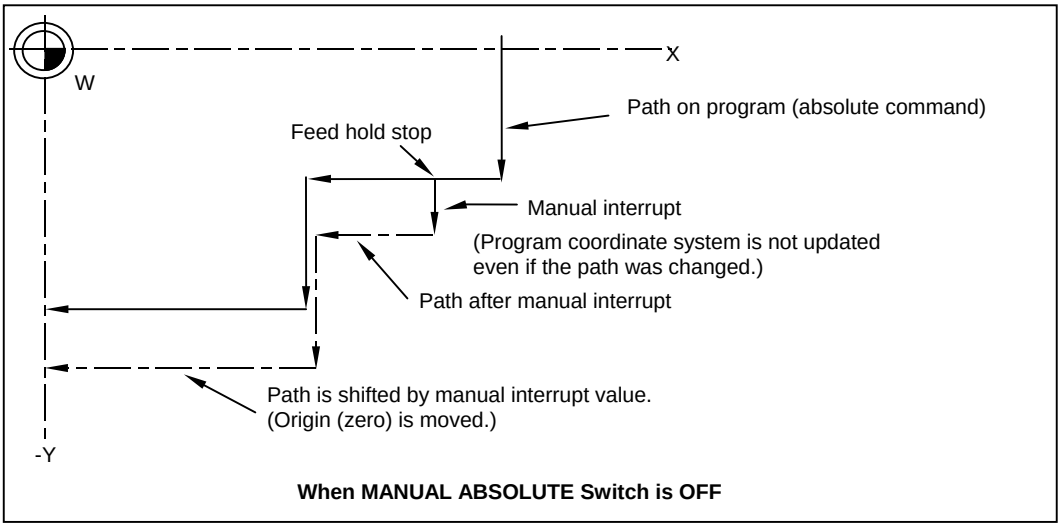


6.10 Manual Absolute

When the MANUAL ABSOLUTE switch is turned on, the program coordinate system is updated by manual tool move distance. If the switch is off, the program coordinate system is updated by manual tool move distance.



When MANUAL ABSOLUTE Switch is ON

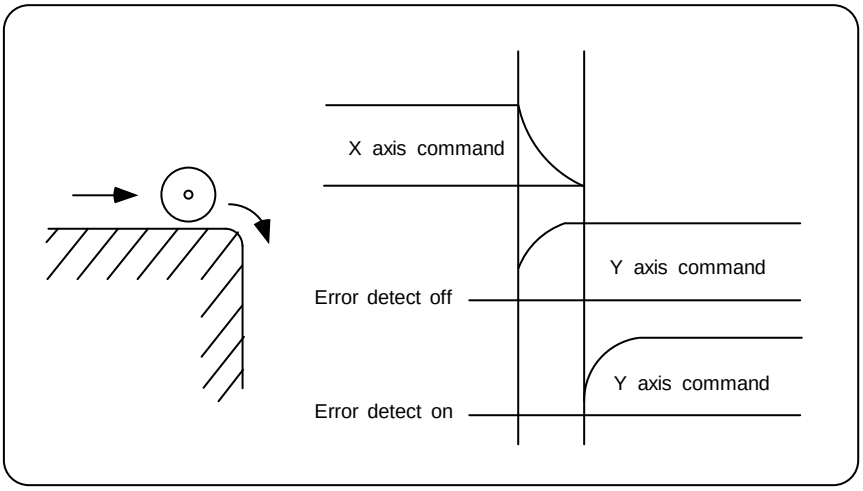


When MANUAL ABSOLUTE Switch is OFF

(Note) Normally, the switch will be "ON" if there is no manual absolute switch. Depending on the machine, the switch may be "OFF" so check the specifications issued by the machine manufacturer.

6.11 Error Detect

For positioning (G00), machine deceleration check is made before next block move is started. For cutting (G01, G02, or G03), the next block is started before the machine reaches the move command end point. Thus, the corner part is slightly rounded.
To prevent rounded corners, turn on the error detect signal. This will cause the machine to decelerate until the remaining distance falls below the value of the parameter. The next block command is stopped during this time.
This function is equivalent to G09 in the program.
The parameter that is used by the error detect switch and the G09 command for determining the remaining distance after deceleration for moving to the next command can be set with the settings monitor device.



6.12 Follow-up Function

The follow-up function monitors machine motion in the emergency stop state and reflects it in the current position and workpiece coordinates. Thus, the work program can be continued without again making return to reference position after emergency stop.

6.13 Axis Removal

When the machine receives the axis removal signal, that axis no longer becomes the controlled axis. Accordingly, the alarm for the stroke end axis and the servo alarms (excessive errors, lack of signal, drive alarm, etc.) will be ignored. At the same time, the axis will become interlocked.

(Note) This cannot be used for the absolute position detector specification axis.

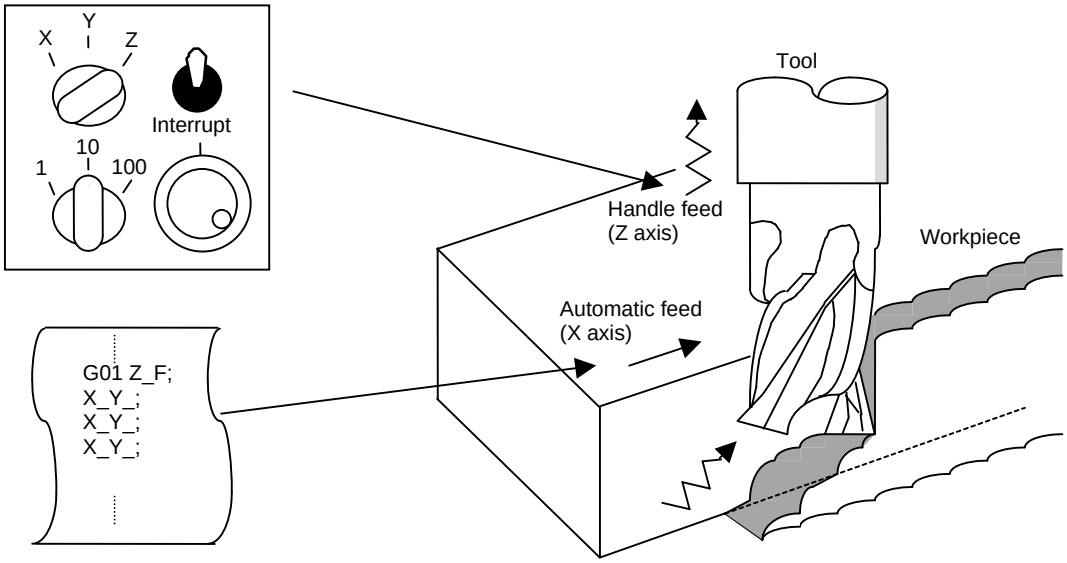
6.14 Manual/Automatic Synchronous Feed

While you are using the automatic operation in the automatic operation mode, you can simultaneously operate the machine manually (jog, return to reference point, incremental feed, handle).
To select the manual mode and automatic mode, refer to the machine's instruction manual.

6.15 Handle Interruption

6.15.1 Outline

Section 6.14 explains automatic handle interruption, which enables the operator to interrupt movement using the manual handle in automatic modes (memory, MDI).
(This is an optional function.)



6.15.2 Interruptible Conditions

(1) The automatic handle interrupt function allows you to interrupt the program manually by selecting the manual handle mode in automatic mode selection (tape, memory, MDI, etc.). However the interrupt cannot be generated from the manual handle when an automatic reference point return command (G28, G29, G30), the thread cutting (G33), or the skip command (G31) has been executed or when tapping in the tapping cycle.

(2) If automatic operation mode such as tape, memory, or MDI is being selected even when an automatic operation pause (including a block stop) is established, automatic handle interruption is enabled.

(3) If the axis is moved during dwell (G04) command processing by using automatic handle interruption, the dwell count operation will stop. A check is made for that the axis movement has been completed, then the dwell count operation continues.

(4) Automatic handle interruption is enabled even if automatic machine lock has been set. If manual machine lock has been set, the machine does not move; it only updates the POSITION display. If manual machine lock has not been set, the machine moves by the interruption distance by the manual handle and the POSITION display is updated.

(5) This function is disabled for an axis to which the interlock signal has been input or an axis, the interruption direction of which is the soft limit.

6.15.3 Interruption Effective Axis

(1) Automatic handle interruption is enabled only for axes to which manual handle axis selection has been input.

(2) Automatic handle interruption is enabled for a maximum of three axes. (The number of axes is restricted by the number of handles.)

6.15.4 Axis Movement Speed Resulting from Interruption

(1) The movement speed of the axis for which handle interruption is executed, may exceed the rapid traverse feed rate during rapid traverse feed command (G00) processing in automatic start. To prevent this, clamp the axis. (The movement speed equals Automatic-start movement speed + Speed resulting from manual handle interruption.)

(2) The movement speed of the axis for which handle interruption is executed, may exceed the cutting feed speed during cutting feed command (G01, G02, G03) processing in automatic start. To prevent this, clamp the axis. (The movement speed equals Automatic start movement speed + Speed resulting from manual handle interruption.)

(3) If, during automatic start, manual handle interruption is executed, in the same direction, for the axis that is moving at an external decelerating speed, the axis movement speed may exceed the external decelerating speed. To prevent this, clamp the axis. (The movement speed equals Automatic start movement speed + Speed resulting from manual handle interruption.)

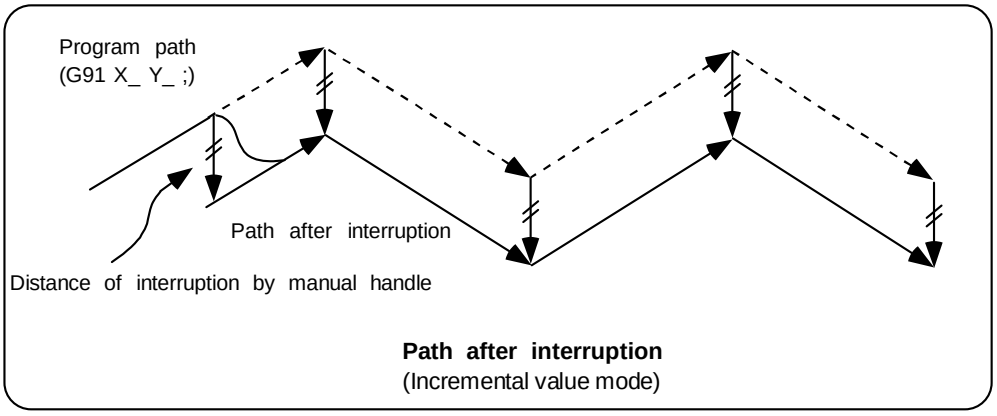
(4) If an attempt is made to execute interruption at a speed exceeding the clamp speed, the reading on the handle scale does not match the distance of interruption.

(5) The handle scale factor depends on the selected input of the manual handle/step scale factor.

6.15.5 Path Resulting After Handle Interruption

(1) For incremental value (G91) mode

The locus deviates from the program path by the distance of interruption. (See the figure below.)



Path after interruption
(Incremental value mode)

(2) For absolute value (G90) mode

If program absolute value update by the distance of handle interruption is disabled, the locus deviates from the program path by the distance of interruption.

If this update is enabled, the locus returns to the program path during processing of the following command:

For single block running — Return command is issued in the block next to the one for which the interruption has been completed.

For continuous running — Return command is issued in the third block, if the block for which the interruption has been completed is the first block.

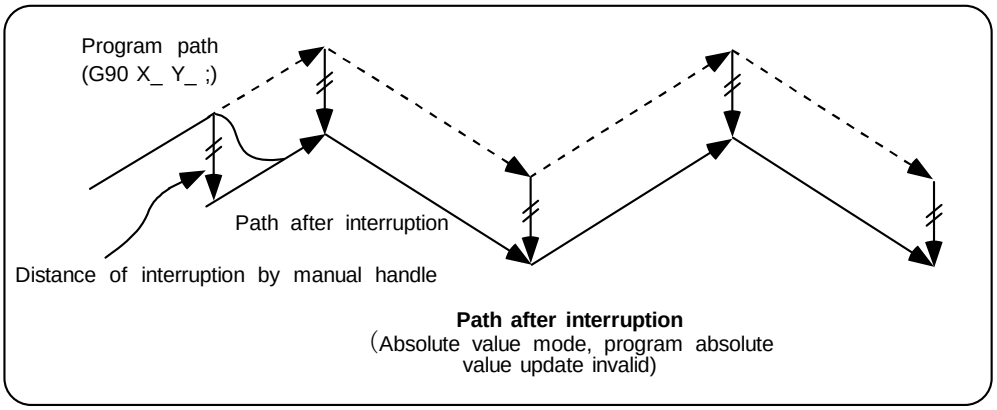
(Note) Each of the POSITION and MACHINE display includes the distance of handle interruption.

For automatic handle interruption, select whether absolute data is updated or not, as follows:

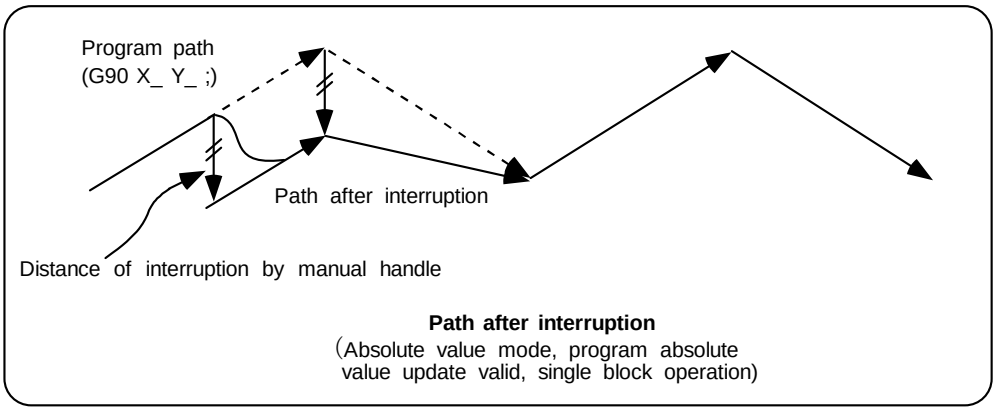
1) Using machine parameters, set whether the manual absolute changeover switch or parameters are used.
2) If the parameters are used, set whether or not absolute data is updated for each axis, by using machine parameters other than those in item.
3) If the manual absolute changeover switch is used, use the machine operation panel switch for selection.

Absolute value update conditions for automatic handle interruption

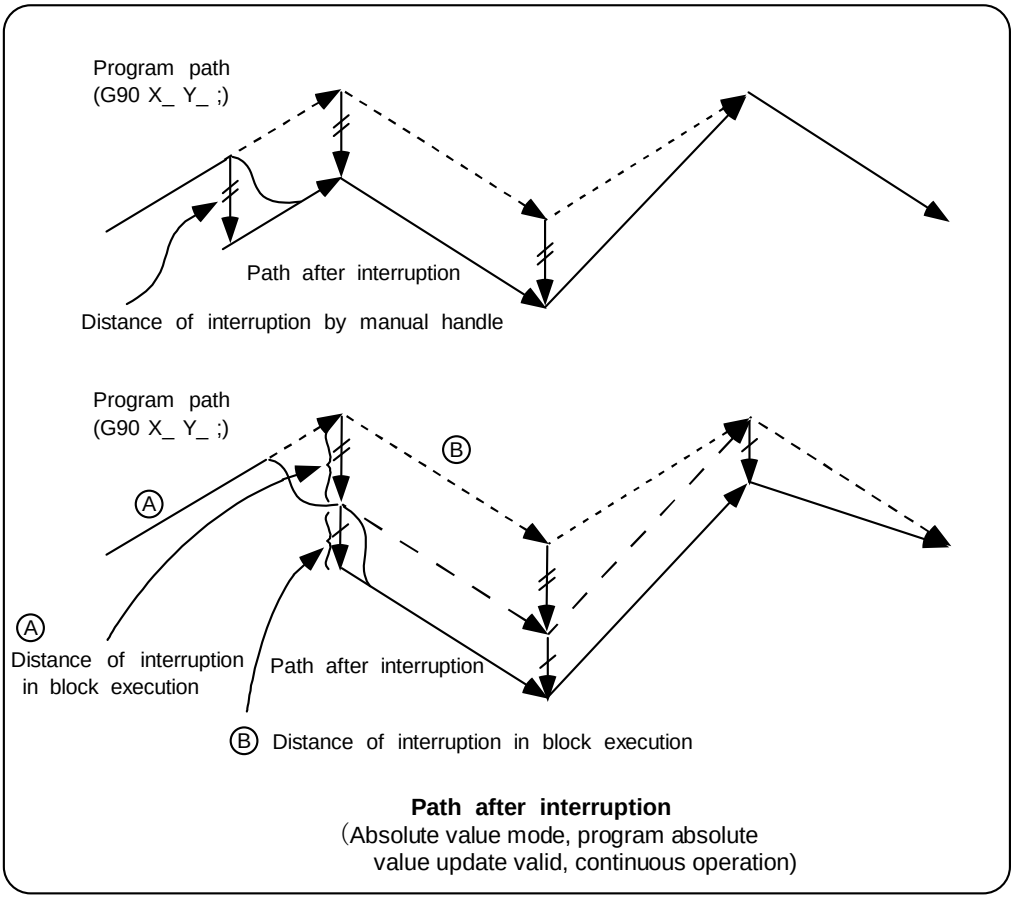
			Absolute data update		MONITOR 2 screen manual interruption distance display
Parameter "1145 I_abs"	ON "1"	Parameter "#1061 intabs" (Every axis)	ON "1"	Absolute value is updated.	Not updated.
			OFF "0"	Absolute value is not updated.	Updated.
	OFF "0"	PLC interface manual absolute switching	ON	Absolute value is updated.	Not updated.
			OFF	Absolute value is not updated.	Updated.



Path after interruption
(Absolute value mode, program absolute value update invalid)



Path after interruption
(Absolute value mode, program absolute value update valid, single block operation)



6.15.6 Handle Interruption in Tool Radius Compensation

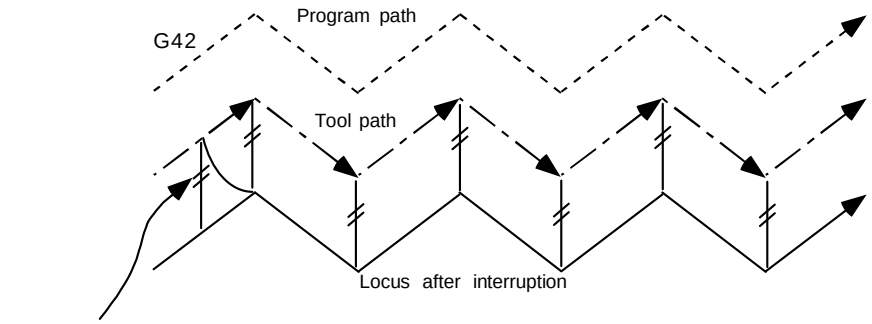
Special movement described below relates only to the tool radius compensation plane axis. It has no influence on the other axes.
At time of tool radius compensation (G41, G42):
In incremental value mode — The quantity of deviation equals the distance of interruption.
In absolute value mode — If handle interruption is executed in the block for which tool radius compensation (G41, G42) is being executed, the proper tool path will return in the succeeding block. This rule applies only when the program absolute update is active during single block running. If program absolute update is active during continuous running, the proper tool path will return with the following:

- Executing the command in the fourth block after the one for which the interruption has been completed.

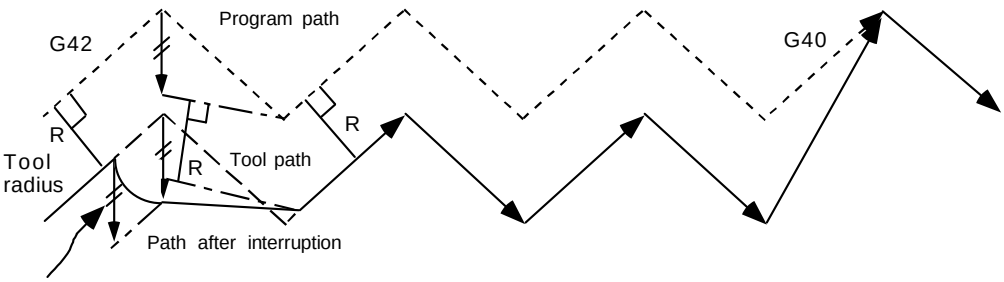
At this time, four or more blocks may not exist between

- Block for which the interruption has been completed and
- Block that contains the tool radius compensation cancel command (G40).

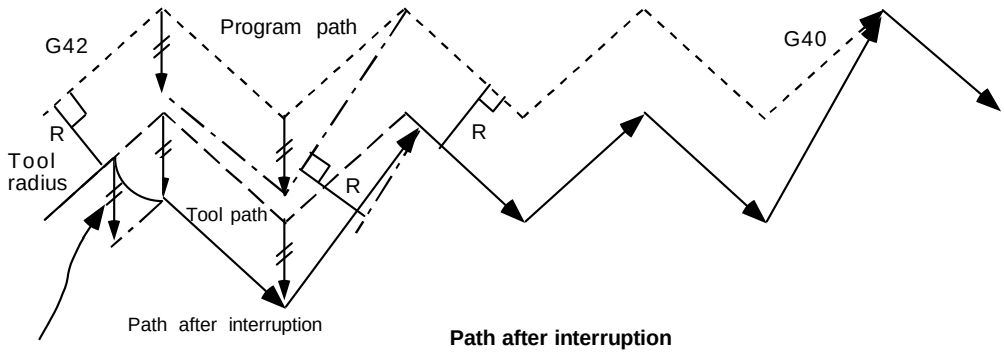
In this case, the proper tool path will return at the block next to the tool radius compensation cancel command (G40).



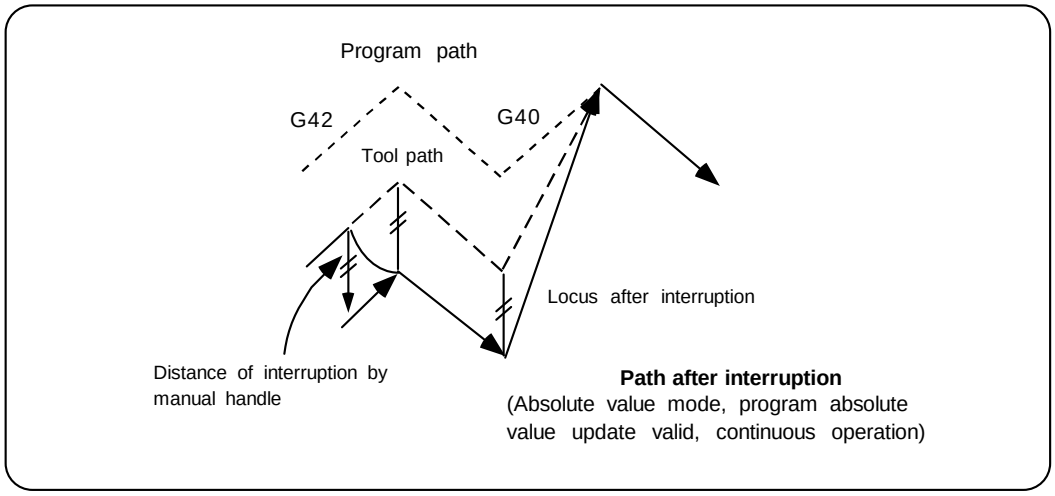
Path after interruption
(Incremental value mode, continuous operation)



Path after interruption
(Absolute value mode, program absolute value update valid, single block operation)



Path after interruption
(Absolute value mode, program absolute value update valid, continuous operation)



6.15.7 Interrupt Amount Reset

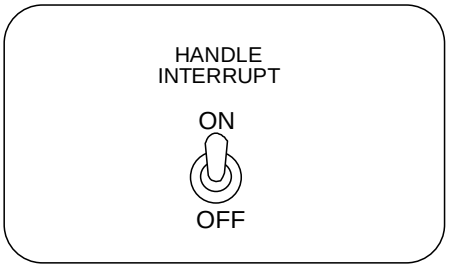
Interrupt amount is reset when
(1) dog reference position return is executed;
(2) emergency stop is released;
(3) reset rewind or reset 2 is executed; or
(4) reset 1 is executed when the interrupt amount reset parameter is ON

6.15.8 Operation Sequence

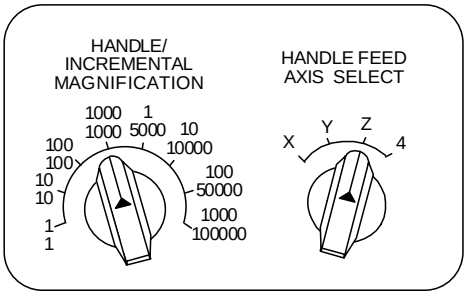
An operation example is given where automatic operation of XYZ axes is executed in the memory operation mode and the Z axis is used as a handle interrupt axis.

Perform automatic operation.

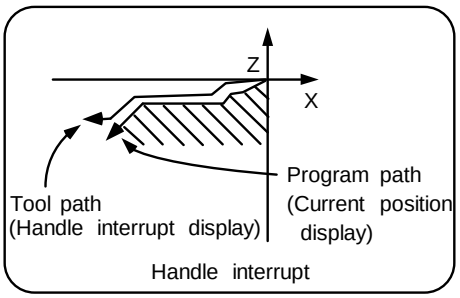
Turn on the HANDLE INTERRUPT switch on the machine operation panel.



Select interrupt axis by setting the HANDLE AXIS SELECTION switch on the machine operation panel.
Select the travel distance per handle graduation by setting the HANDLE/ NCREMENTAL RATIO switch.



Turn the manual handle in the + or – direction and change the Z axis cut depth amount.



When M2 or M30 is executed, auto operation will stop.

III. MAINTENANCE

⚠DANGER

- Do not touch the terminals while power is on. Doing so could cause electric shock.
- Correctly connect the battery. Also, do not charge, disassemble, heat, place in fire, short circuit, or solder the battery. Mishandling of a battery may cause overheating, cracks or ignition which could result in injury and fire.
- Switch off all phases of the externally supplied power used in the system when cleaning the module or retightening the terminal or module mounting screws.
  Not doing so could result in electric shock.
  Under tightening of terminal screws can cause a short circuit or malfunction.
  Over tightening of screws can cause damages to the screws and/or the module, resulting in fallout, short circuits, or malfunction.
- The capacitor is mounted to the modules. Do not incinerate the modules so that the incineration of capacitor may cause burst.
  For disposal of the modules, request for specialized industrial waste disposal services who has incineration facility.

⚠CAUTION

- Read the manual carefully and pay careful attention to safety for the on-line operation (especially program change, forced stop or operation change) performed by connecting peripheral devices to the controller during operation.
  Erroneous operation may cause machine breakage or accident.
- Never try to disassemble or modify module. It may cause product failure, operation failure, injury or fire.
- Use any radio communication device such as a cellular phone or a PHS phone more than 25cm (9.84 inch) away in all directions of C70.
  Failure to do so may cause a malfunction.
- Completely turn off the externally supplied power used in the system before installation or removing the module.
  Not doing so could result in electric shock, damage to the module or operation failure.
- Do not install/remove the module on to/from base unit or terminal block more than 50 times, after the first use of the product. Failure to do so may cause the module to malfunction due to poor contact of connector.
- Do not drop or impact the battery installed to the module. Doing so may damage the battery, causing battery liquid to leak in the battery.
  Do not use the dropped or impacted battery, but dispose of it.
- Before touching the module, always touch grounded metal, etc. to discharge static electricity from human body. Failure to do so may cause the module to fail or malfunction.
- Do not directly touch the module's conductive parts and electronic components. Touching them could cause an operation failure or give damage to the module.

In order that you can use C70 in normal and optimal condition at all times, this section describes those items that must be maintained or inspected daily or at regular intervals.

1. Maintenance Works

1.1 Instruction of Inspection Works

In order that can use C70 in safety and normal, those items that must be inspected list below.

⚠DANGER

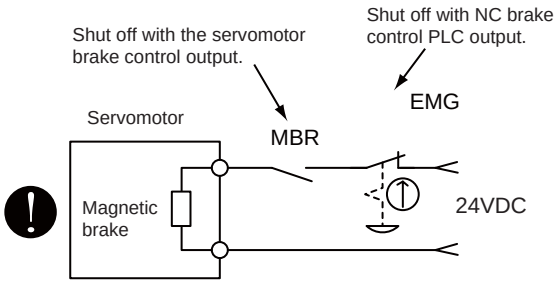
- Never open the front case or terminal covers while the power is ON or the unit is running, as this may lead to electric shocks.
- Never run the unit with the front case or terminal cover removed. The high voltage terminal and charged sections will be exposed and may lead to electric shocks.
- Never remove the front case or terminal cover at times other than wiring work or periodic inspections even if the power is OFF. The insides of the controller and servo drive unit are charged and may lead to electric shocks.
- When performing wiring work or inspections, turn the power OFF, wait at least ten minutes, and then check the voltage with a tester, etc. Failing to do so may lead to electric shocks.
- Never operate the switches with wet hands, as this may lead to electric shocks.
- Do not damage, apply excessive stress, place heavy things on or sandwich the cables, as this may lead to electric shocks or fire.
- Do not touch the controller, servo drive unit or servomotor terminal blocks while the power is ON, as this may lead to electric shocks or fire.
- Do not touch the built-in power supply, built-in grounding or signal wires of the controller and servo drive unit, as this may lead to electric shocks.

⚠CAUTION

- Be sure to ground the controller, servo amplifier and servomotor. Do not ground commonly with other devices. (Ground resistance: 100 Ω or less)
- The wiring work and inspections must be done by a qualified technician.
- Wire the units after mounting the controller, servo amplifier and servomotor. Failing to do so may lead to electric shocks or damage.
- Perform the daily and periodic inspections according to the instruction manual.
- Perform maintenance and inspection after backing up the program and parameters for the controller and servo amplifier
- Do not place fingers or hands in the clearance when opening or closing any opening.
- Periodically replace consumable parts such as batteries according to the instruction manual.
- Do not touch the lead sections such as ICs or the connector contacts.
- Do not place the controller or servo amplifier on metal that may cause a power leakage or wood, plastic or vinyl that may cause static electricity buildup.
- Do not perform a megger test (insulation resistance measurement) during inspection.
- When replacing the controller or servo amplifier, always set the new unit settings correctly.
- When replacing the controller or servo amplifier, always set the new unit settings correctly.
- After maintenance and inspections are completed, confirm that the position detection of the absolute position detector function is correct.
- Do not short circuit, charge, overheat, incinerate or disassemble the batteries.
- The electrolytic capacitor will generate gas during a fault, so do not place your face near the controller or servo amplifier.

⚠ CAUTION

- The electrolytic capacitor and fan will deteriorate. Periodically change these to prevent secondary damage from faults. Replacements can be made by our sales representative, or at the nearest service center.
- If an error occurs in the self diagnosis of the controller or servo amplifier, confirm the check details according to the instruction manual, and restore the operation.
- If a dangerous state is predicted in case of a power failure or product failure, in order to prevent that state, use a servomotor with electromagnetic brakes for maintenance or mount a brake mechanism externally.
- Use a double circuit construction so that the electromagnetic brake operation circuit can be operated by emergency stop signals set externally.



- If an error occurs, remove the cause, secure the safety and then resume operation after alarm release.
- The unit may suddenly restart after a power failure is restored, so do not go near the machine. (Design the machine so that personal safety can be ensured even if the machine restarts suddenly.)
- Confirm and adjust the program and each parameter before operation. Unpredictable movements may occur depending on the machine.
- Extreme adjustments and changes may lead to unstable operation, so never make them.
- Do not apply a voltage other than that specified in the instruction manual on any terminal. Doing so may lead to destruction or damage.
- Do not mistake the terminal connections, as this may lead to destruction or damage.
- Do not mistake the polarity (+ / -), as this may lead to destruction or damage.
- Do not touch the heat radiating fins of controller or servo drive unit, regenerative resistor and servomotor, etc., while the power is ON and for a short time after the power is turned OFF. In this timing, these parts become very hot and may lead to burns.
- Always turn the power OFF before touching the servomotor shaft or coupled machines, as these parts may lead to injuries.
- Do not go near the machine during test operations or during operations such as teaching. Doing so may lead to injuries.
- Do not bunch the control wires or communication cables with the main circuit or power wires, or lay them closely. They should be installed 100mm (3.94inch) or more from each other. Trying to bunch or install could result in noise that would cause operation failure.

2. Daily Inspection

The items that must be inspected daily are shown below.

(Note) Refer to "3. Display on 7-segment LED" for CNC CPU module's LED display.

Item	Inspection Item			Inspection	Criterion	Action
1	Mounting of base unit			Check that the fixing screws are not loose and the cover is not dislocated.	The screws and cover must be mounted securely.	Retighten the screws.
2	Installation of I/O modules			Check that the module is not dislocated and the unit fixing hook is engaged securely.	The module fixing hook must be engaged and installed correctly.	Securely engaged the module fixing hook.
3	Connecting conditions			Check for loose terminal screws.	Screws should not be loose.	Retighten the terminal screws.
				Check for distance between solderless terminals.	The proper clearance should be provided between solderless terminals.	Correct.
				Check the connector part of the cable.	Connections should not be loose.	Retighten the connector fixing screws.
4	Module indication LED	Power supply module	[POWER] LED	Check that the LED is ON.	The LED must be ON (green). (Abnormal if the LED is OFF or ON (red)).	Refer to QCPU User's Manual (Function Explanation, Program Fundamentals)
		PLC CPU module	[MODE] LED	Check that the LED is ON.	The LED must be ON (green). (Abnormal if the LED is OFF or flickering.)	
			[RUN] LED	Check that the LED is ON in RUN status.	The LED must be ON. (Abnormal if the LED is OFF.)	
			[ERR.] LED	Check that the LED is OFF.	The LED must be OFF. (Abnormal if the LED is ON or flickering.)	
			[BAT.] LED	Check that the LED is OFF.	The LED must be OFF. (Abnormal if the LED is ON.)	
		I/O module	Input LED	Check that the LED is ON/OFF.	The LED must be ON when the input power is turned ON. The LED must be OFF when the input power is turned OFF. (Abnormal if the LED does not turn ON or turn OFF as indicated above.)	Refer to I/O Module Type Building Block User's Manual
			Output LED	Check that the LED is ON/OFF.	The LED must be ON when the output power is turned ON. The LED must be OFF when the output power is turned OFF. (Abnormal if the LED does not turn ON or turn OFF as indicated above.)	

3. Display on 7-segment LED

The following characters can be used on 7-segment LED display of CNC CPU module.

0	1	2	3	4	5	6	7	8	9	(SP)	~	=	-	*	_
															
A:	B:	C:	D:	E:	F:	G:	H:	I:	J:	K:	L:	M:	N:	O:	P:
															
Q:	R:	S:	T:	U:	V:	W:	X:	Y:	Z:						
															

CNC state is displayed with the characters in the table above.

Display on 7-segment LED	State	Remarks
	Normal	Only the period in the first digit flickers.
	WDT error	
	- CNC CPU battery voltage detection level low (Less than 2.7V) - The battery voltage to be supplied to the absolute position detector dropped and the alarm "S52 Servo warning 009F" occurred. (Note 1)	
	- CNC CPU battery voltage detection level low (Less than 2.5V) - The battery voltage to be supplied to the absolute position detector dropped and the alarm "Z71 Abs encoder failure" occurred. - The absolute position was lost and the alarm "Z70 Abs data error 0101" occurred. (Note 1)	
	Alarm	“AL” flickers 3 times, and then the details appear. (See Appendix 3.1 "Detailed Display of Alarm/Stop Codes")
	Stop code	“STP” flickers 3 times, and then the details appear.(See Appendix 3.1 "Detailed display of alarm/stop codes")
Others	Detailed display of alarm/stop codes	7-segment LED displays the details of the alarm/stop code occurred (See Appendix 3.1 "Detailed display of alarm/ stop codes")

(Note 1) CNC CPU LED display will not change even if a battery alarm occurs to another CPU, such as a PLC CPU.

(Note 2) If any of the following alarms occurs independently, 7-segment LED still displays the normal state, “run.”.

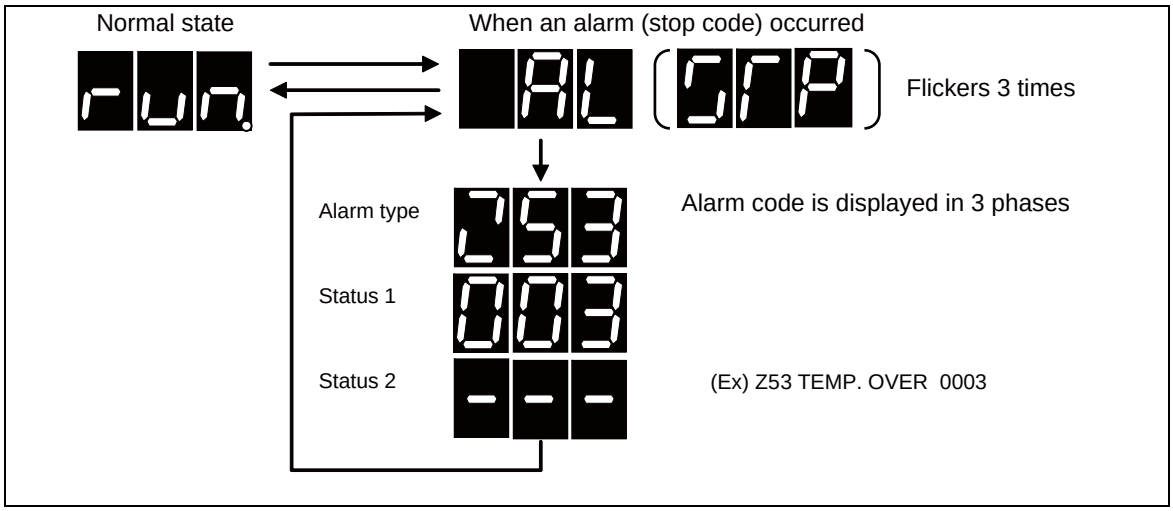
- EMG EMERGENCY STOP EXIN
- M01 OPERATION ERROR 0109 (Block start interlock)
- M01 OPERATION ERROR 0110 (Cutting block start interlock)

3.1 Detailed display of alarm/stop codes

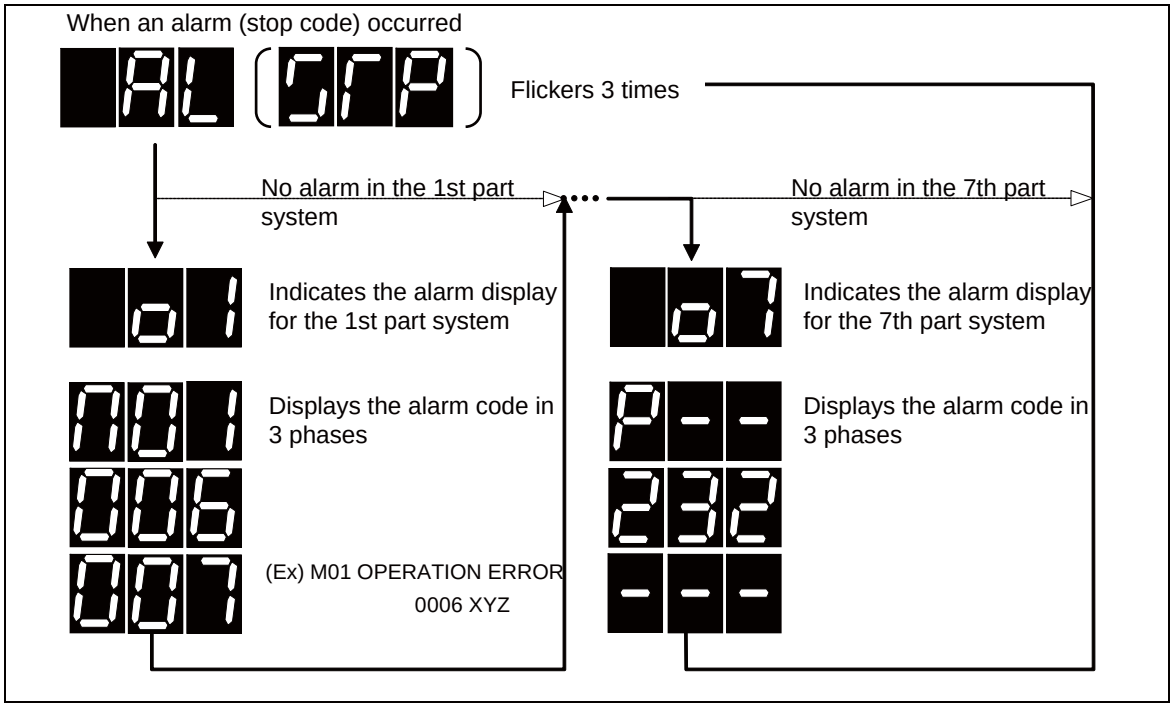
An alarm/stop code occurrence is displayed in 2 steps, report and details .

Alarm (“AL”) or stop code (“STP”) display flickers 3 times.

Details: Alarm code is displayed in 3 phases.



When multi-part system is used, the part system No. shows first, and then the alarm code appears.



3.2 Notes

(1) Display priority

When several alarms occurred at the same time, the most crucial alarm is selected according to the following chart and displayed.

Alarm type	Priority
WDT error	High ↑ ↓ Low
Battery alarm	
Multi-CPU error	
System alarm	
Servo alarm	
MCP alarm	
Emergency stop	
Built-in PLC alarm	
Program error	
Servo warning	
MCP warning	
System warning	
Operation error	
Stop code	

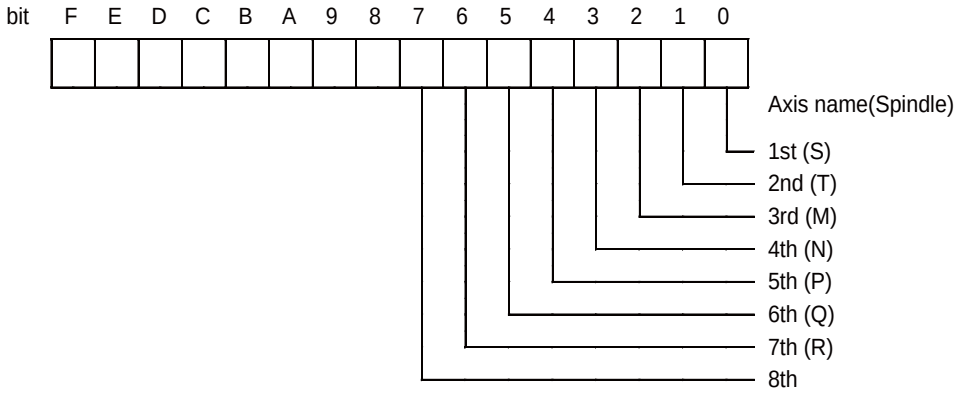
(2) Display of axes

Each axis is allocated to each bit according to the following rule. Hexadecimal number is displayed on 7-segment LED.

(Note 1) "*" in the first digit indicates the spindle.

(Note 2) "_" in the first digit indicates PLC axis.

(Note 3) When an error occurred on several axes, one of the axes is indicated. The indication priority is; (1) NC axis, (2) PLC axis and then (3) spindle.



(Example 1) “004” (bit2 is ON) for 3rd NC axis

(Example 2) “003” (bit0 and bit1 is ON) for 1st and 2nd NC axis

(Example 3) “*01” (bit0 is ON) for spindle(S)

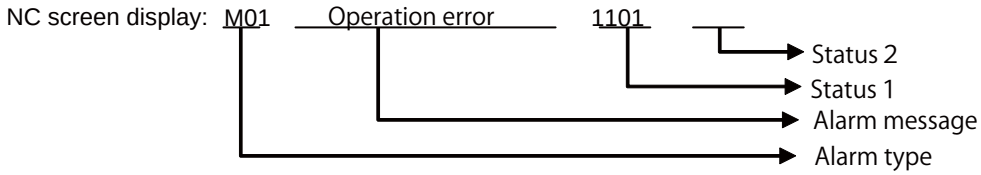
(Example 4) "_28" (bit3 and bit5 are ON) for 4th PLC axis and 6th PLC axis

(Example 5) "011" (bit0 and bit1 are ON) for 1st and 5th NC axes, 2nd PLC axis and spindle(S)

(3) Status display

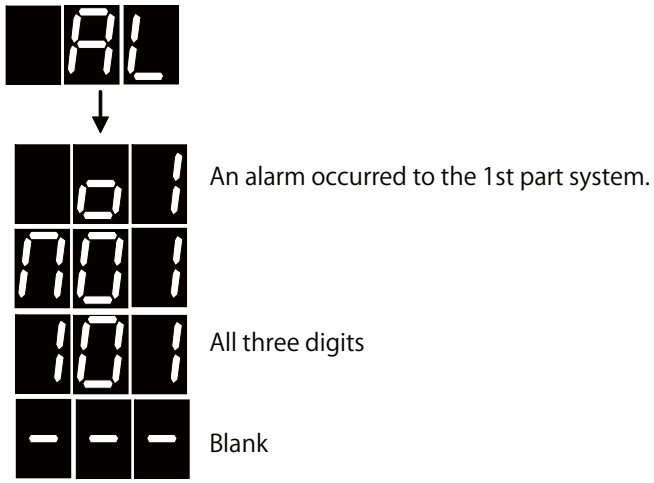
Each alarm type has a different display method for the status with over 3 digits. Refer to Appendix 3.3 "Examples of LED Display" for the display methods in details.

Also, for an alarm without Status 2, the Status 1 will be displayed in twice according to its number of digits.



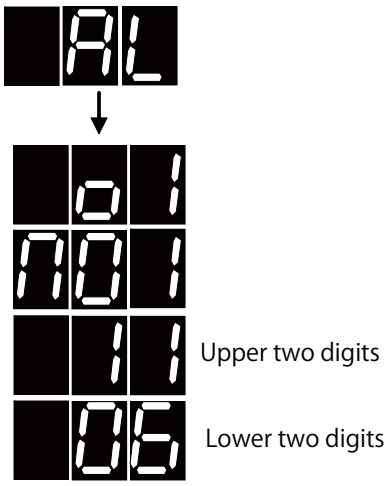
(a) When the Status 1 is a three-digit number (The highest-order digit is "0" when displayed in HEX.)

(EX) M01 Operation error 0101

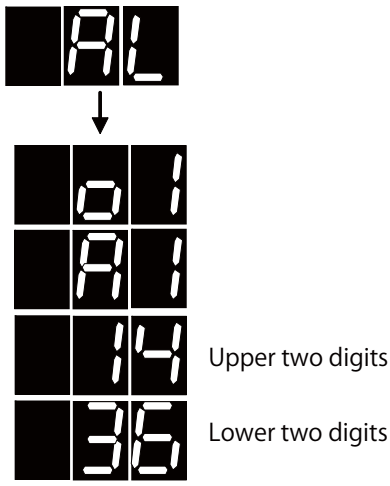


(b) When the Status 1 is a four-digit number

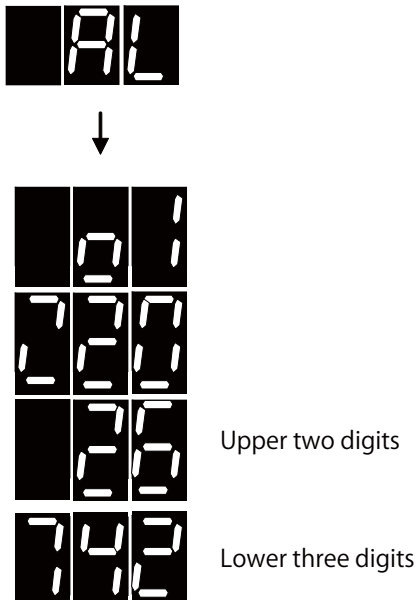
(Ex.1) M01 Operation error 1106



(Ex.2) A01 Multi CPU error 1436



(c) When the Status 1 is a five-digit number
(Ex.) Z20 Power ON again 26742



3.3 Examples of LED Display

This section shows an example of LED display for each error.

(1) Multi-CPU error

Alarm/warning code									7-segment LED display (Transition)
									1L 1C 1R / 2L 2C 2R / 3L 3C 3R
A01 MULTI CPU ERROR 1436									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Error code in high order			Error code in low order			

(Note) An error code is displayed in high/low order with 2 digits each.

(2) System alarm

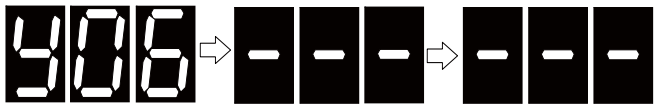
Alarm/warning code									7-segment LED display (Transition)
									1L 1C 1R / 2L 2C 2R / 3L 3C 3R
Z70 ABS ILLEGAL 0001 X									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			
Z71 DETECTOR ERROR 0002 Y									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			
Z73 ABS WARNING 0003 XYZ									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			

(3) Servo alarm

Alarm/warning code									7-segment LED display (Transition)
									1L 1C 1R / 2L 2C 2R / 3L 3C 3R
S01 SERVO ALARM : PR 0031 X									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			
S02 INIT PARAM ERR 2225 3 (3rd PLC axis)									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			
S02 INIT PARAM ERR 13225 S									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			
S03 SERVO ALARM : NR 0052 Y									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			
S04 SERVO ALARM : AR 006F Y									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			

(Note) Status 1 for [S02 Initial parameter error] displays the last 3 digits.

(4) MCP alarm

Alarm/warning code									7-segment LED display (Transition) 1L 1C 1R / 2L 2C 2R / 3L 3C 3R
Y02 SYSTEM ALARM 0051 0004									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Error details			
Y03 AMP UNEQUIPPED XYZ									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Axis name			Blank			
Y06 mcp_no ERROR									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Blank			Blank			
Y07 Conn Ax Excess 000F									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Blank			
Y09 Set Ax No Exces 0001 0001									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Error channel			
Y10 Drv SW different X									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Axis name			Blank			
Y11 Node detect error 8002 0100									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Channel, station No.			
Y14 VIR. AX AMP EQU. X									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Axis name			Blank			
Y20 Safety OBS ERR 0008									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Blank			

(Note) Axis Nos. of “Data ID” or “Recv frame No.” error under [Y02 System alarm 0051] are not displayed in the error display area.

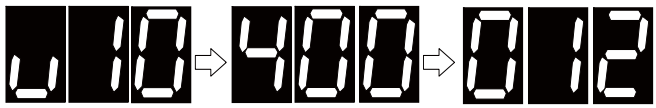
(5) Emergency stop

Alarm/warning code									7-segment LED display (Transition)		
									1L 1C 1R	2L 2C 2R	3L 3C 3R
EMG EMERGENCY STOP											
1L	1C	1R	2L	2C	2R	3L	3C	3R			
Alarm type			Cause			Blank					

(Note) The following table shows the LED display of emergency stop causes for status 1.

Emergency stop cause	LED display
PLC	PLC
SRV	SRV
STOP	STP
SPIN	SPN
DATA	DAT
PARA	PAR
STP2	SP2
LAD	LAD
MULT	MLT
IPWD	IPD
CVIN	CVN
MCT	MCT
SUIN	SIN
LINK	LNK
APLC	APL

(6) Bulit-in PLC Alarms

Alarm/warning code									7-segment LED display (Transition)		
									1L 1C 1R	2L 2C 2R	3L 3C 3R
U10 Illegal PLC 0400 0012											
1L	1C	1R	2L	2C	2R	3L	3C	3R			
Alarm type			Alarm No.			Step No.					

(Note) Status 1 and 2 display the last 3 digits.

(7) Program error

Alarm/warning code									7-segment LED display (Transition)		
									1L 1C 1R	2L 2C 2R	3L 3C 3R
P273 MACRO OVERCALL											
1L	1C	1R	2L	2C	2R	3L	3C	3R			
Alarm type			Alarm No.			Blank					

(8) Servo warning

Alarm/warning code									7-segment LED display (Transition)
									1L 1C 1R / 2L 2C 2R / 3L 3C 3R
S51 PARAMETER ERROR 2205 Z									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			
S51 PARAMETER ERROR 13225 T									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			
S52 SERVO WARNING 00E0 X									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			

(Note) Status 1 for [S51 Parameter error] displays the last 3 digits.

(9) MCP warning

Alarm/warning code									7-segment LED display (Transition)
									1L 1C 1R / 2L 2C 2R / 3L 3C 3R
Y21 Safety OBS WRG 0001 Z									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			
Y51 PARAMETER ERROR 0012 Z									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			

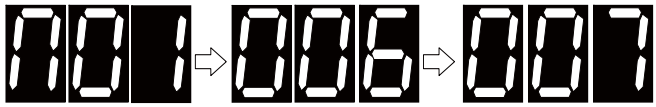
(10) System warning

Alarm/warning code									7-segment LED display (Transition) 1L 1C 1R / 2L 2C 2R / 3L 3C 3R
Z30 ETHERNET ERROR 0006 0003									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Communication type			
Z53 TEMP. OVER 0003									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Blank			
Z55 RIO NOT CONNECT 0007 0007									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Control unit			Board unit			
Z59 TIME CONSTANT									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Blank			Blank			
Z20 Power ON again 1002									 (Note 1)
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Para No. Up 2 digits			Para No. Low 2 digits			
Z20 Power ON again 26701									 (Note 1)
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Para No. Up 2 digits			Para No. Low 3 digits			
Z20 Power ON again									 (Note 2)
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Blank			Blank			

(Note 1) The Nos. of parameters which are validated by turning the power ON again will be displayed divided into upper and lower digits.
A 4-digit parameter No. will be divided into upper 2 digits and lower 2 digits.
A 5-digit parameter No. will be divided into upper 2 digits and lower 3 digits.

(Note 2) The alarm "Z20 Power ON again" will occur when inputting parameter data or SRAM.BIN data, but the parameter No. will not be displayed. And the display on 7-segment LED will be blank.

(11) Operation error

Alarm/warning code									7-segment LED display (Transition) 1L 1C 1R / 2L 2C 2R / 3L 3C 3R
M01 OPERATION ERROR 0006 XYZ									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			
M01 OPERATION ERROR 1005 XYZ									 (Note 1)
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Axis name			
M01 OPERATION ERROR 1106									 (Note 2)
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No. Up 2 digits			Alarm No. Low 2 digits			

(Note 1) The alarms with two status, alarm No. and axis name, like [M01 Operation error 1005], "10" will be displayed as "A".

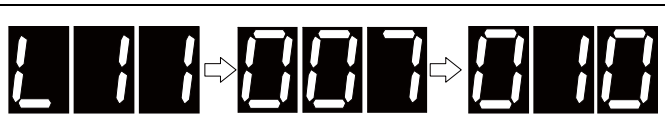
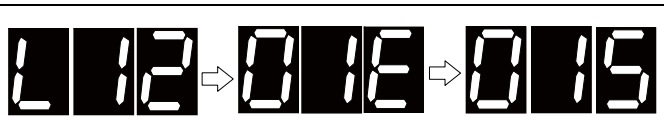
(Note 2) A 4-digit alarm No., like [M01 Operation error 1106], will be divided into upper 2 digits and lower 2 digits when displayed.

(12) Stop code

Alarm/warning code									7-segment LED display (Transition) 1L 1C 1R / 2L 2C 2R / 3L 3C 3R
T01 CAN’T CYCLE ST 0105									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Blank			
T02 FEED HOLD 0202									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Blank			
T03 BLOCK STOP 0301									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Blank			
T10 FIN WAIT 0001									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Blank			

(Note) The factors “In dwell execution” or “unclamp signal wait” under [T10 FIN WAIT] are not displayed in the error display area.

(13) Network Errors

Alarm/warning code									7-segment LED display (Transition) 1L 1C 1R / 2L 2C 2R / 3L 3C 3R
L10 DN INIT. ERR. 1 00E0 0010									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			I/O No.			
L11 DN INIT. ERR. 2 0007 0010									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			I/O No.			
L12 DN LINK ERROR 001E 15									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Slave No.			
L13 DN MESSAGE ERR. 258									
1L	1C	1R	2L	2C	2R	3L	3C	3R	
Alarm type			Alarm No.			Blank			

(Note) The slave node No. of [L12 DN LINK ERROR] will be displayed in decimal.

4. Periodic Inspection

The items that must be inspected one or two times every 6 months to 1 year are listed below.
When the equipment is moved or modified, or layout of the wiring is changed, also implement this inspection.

Item	Inspection item		Inspection	Criterion	Action
1	Ambient environment	Ambient temperature	Measure with a thermometer and a hygrometer. Measure corrosive gas.	0 to 55 ° C (32 to 131 ° F)	When the controller is used in the board, the ambient temperature in the board becomes the ambient temperature
		Ambient humidity		5 to 95 % RH	
		Atmosphere		No corrosive gases	
2	Power voltage		Measure a voltage across the terminals of 100/ 200VAC and 24VDC.	85 to 132VAC	Change the power supply.
				170 to 264VAC	
				15.6 to 31.2VDC	
3	Installation	Looseness, rattling	Move the module to check for looseness and rattling.	The module must be installed solidly.	Retighten the screws. If the CPU modules, Motion modules, I/O modules, or power supply modules are loose, fix it with screws.
		Adhesion of dirt and foreign matter	Check visually.	Dirt and foreign matter must not be present.	Remove and clean.
4	Connection	Looseness of terminal screws	Try to further tighten screws with a screwdriver.	Screws must not be loose.	Retighten the terminal screws.
		Proximity of solderless terminals to each other	Check visually.	Solderless terminals must be 4 positioned at proper intervals.	Correct.
		Looseness of connectors	Check visually.	Connectors must not be loose.	Retighten the connector fixing screws.
5	Battery		Check the 7-segment LED at the front side of CPU module.	Must be turned OFF "BT1" or "BT2" display.	Even if the lowering of a battery capacity is not shown, replace the battery with a new one if a service life time of the battery is exceeded.
			Check the length of term after purchasing the battery	Must not be used more than 5 years.	

5. Hardware Replacement Methods

5.1 Module Installation

5.1.1 Instructions for Handling

⚠ CAUTION

(1) Use C70 in an environment that meets the general specifications contained in this manual. Using C70 in an environment outside the range of the general specifications could result in electric shock, fire, operation failure, and damage to or deterioration of the product.

(2) While pressing the installation lever located at the bottom of module, insert the module fixing projection into the fixing hole in the base unit until it stops. Then, securely install the module with the fixing hole as a supporting point. Incorrect installation of the module can cause an operation failure, damage or drop.

(3) While pressing the installation lever located at the bottom of module, insert the module fixing projection into the fixing hole in the base unit until it stops. Then, securely install the module with the fixing hole as a supporting point. Incorrect installation of the module can cause an operation failure, damage or drop.

(4) All screws for the module must be tightened to avoid drop.
Module fixation screws (M3 x 12) must be prepared by the user. Use provided module fixation screws (M3 x 13) for CNC CPU module.

(5) Tighten the screw in the specified torque range. Loose screw may cause a drop, short circuit or operation failure. Over tightening may cause a drop, short circuit or operation failure due to damage to the screw or module.

(6) Be sure to connect the extension cable to connectors of the basic base unit correctly. After connecting, check them for looseness. Poor connections could cause an input or output failure.

(7) Completely turn off the externally supplied power used in the system before installation or removing the module. Not doing so could result in electric shock or damage to the product.

(8) Do not install/remove the module onto/from base unit or terminal block more than 50 times, after the first use of the product. Failure to do so may cause the module to malfunction due to poor contact of connector.

(9) Do not directly touch the module's conductive parts and electronic components. Doing so may cause an operation failure or give damage to the module.

(10) Do not touch the heat radiating fins of controller or servo amplifier's, regenerative resistor and servomotor, etc. while the power is ON and for a short time after the power is turned OFF. In this timing, these parts become very hot and may lead to burns. Remove the modules while paying attention.

(11) Always remove the module by removing the module fixing screw and then taking the module fixing projection off the module fixing hole of the base unit. Attempting to remove the module by force may damage the module fixing projection.

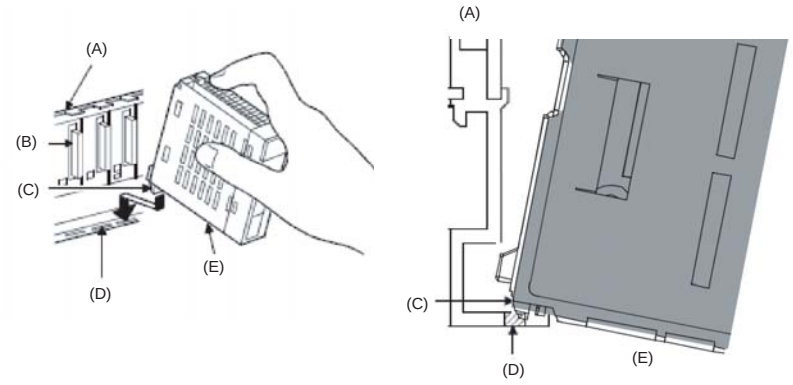
5.1.2 Installation and removal of module

This section describes how to install and remove a power supply module, PLC CPU module, CNC CPU module, I/O module, intelligent function module or another module to and from the base unit.

Installation and removal of the module from Q3□DB/Q6□B base unit

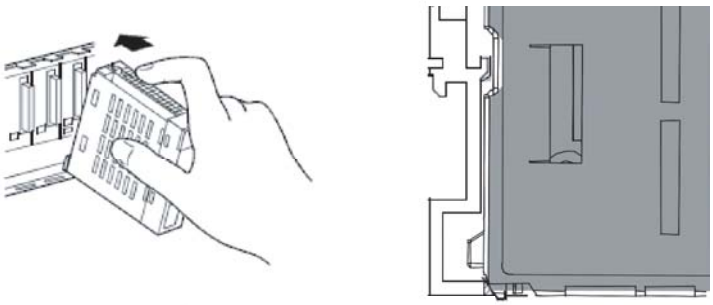
Installation

(1) Securely insert the module fixing projection into the module fixing hole so that the latch is not misaligned.



(A) Base unit
(B) Module connector
(C) Module fixing projection
(D) Module fixing hole
(E) Module

(2) Using the module fixing hole as a fulcrum, push the module in the direction of arrow to install it into the base unit.



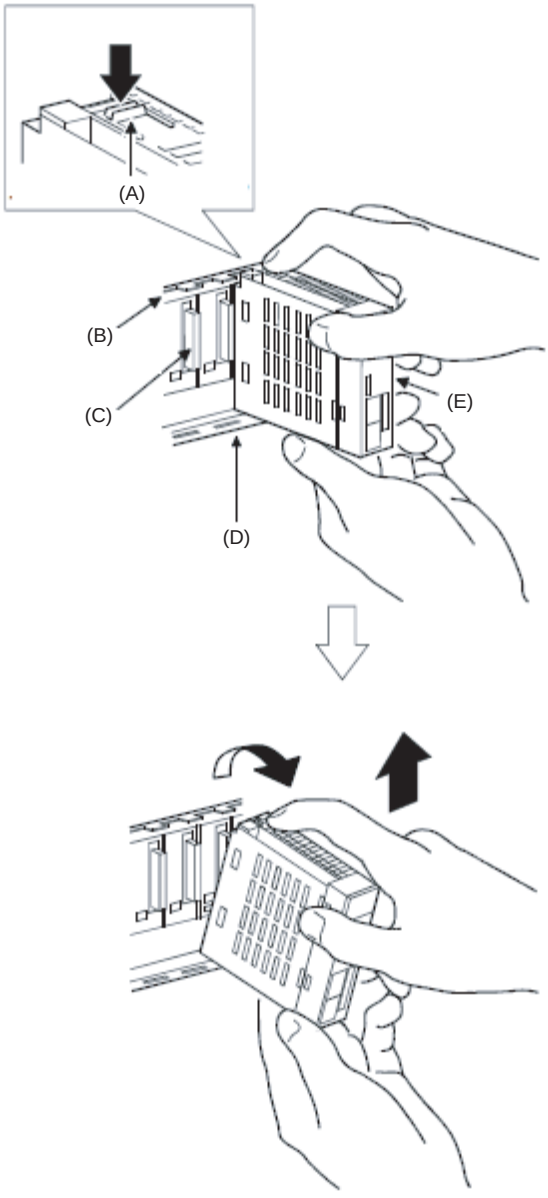
(3) Make sure that the module is installed in the base unit securely.

(4) Tighten the screw of the base unit.

(Note) To avoid drop, all screws of the module must be tightened.
Module fixation screw must be prepared by the user.
Use provided module fixation screw (M3 x 13) for CNC CPU module.

Removal

(1) Remove the module fixation screws.
(2) Support the module with both hands and securely press the module fixing hook with your finger.
(3) Pull the module based on the supporting point of module bottom while pressing the module fixing hook.
(4) While lifting a module, take off the module fixing projection from the module fixing hole.



(A) Module fixing hook
(B) Base unit
(C) Module connector
(D) Module fixing hole
(E) Module

5.2 Battery for CNC CPU

5.2.1 Battery Life for CNC CPU

Module type	Battery type	Battery life (Total power failure time) [h]*1		
		Power-on time ratio*2	Guaranteed value*3 (MIN) (75 ℃)	Guaranteed value*4 (TYP) (40 ℃)
Q173NCCPU	External battery (Q6BAT)*6	0%	20000	43800
		30%	27000	
		50%	31000	
		70%	36000	
		100%	43800	

*1 The actual service value indicates the average value, and the guaranteed value indicates the minimum value.

*2 The power-on time ratio indicates the ratio of Multiple CPU system power-on time to one day (24 hours). (When the total power-on time is 17 hours and the total power-off time is 7 hours, the power-on time ratio is 70%.)

*3 The guaranteed value (MIN) ; equivalent to the total power failure time that is calculated based on the characteristics value of the memory (SRAM) supplied by the manufacturer and under the storage ambient temperature range of -25°C to 75°C (-13 to 167°F) (operating ambient temperature of 0°C to 55°C (32 to 131°F)).

*4 The guaranteed value (TYP) ; equivalent to the total power failure time that is calculated based on the normal air-conditioned environment (40°C (104°F)).

*5 Set the battery (Q6BAT) to battery holder unit (Q170DBATC).

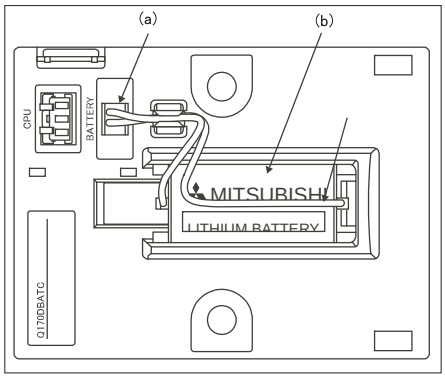
(Note 1) Do not use the battery exceeding its guaranteed life.

(Note 2) The self-discharge influences the life of battery without the connection to Q6BAT. The external battery should be exchanged approximately every 4 or 5 years. And, exchange the battery with a new one in 4 to 5 years even if a total power failure time is guaranteed value or less.

(Note 3) In the following status, the backup time after power OFF is 3 minutes.
- The battery connector/Q6BAT lead connector is disconnected.
- The battery cable/lead wire of Q6BAT is broken.

5.2.2 CNC CPU Battery Replacement Procedure

Battery holder unit Q170DBATC

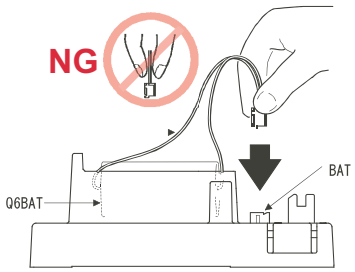


(a) Lead connector
(b) Battery Q6BAT

(1) Turn C70 power OFF

(2) Remove the old battery from its holder.

(3) Install a new battery into the holder in the correct direction. Connect the lead connector to the connector (BATTERY).



(4) Turn C70 power ON.

(Note 1) Press the connector firmly while replacing the battery.
Always make sure that short-circuit between electric cables is impossible or not to lead to a dangerous state when wiring the connector. Put in the connector to the point where it clicks.
(Note 2) Make sure that short-circuit between cables is impossible or not to lead to a dangerous state when wiring.

5.3 Battery inside PLC CPU

5.3.1 Battery Life

PLC CPU module type	Power-on time ratio *1	Battery life		
		Guaranteed value *2 (70 ℃)	Actual service value *3 (40 ℃)	Backup time after alarm *4
Q03UDCPU	0%	25300hr	43800hr	600hr
	30%	36100hr	43800hr	600hr
	50%	43800hr	43800hr	600hr
	70%	43800hr	43800hr	600hr
	100%	43800hr	43800hr	600hr
Q04UDHCPU	0%	4300hr	32100hr	384hr
	30%	6100hr	43800hr	384hr
	50%	8600hr	43800hr	384hr
	70%	14300hr	43800hr	384hr
	100%	43800hr	43800hr	384hr
Q06UDHCPU (When a file register is used within 128k points)	0%	4200hr	32100hr	384hr
	30%	6000hr	43800hr	384hr
	50%	8400hr	43800hr	384hr
	70%	14000hr	43800hr	384hr
	100%	43800hr	43800hr	384hr
Q06UDHCPU (When a file register is used within 128k points)	0%	2300hr	19200hr	192hr
	30%	3200hr	27400hr	192hr
	50%	4600hr	38400hr	192hr
	70%	7600hr	43800hr	192hr
	100%	43800hr	43800hr	192hr

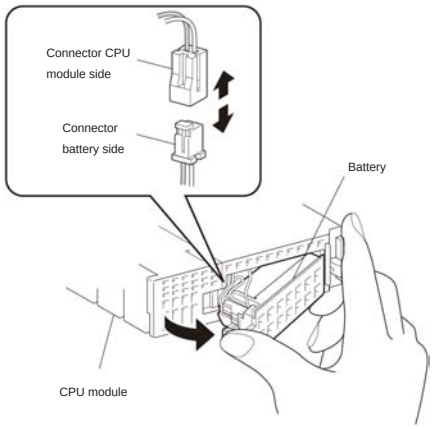
*1 The power-on time ratio indicates the ratio of Multiple CPU system power-on time to one day (24 hours). (When the total power-on time is 17 hours and the total power-off time is 7 hours, the power-on time ratio is 70 %.)

*2 The guaranteed value; equivalent to the total power failure time that is calculated based on the characteristics value of the memory (SRAM) supplied by the manufacturer and under the storage ambient temperature range of -25°C to 75°C (-13 to 167°F) (operating ambient temperature of 0°C to 55°C (32 to 131°F)).

*3 The actual service value; equivalent to the total power failure time that is calculated based on the measured value and under the storage ambient temperature of 25°C (77°F). This value is intended for reference only, as it varies with characteristics of the memory.

*4 In the following status, the backup time after power OFF is 3 minutes.
- The battery connector is disconnected.
- The battery lead wire is broken.

5.3.2 Replacement Procedure for battery inside PLC CPU



(1) Turn the C70 power OFF.

(2) Remove the CPU module from the base unit.

(3) Open the CPU module bottom cover.

(4) Remove the old battery from its holder.

(5) Remove the connector from the connector stopper, and remove the lead from the connector.

(6) Insert a new battery into the holder in the correct direction. Connect the lead to the connector.

(7) Close the CPU module bottom cover.

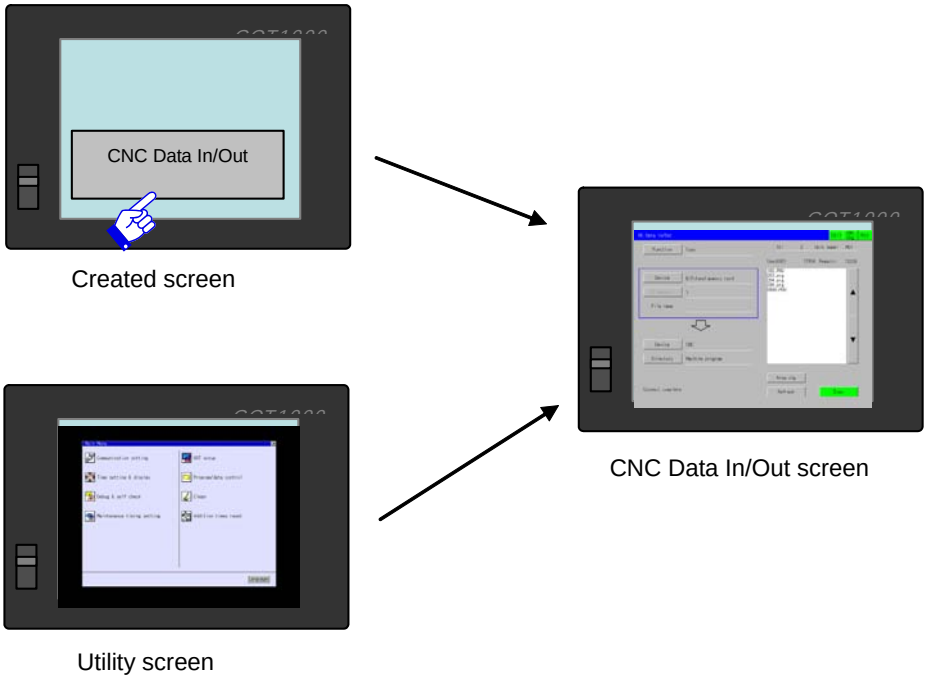
(8) Install the CPU module on the base unit.

(9) Turn the C70 power ON.

6. CNC Data Input/Output

6.1 Screen Transition to the "CNC Data In/Out" Screen

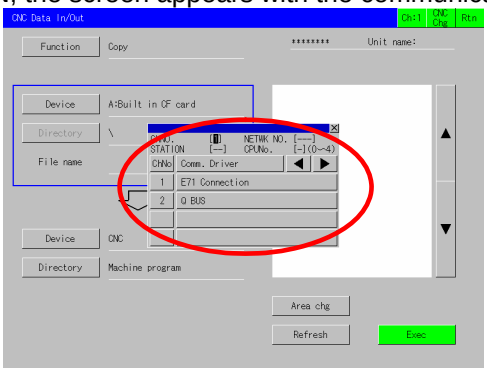
The "extended function button", which is allocated to the drawing data, initially shows the operation "CNC Data In/Out". Touching the button switches the screen to the "CNC Data In/Out" screen.
Touching the "CNC Data In/Out" button in the Utility screen also switches the screen to the "CNC Data In/Out" screen.
The "CNC Data In/Out" screen is scaled to SVGA (800x600). If the display used is wider than SVGA, the screen appears in the middle of the display. The area other than the screen appears black.



[Screen transition to the "CNC Data In/Out" screen]

6.2 Setting Communication

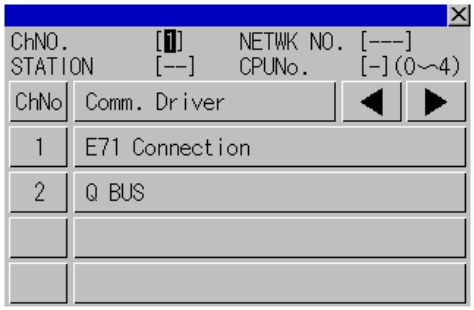
(1) When the screen is switched to the "CNC Data In/Out" screen at the first time since the power has been turned ON, the screen appears with the communication setting window opened.



[Initial display of "CNC Data In/Out" screen]

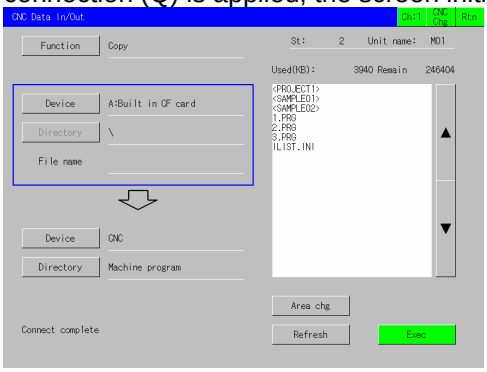
(2) Select the channel to connect in the communication setting window. If any of the channel Nos. or the network driver names is pressed, the channel is set and the window is closed.

(Example) When connecting to a display I/F, touch "1" or "E71"



[Display of the communication setting window]

(3) The initial "CNC Data In/Out" screen appears.
If the display I/F connection is applied, the screen initially shows the CNC that has been set in the first drawing data.
If the bus connection (Q) is applied, the screen initially shows the CNC with the smallest No.



[Initial display of "CNC Data In/Out" screen]

(4) To change CNC modules, press CNC Chg button.

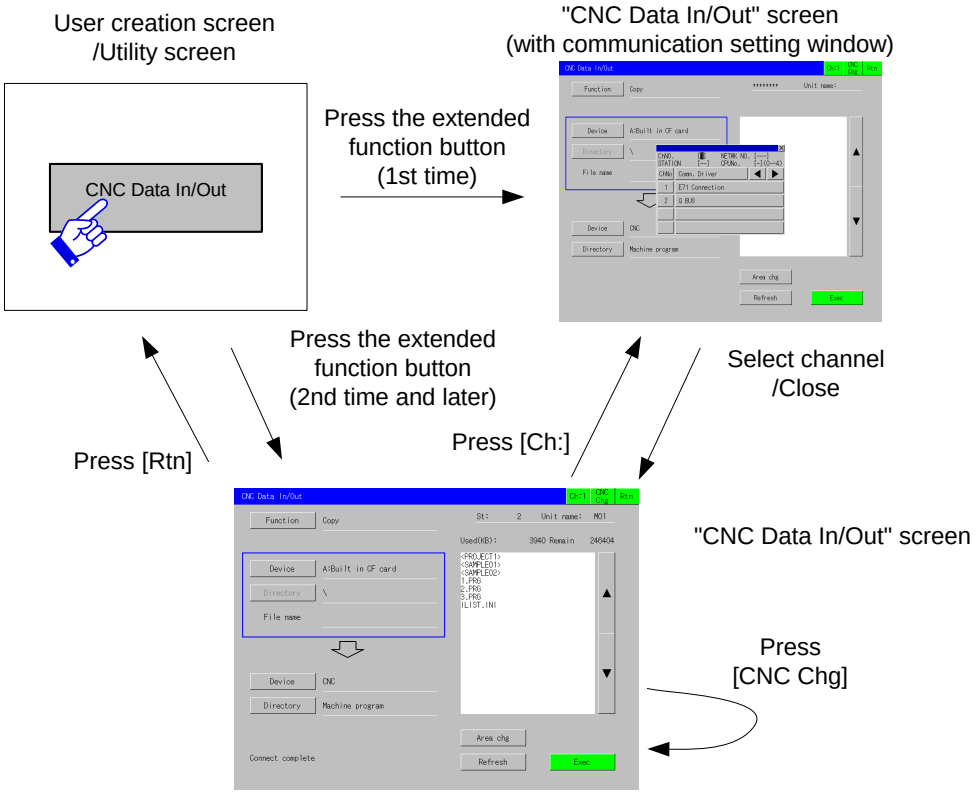
If display I/F connection is applied, the CNC that is set next in the drawing data is displayed. When the CNC to be changed is the last one, the CNC at the top is displayed.
If bus connection (Q) is applied, the No. is incremented. Devices other than CNC are skipped. When the CNC to be changed is the last one, the first CNC is displayed.
The screen is as same as that in the step (3).
If only one connection device is set, the CNC Chg button is grayed and unavailable.

(5) To change the channel, press the Ch: button. The communication setting window pops up.
The operations and the screen features are same as those in the step (2).

(6) To close the "CNC Data In/Out" screen, press the Rtn button. The screen returns to the initial screen.

(7) If the "extended function switch" ("CNC data In/Out" button) is pressed again, the communication is performed through the selected channel. The communication setting window does not appear.

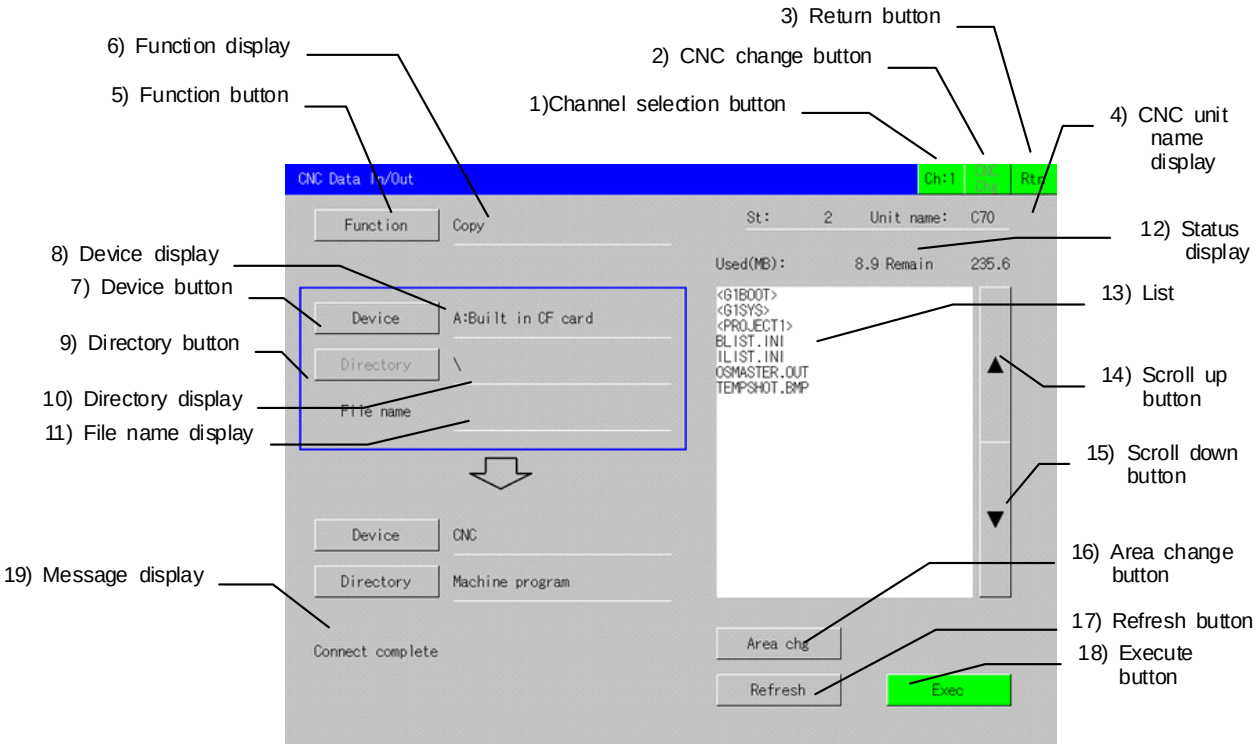
The following picture shows how the screen is changed.



[Screen transition from "CNC Data In/Out" screen]

6.3 "CNC Data In/Out" Screen Display

The "CNC Data In/Out" screen can be used to copy the files in the CNC main unit or memory card (*Note) in GOT to the other device, as well as to delete these files. Files in the memory card cannot be copied to the other memory card.
Memory card is a generic term for a USB drive, an extended memory card, and a built-in CF card. An extend memory card or a built-in CF card can be used for GT15. A USB drive, an extend memory card or a built-in CF card can be used for GT16.



[CNC Data In/Out screen]

[Display items of "CNC Data In/Out" screen]

No.	Item	Details
1	Channel selection button	Displays the communication setting window. "#" in "Ch:#" shows the selected channel No.
2	CNC change button	Changes the CNC to connect. Changes station Nos. during display I/F connection, the device Nos. during bus connection. If only one CNC is set to connect, this button is grayed and unavailable.
3	Return button	Returns to the screen from which the current screen was called. This button can return the screen while the copy confirmation window appears.
4	CNC unit name display	Station No. is shown during display I/F connection. CPU No. is shown during bus connection (Q). "********" is shown during any other connection. The unit name of each CNC (#1135 unt_nm) is shown in "Unit name".
5	Function button	Displays the function selection window. Refer to "6.5 Selecting the Function"
6	Function display	Displays the selected function (Copy or Delete).
7	Device button	Displays the device selection window. Refer to "6.6 Selecting the Device"
8	Device display	Displays the selected device. <For GT15> CNC/B: Extend Memory card /A: Built-in CF card (The extend memory card will be selected after the transition to this screen or after the CNC changeover. When the extend memory card is not inserted, the built-in CF card will be displayed.) <For GT16> CNC/E: USB drive /B: Extend Memory card /A: Built-in CF card (The USB drive will be selected after the transition to this screen or after the CNC changeover. When the USB drive is not mounted, the extend memory card will be displayed. When the extended memory drive is also not inserted, the built-in CF card will be displayed.)
9	Directory button	If CNC is selected, the CNC data selection window is displayed. Refer to "3.7 Selecting the Directory" If the memory card is selected, this button is grayed and unavailable.
10	Directory display	Displays the selected directory. (Note) If CNC is selected, names of the CNC data (machining program/ parameter/ tool offset/ workpiece offset/ common variable/ maintenance data/ cycle monitor) are displayed. If a memory card is selected, the directory name is displayed with left aligned. (Example) \ is used for root directory SAMPLE01 directory is indicated "\ SAMPLE 01\".
11	File name display	Displays the file name selected from the list.(Note)

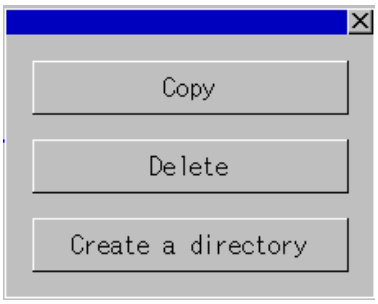
(Note) Up to 28 characters can be displayed on the screen. Characters from 29th are not displayed.
A directory or file name can use up to 78 characters as full pass.
Up to 75 characters can be set when any file exists in the root directory.
Up to 20 directory trees can be set.

No.	Item	Details
12	Status display	Displays information of the selected device. When CNC is selected <Display example> Entry: 57 Remain 343 Character: 31802 Remain 197250 "Entry" and "Remain": Displays the number of user machining programs entered and the remaining number of programs that can be entered. "Character" and "Remain": Displays the number of characters entered in user machining programs and the remaining number of characters that can be entered. The remaining number of characters is displayed in 250-character units. When memory card is selected <Display example> Used(MB): 8.9 Remain 235.6 "Used" and "Remain" Displays the used capacity and open capacity. If the value is not dividable by 1024, it is rounded to the nearest tenth above. **(Example)** If 17863 byte is used: 17863 / 1024 = 17.44… →rounded up to 17.5KB
13	List	The directories and files, which are stored in the directory as the display shows, are listed. If the device is a memory card, directories are displayed in brackets "< >". Press one of the directory names in the list to move to the directory. "<..>" indicates the directory one above. A directory name and file name can be displayed within 28 characters. The name with over 28 characters is not full name displayed.
14	Scroll up button	Scrolls one page up.
15	Scroll down button	Scrolls one page down.
16	Area change button	The area bounded by heavy line is changed and the list is refreshed. This button is grayed and unavailable when "Delete" is selected.
17	Refresh button	Refresh the list currently selected. Refer to "6.11 Refreshing the List".
18	Execute button	Executes the operation as set in the screen.
19	Message display	Displays a result for the operation. For display details, see "6.16.3 Messages". While a message of the function being executed is displayed, "■" before the message blinks.

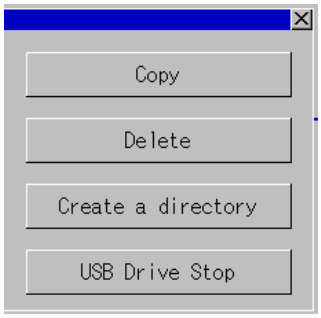
6.4 Operation Windows

6.4.1 Function Selection Window

Select the function.



〈For GT15〉

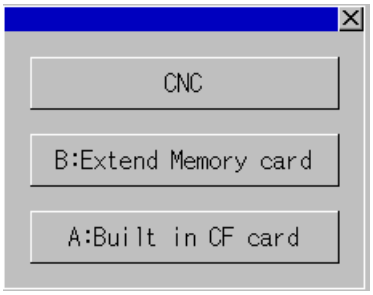


〈For GT16〉

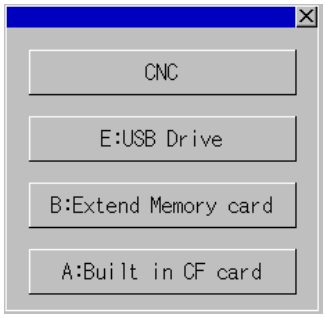
Copy: Inputs/outputs files between CNC and a memory card.
Delete: Deletes files in CNC (machining programs only) and those in a memory card.
Create a directory: Create and name a directory at will in the memory card.
USB Drive Stop: USB drive will be stopped (GT16 only).

6.4.2 Device Selection Window

Select any device.



〈For GT15〉

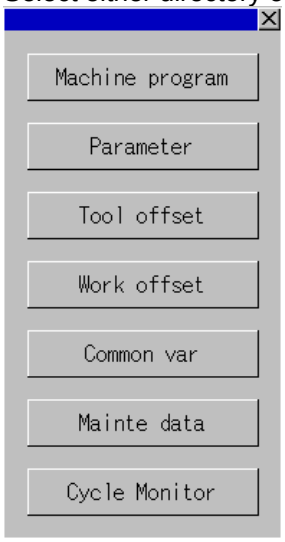


〈For GT16〉

CNC	Selects CNC.
E: USB drive	Selects USB drive (GT16 only).
B: Extend memory card	Selects extend memory card.
A: Built in CF card	Selects built-in CF card.

6.4.3 CNC Data Selection Window

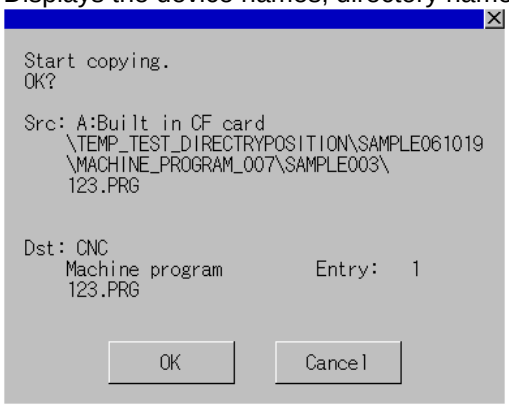
Select either directory of CNC.

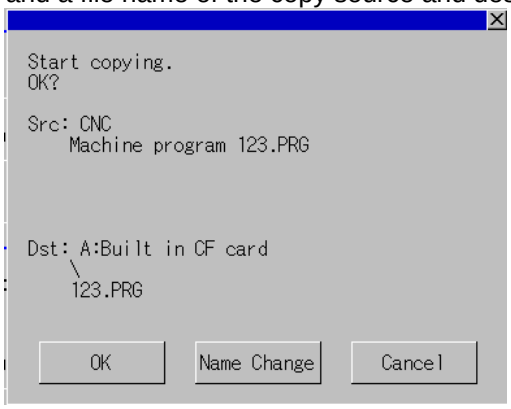


When any directory is set, files in the directory are shown in the list.

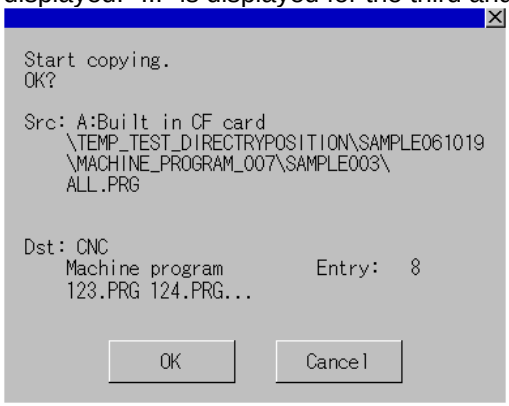
6.4.4 Copy Confirmation Window

This window appears when the Exec button is pressed for copy.
Displays the device names, directory names and a file name of the copy source and destination.





When the data is going to be copied to the machining programs in CNC, the file names and the number of the machining programs are displayed. When the data is copied to the memory card, a file name of the copy destination can be changed by pressing "Name Change" button on the window. Up to two file names are displayed. "..." is displayed for the third and later file names.

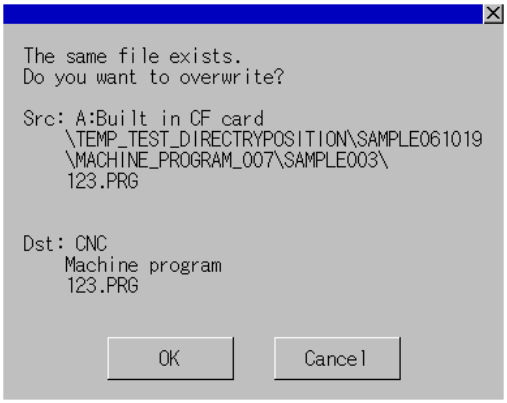


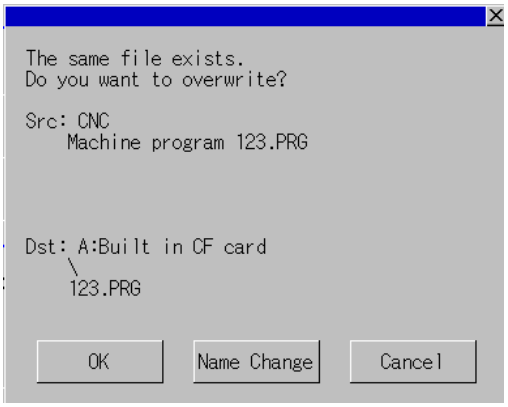
6.4.5 Overwrite Confirmation Window

This window appears when the OK button is pressed in the copy confirmation window while the same file name exists in the copy destination.

(1) One file display

Displays the information (device name, directory name and file name) of both copy source and destination.



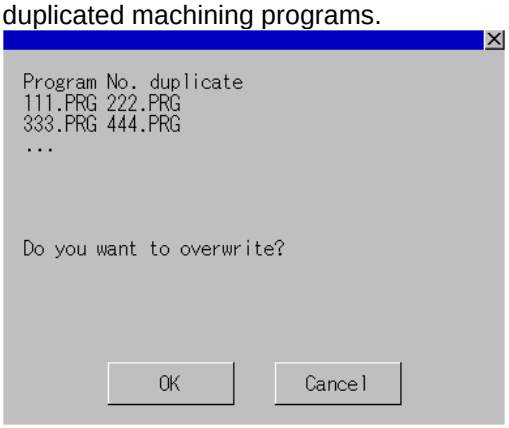


(Note 1) When the file is copied to CNC, the overwrite confirmation window is displayed only when machining programs are overwritten. The window will not be displayed for parameters or other files, which are automatically overwritten.

(Note 2) When the file is copied to a memory card, a file name of the copy destination can be changed. For the detail of how to change the name, refer to "3.14 Change a file name when outputting a file".

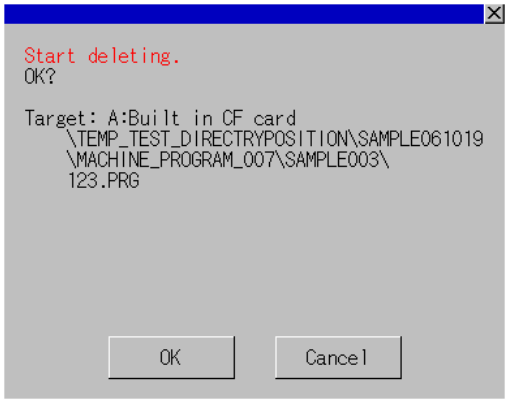
(2) Multiple files display

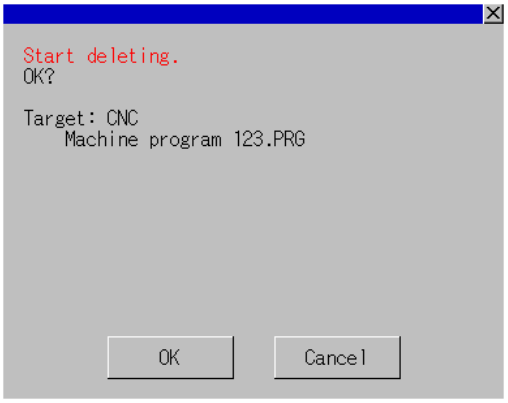
This window appears when the file with multiple machining programs is copied from a memory card to CNC in which one or more files with the same program No. exist. Up to four duplicated program Nos. are displayed. "..." is displayed for the fifth and later programs. Pressing OK overwrites all the duplicated machining programs.



6.4.6 Delete Confirmation Window

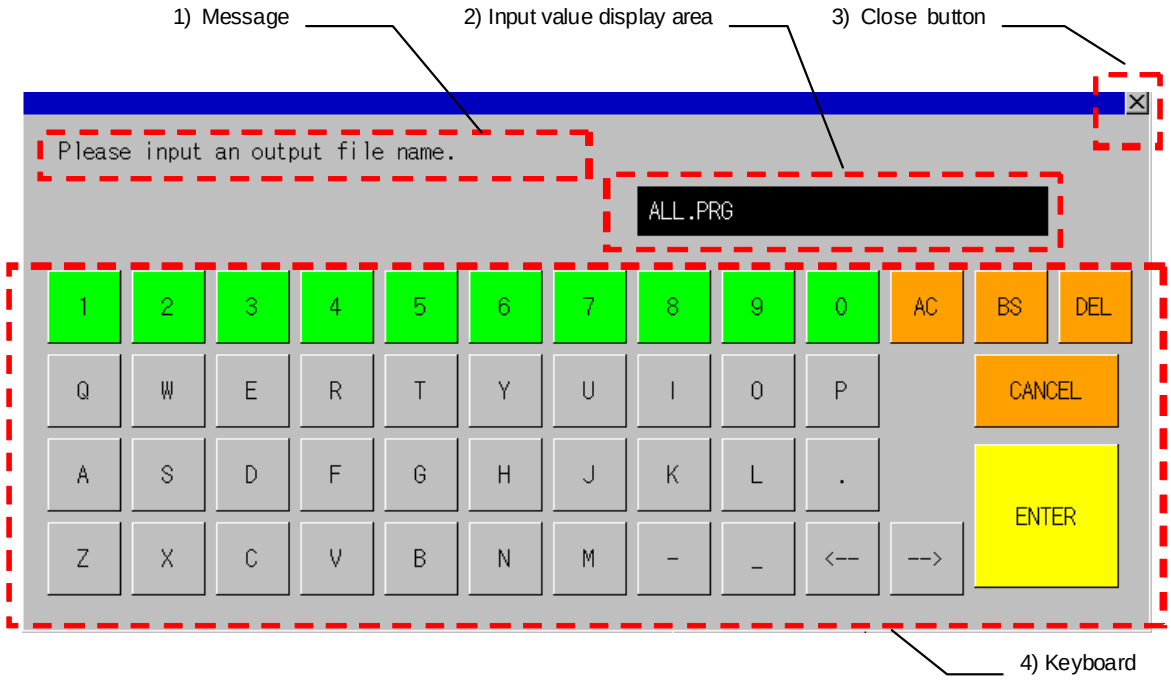
This window appears when the Exec button is pressed for delete.
Displays a device name, directory name and name of the file to be deleted.





6.4.7 Key Window

Key window is used to create a direcotry or output a file name.



[Key window display items]

No.	Items	Details
1	Message	Displays a message. In creating a directory, "Please input a directory name。" will be displayed. In changing the name of an output file, "Please input an output file name." will be displayed.
2	Input value display area	Displays the input value. Nothing is initially displayed in creating a directory. When changing an output file name, the old file name is displayed. An input value can be displayed within 28 characters.
3	Close button	Pressing the button closes the window. Then the input value will not be registered.

(Continued to the next page)

(Continued from the last page)

No.	Items		Details
4	Keyboard		
		"0" to "9", "A" to "Z", "-", "_" and "." buttons	Pressing the button displays the character of the button at the cursor position. 29th character will not be entered.
		"<-" and "->" buttons	The position of the cursor will be moved whe it is pressed. The cursor can move from the left of the first character to the right of the last character. Pressing the "<-" or "->" button when the cursor is at the beginning or end does not move the cursor.
		BS	Deletes the left character of the cursor. Pressing the button does not delete the character when there is no input value or when the cursor is at the left end.
		DEL	Deletes the right character of the cursor. Pressing the button does not delete the character when there is no input value or when the cursor is at the right end.
		AC	Clears all characters in the input value display area.
		CAN	Pressing the button closes the window. Then the input value will not be registered.
		ENTER	Closes the window with setting the input value. In creating a directory, pressing the button creates a directory. In changing the name of an output file, pressing the button displays a copy confirmation window again. When there is no input value, it will behave as the same manner as pressing the close button.

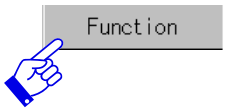
(Note) The "ENTER" button is shown only in insert mode. The button is not shown in overwrite mode.

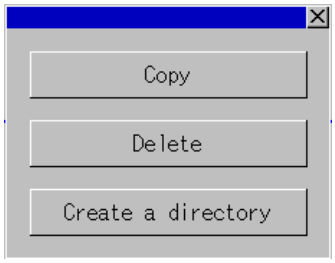
6.5 Selecting the Function

"CNC Data In/Out" screen has two functions, copy and delete.
"Copy" is used to copy a file selected in the upper area to the lower area. Devices in the upper and lower area are not changed by this function.
"Delete" is used to delete a file selected in the upper area. When CNC is set in the upper area, the directory in the same area is changed to "machining program".

- Changing to "Copy"

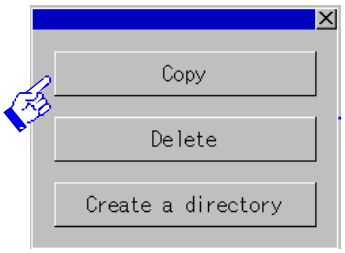
1) Touch the Function button. ⇨ The function selection window appears.





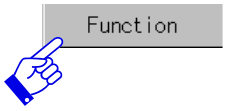
(For GT15)

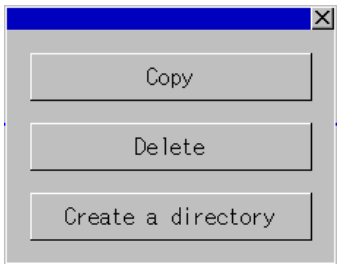
2) Touch the Copy button. ⇨ The function name is changed to "Copy".
The upper area is bounded by the heavy line.



- Changing to the "Delete" function

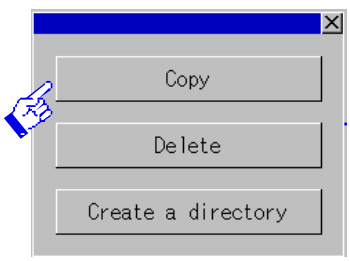
1) Touch the Function button. ⇨ The function selection window appears.





(For GT15)

2) Touch the Delete button. ⇨ The function name is changed to "Delete".



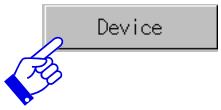
The display in the lower area is cleared as the following image.

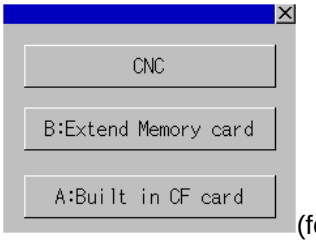
6.6 Selecting the Device

Select either CNC or Built-in CF card to perform copy or delete.

(Example) Selecting CNC

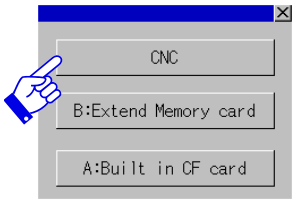
1) Touch the Device button. ⇨ The device selection window appears.





(for GT15)

2) Touch the CNC button. ⇨ The device is changed to "CNC".



The following table shows the displays that differ between devices.

	CNC is selected	Memory card is selected
Device name	CNC	E: USB drive (GT16 only) B: Extend memory card/ A: Built-in CF card
Directory	Machining program	Root directory
File name	Blank	
List	Displays from the top. Files are deselected.	

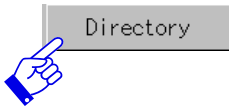
(Note) When CNC is selected for copy, a memory card is selected for the other device.
When a memory card is selected, CNC appears for the other device.

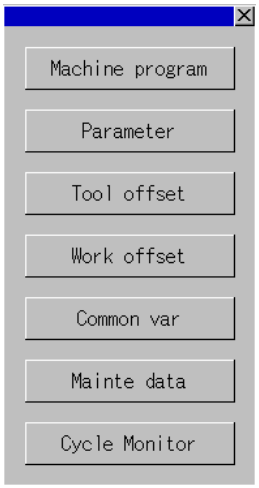
6.7 Selecting the Directory

Press the Directory button to select CNC data type when CNC is selected.
When a memory card is selected, this button is grayed and unavailable.

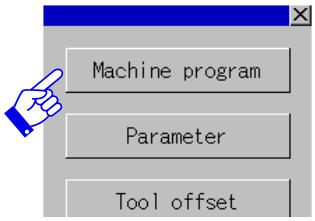
(Example) Selecting a directory of machining programs in CNC unit

1) Touch the Directory button. ⇨ The CNC data selection window appears.





2) Touch the Machining program button. ⇨ A directory of machining programs appears. The list is refreshed and shows files in the selected directory.



For details of CNC directories, see "6.16.2 CNC Data".

6.8 Selecting the Directory and a File

Operations behave differently for the different devices and areas selected.

[Operations of items selected in the list]

	Device (CNC)	Device (card)
Upper area	Name of the selected machining program is displayed in the file name area.	Name of the selected directory is displayed in the directory area. The directory appears. Name of the selected file is displayed in the file name area. The file name is highlighted in the list. Then the directory in lower area shows the directory corresponding to the type of the selected file.
Lower area	The selected machining program is highlighted just for a second, and then the selection is cancelled.	Name of the selected directory is displayed in the directory area. The directory appears. The selected file name is highlighted just for a second, and then the selection is cancelled.

A directory name is displayed in brackets "< >". Press <..> to move to the directory one above.
Items in the list are displayed in the order as follows.

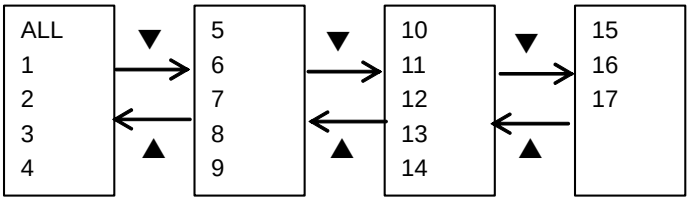
[Sort in the list]

Device	Order
CNC (machining programs)	In ascending order of "O" Nos.
CNC (other than machining programs)	In ASCII order
Memory card	1. The directory one above; <..> 2. Directories in ASCII order 3. "O_.PRG" files in ascending order of "O" No. 4. Files in ASCII order

(Note) "ASCII order" compares each character of the file names.
1' (code 0x31) comes above 'A' (code 0x41) in ascending order.

(Example) Sorting items in CF card
<..> → <1> → <SAMPLE01> → 1.PRG → 2.PRG → 10.PRG → 5 → AB → AB.TXT → ABCD

(Note 1) Scroll buttons (▲ and ▼) scroll one page when pressed once. The page is not changed when ▲ is pressed on the first page or ▼ is pressed on the last page.



[Page scroll (for CNC machining program)]

(Note 2) Files in the list refreshed by selecting a device or by changing directories are deselected.

(Note 3) If pages are scrolled with any file selected, the selected file comes on a top after the scroll. Files are deselected if any folder comes at the top after pages have been scrolled.

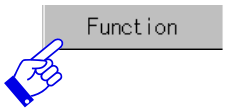
(Note 4) No file is selected after the scroll with no file selected.

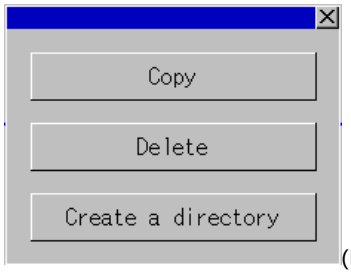
6.9 Copying a File

6.9.1 Copying any File other than the SRAM.BIN file

(Example) Copying \sample01\123.PRG from the extend memory card to CNC

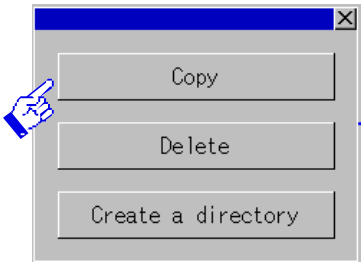
1) Touch the Function button. ⇨ The function selection window appears.



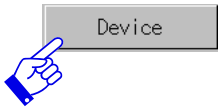


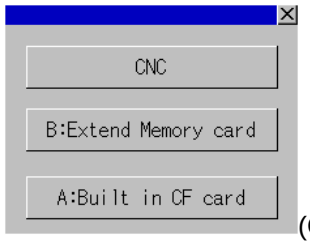
(For GT15)

2) Touch the Copy button. ⇨ The function name is changed to "Copy". The upper area is bounded by heavy line.



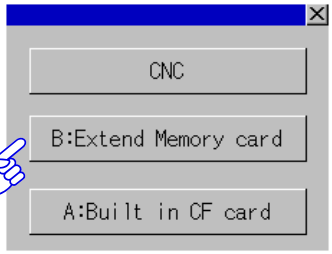
3) Touch the Device button in the upper area. ⇨ The device selection window appears.

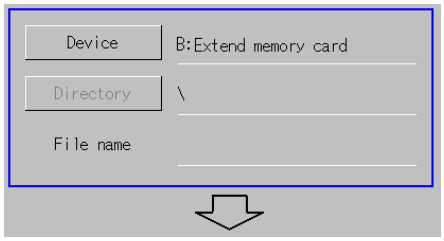




(GT15 only)

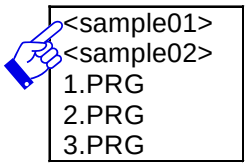
4) Touch the B: Extend memory card button. ⇨ The device is changed to extend memory card. The directory area shows the route.



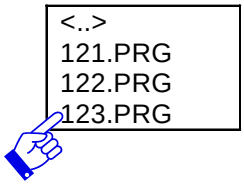


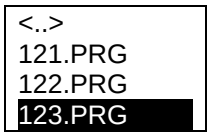
The list displays the route of the extend memory card.

5) Touch "<sample01>" in the list. ⇨ Displays files in the sample01 directory. The directory area shows "\sample01\".

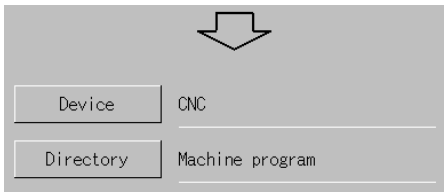


6) Touch "123.PRG". ⇨ "123.PRG" is highlighted.
The file name area shows "123.PRG".

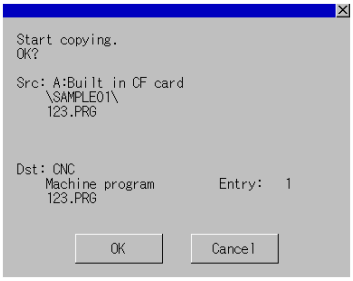




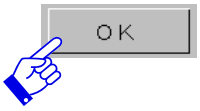
The directory in the lower area shows "Machine program".



7) Touch the Exec button. ⇨ The copy confirmation window appears.



8) Touch the OK button. ⇨ The message area shows "Copying".
"Copy complete" is shown upon the completion of copy.



(Note 1) If the OK button is touched in the copy confirmation window while the same file exists in the copy destination, the overwrite confirmation window appears. Press the OK button to overwrite. Press the Cancel button to close the overwrite confirmation window without any execution.

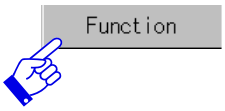
(Note 2) When a machining program file to be copied contains multiple programs, the overwrite confirmation window (for the multiple machining programs) appears and shows the duplicated machining programs.

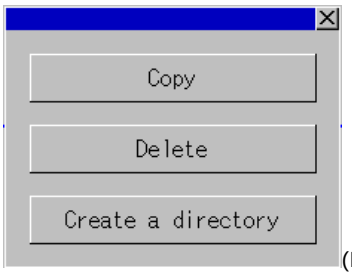
(Note 3) When the data protection key 3, edit lock B and C are enabled, a file containing multiple programs (such as ALL.PRG) cannot be copied to CNC nor PC.

6.10 Deleting Files

(Example) Deleting "3.PRG" in the route of built-in CF card

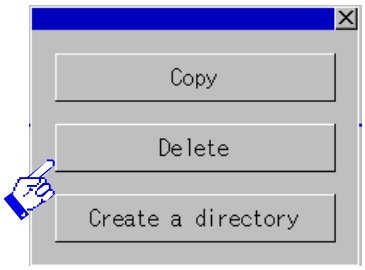
1) Touch the Function button. ⇨ The function selection window appears.



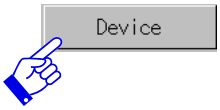


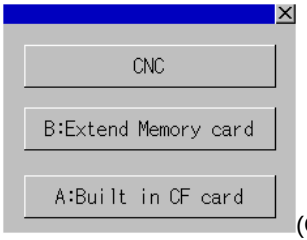
(For GT15)

2) Touch the Delete button. ⇨ The function name is changed to "Delete".



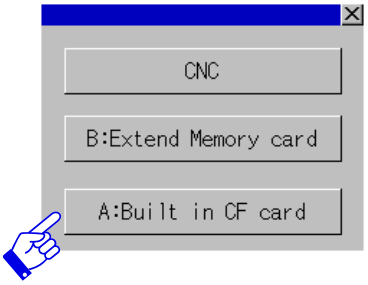
3) Touch the Device button in the upper area. ⇨ The device selection window appears.





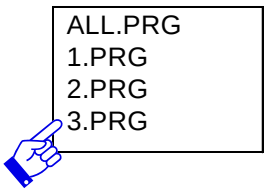
(GT15 only)

4) Touch A: Built in CF card. ⇨ The device is changed to Built-in CF card. The list shows the route of the Built-in CF card.

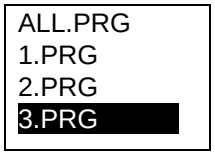


ALL.PRG
1.PRG
2.PRG
3.PRG

5) Select "3.PRG" in the list.



⇨ "3.PRG" is highlighted. The file name area shows "3.PRG".



6) Touch the Exec button.

⇨ The delete confirmation window appears.

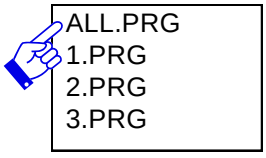
7) Touch the OK button.

⇨ The message area shows "Deleting". "Delete complete" is shown upon the completion of delete.

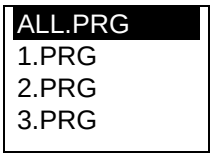
- Deleting all the machining programs in CNC

  After selecting "CNC" at the step 4, follow the steps as shown below to delete.
  .

5) Select "ALL.PRG" in the list.

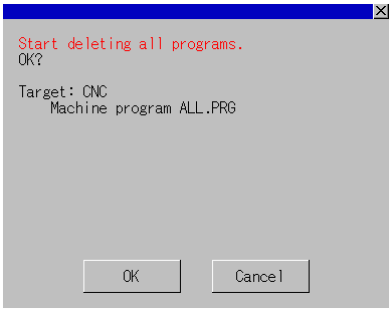


→ "ALL.PRG" is highlighted. The file name area shows "ALL.PRG".

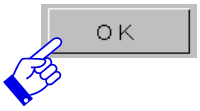


6) Touch the Exec button.

→ The delete confirmation window appears.



7) Touch the OK button.



→ The message area shows "Deleting". "Delete complete" is shown upon the completion of delete.

(Note 1) When the data protection key 3, edit lock B and C are valid, the all deletion of machining programs cannot be executed with ALL.PRG.

(Note 2) Cannot execute all files' deletion on a memory card.

6.11 Refreshing the List

Refresh the list currently displayed.

(Example) Refreshing "sample01" directory in the built-in CF card

1) Execute steps in "6.9.1 Copying any File other than the SRAM.BIN file" from 1) to 6)

⇨ "123.PRG" is highlighted.
The file name area shows "123.PRG".

<..>
121.PRG
122.PRG
123.PRG

2) Touch the Refresh button.

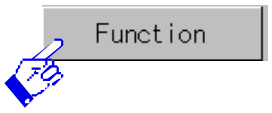
⇨ The display of "sample01" directory is refreshed.
The highlight on "123.PRG" is cancelled.

<..>
121.PRG
122.PRG
123.PRG

(Note 1) Files in the list are deselected.
(Note 2) When no directory exists after the replacement of cards or any other operations, the root is displayed instead.
(Note 3) A top page is displayed in the list.

6.12 Stop the USB Drive

1) Touch the [Function] button.

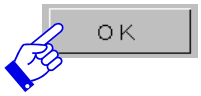


➡ The function selection window appears.

2) Touch the [USB Drive Stop] button.

➡ The USB drive stop confirmation window appears.

3) Touch the [OK] button.



➡ The confirmation window will be closed.
The message "USB drive has been stopped" will be displayed.

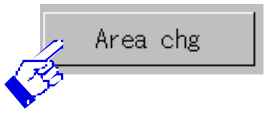
4) Remove the USB drive.

6.13 Creating a Directory

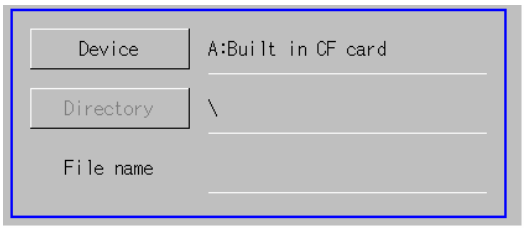
Create a directory in a memory card.

Example: Create a directory "SAMPLE01" in the "A: Built-in CF card" root.

1) Touch the "Area Chg" button to move the blue frame to the memory card side.

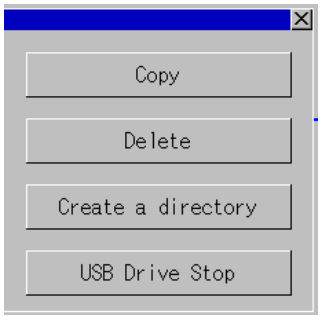


➡ The blue frame will move to the memory card side.



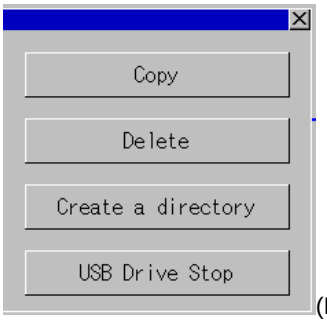
2) Touch the [Function] button.

➡ The function selection window appears.



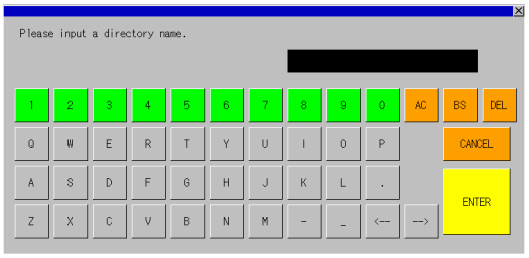
(For GT16)

3) Touch the [Create a directory] button.



(For GT16)

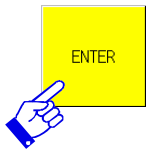
➡ The key window appears.



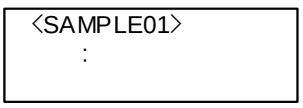
4) Enter "SAMPLE01" by touching the buttons one by one.

➡ The input value display area shows "SAMPLE01".

5) Touch the [ENTER] button.



➡ The key window will be closed.
The list will be refreshed and the "SAMPLE01" directory will be created.



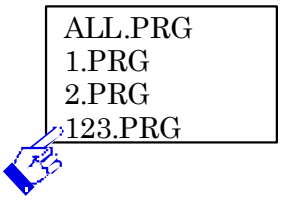
6.14 Changing a File Name when Outputting the File

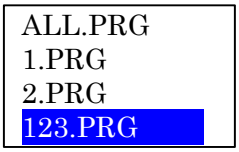
Example: Change the name of machining program for CNC "123.PRG" to "SAMPLE.TXT", and then output it to the root directory of "A:Built-in CF card".

1) Select "CNC" for device in the upper area, "A: Built-in CF card" in the lower area.

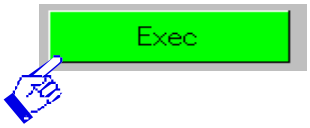
2) Select "Machining program" for directory in the upper area. ➡ CNC machining programs are displayed in the list.

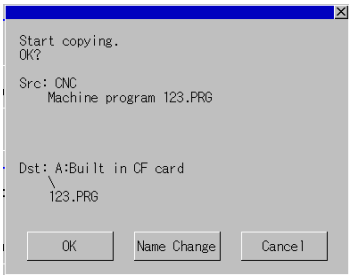
3) Touch "123.PRG". ➡ "123.PRG" is highlighted. The file name area shows "123.PRG".



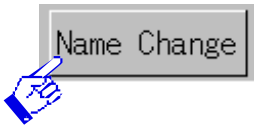


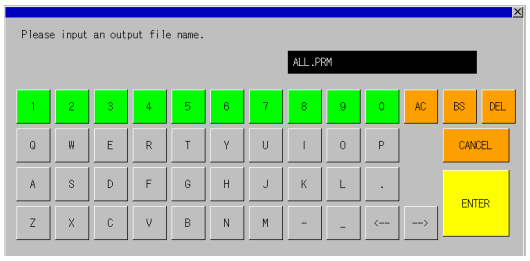
4) Touch the [Exec] button. ➡ The copy confirmation window appears.





5) Touch the [Name Change] button. ➡ The key window appears.

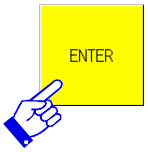




6) Touch the [AC] button. ➡ The list is cleared.

7) Enter "SAMPLE.TXT" by pressing the buttons one by one. ➡ The input value display area shows "SAMPLE.TXT".

8) Touch the [ENTER] button. ➡ The key window will be closed and the copy confirmation window will be displayed again. Then the file name of the copy destination is changed to "SAMPLE.TXT".



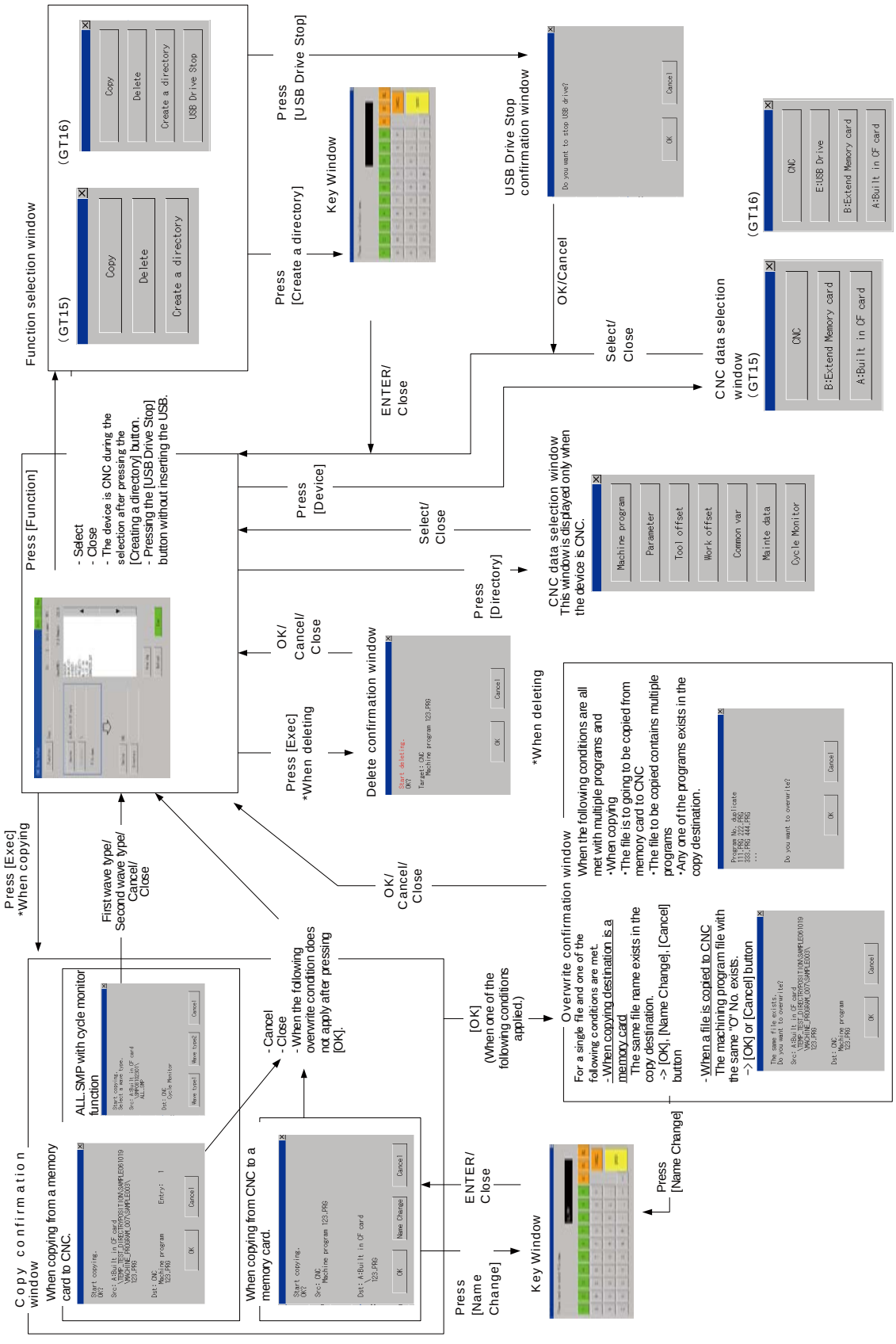
9) Touch the [OK] button. ➡ Start copying.
The contents of "ALL.PRG" will be copied as "SAMPLE.TXT" in the root directory of "A: Built-in CF card".
The message "Copy complete" will be displayed.

(Note) There is restriction as follows on changing the file name, depending on how the file is recognized when input into CNC.

How the file is recognized when input into CNC	Restriction on changing the file name
Format in the file	There is no restriction.
File name	The file name can be changed to "* (file name in the CNC)*.(file extention)" style. ("*" represents an arbitrary alphabet or number) Unless the file name is changed as above, the file will not be recognized and will not be input into CNC again. [Example: Restriction of changing the file name "USERPLC.LAD"] - File names that can be input to CNC USERPLC.LAD USERPLC070408.LAD _USERPLC.LAD BKUP_USERPLC1.LAD - File names that is not recognized by CNC USERPLC.LAD1 <- File extension is different.
File extension	Set the same file extension as that in the file name in CNC. Besides a file extention name, there is no restriction.

6.15 Summary of Screen Transition from the "CNC Data In/Out" Screen

Screen transition from "CNC Data In/Out" screen is as follows.



6.16 Various Status

6.16.1 Data Protection

Data protection key 1 to 3, edit lock B and C prevent the data from being input/output.
Data cannot be copied from/to CNC, nor deleted in CNC. (Files on the memory card are not deleted.)

Parameters and keys	Data protected
Data protection key 1 (KEY1)	Tool offsets and workpiece offsets
Data protection key 2 (KEY2)	Parameters and common variables, CNC ladders, C register data, R register data and T register data
Data protection key 3 (KEY3)	Machining program A ALL.PRG 1.PRG to 7999.PRG, 10000.PRG to 99999999.PRG, File with multiple programs
Edit lock B (#8105) (The value "1" prohibits input/output and delete)	Machining program B 8000.PRG to 8999.PRG (User-prepared standard subprogram)
Edit lock C (#1121) (The value "1" prohibits input/output and delete)	Machining program C 9000.PRG to 9999.PRG (Machine manufacturer customized program)

(Note) Machining program files to be protected are recognized with the file name that has been entered to CNC. When SAMPLE.PRG in a memory card is copied to CNC, for example, the "O" No. in the file will be used to determine whether the data protection is applied or not.

6.16.2 CNC Data

The following table shows the directories and files selected or displayed on the CNC data selection window.

Directory name	Details	File name in the CNC	How the file is recognized when input into CNC	INPUT	OUTPUT
Machining programs	Machining programs	ALL.PRG, .PRG following 0 (Note 1) (Note 2)	Format in the file	○	○
Parameters	Parameters	ALL.PRM	Format in the file	○	○
Tool offset	Tool offset data	TOOL.OFS	Format in the file	○	○
Work offset	Workpiece offset data	WORK.OFS	Format in the file	○	○
Common variables	Common variables	COMMON.VAR	Format in the file	○	○
Maintenance data (Note 3)	CNC Ladder	USERPLC.LAD	File name	○	○
	R resister data	RREG.REG	File name	×	○
	C resister data	CREG.REG	File name	×	○
	Operation history data	TRACE.TRC	File name	×	○
	T resister data	TREG.REG	File name	×	○
	CNC sampling data	NCSAMP.CSV	File name	×	○
	SRAM data files	SRAM.BIN	File name	×	○
Cycle monitor	Batch files related to sampling (Note 4)	ALL.SMP	File name	○	○
	Sampling setting file	SAMPLING.PRM	File name	○	○
	PLC data collection setting file	PLCSAMP.CTF	File extension	○	○
	PLC drawing setting file	PLCSAMP.MMG	File extension	○	○
	PLC data collection file	PLCSAMPH.CSV	File name	×	○

(Note 1) If the base specifications parameter "#1166 fixpro" is set to "1", the fixed cycle programs will be the target.
(Note 2) When CNC has more than one machining programs, "ALL. PRG" is shown at the top of the list.
(Note 3) The CNC ladder or register data file may not exist. In this case, the ladder with no step will be the CNC ladder. When the data file of any register does not exist, the value of the register is set to "0"
(Note 4) Outputting "ALL.SMP" from CNC to a memory card outputs "SAMPLING.PRM", "PLCSAMP.CTF", "PLCSAMP.MMG", and "PLCSAMPH.CSV" in batch. Inputting "ALL.SMP" from a memory card to CNC inputs SAMPLING.PRM and PLCSAMP.CTF in batch. PLCSAMP.MMG and PLCSAMPH.CSV will not be input but will be included in the SAMPLING.PRM.
(Note 5) "Can not write file" message will be displayed when attempting to enter a file that cannot be entered.
(Note 6) When the "File name" is selected for "file identification method", not only the file names listed above but also the names in "*(file name in CNC)*.(extension name)" format are selected. "*" indicates any alphanumeric character.

[File names that are set as maintenance data file "USERPLC.LAD" in CNC]
USERPLC.LAD
USERPLC070408.LAD <- A part of the file name is contained.
_USERPLC.LAD
BKUP_USERPLC1.LAD

[File names that are not selected for the maintenance data file "USERPLC.LAD"]
USERPLC.LAD1 <- The file has different extension name.

Arbitrary file names and extensions of the files for machining programs, parameters, tool offset data and common variables are recognized, because the file type is determined by the format in the files. The file whose type is uncertain will be treated as machining program.

(Note 7) There is a relationship between a PLC collection setting file (PCSAMP.CFT) and a PLC drawing setting file (PLCSAMP.MMG). Keep in mind the following two points when entering CNC data
- Enter both files with the combination when the sampling was executed and displayed.
- Enter a PLC collection setting file first, and then a PLC drawing setting file.

If not following these points, a sampled waveform may not be displayed. To display the sampled waveform, move to the cycle monitor drawing setting screen and allocate a device.

6.16.3 Messages

The following tables show the messages displayed.

[Messages displayed for communication errors]

Message	Details
Communication error	Communication with CNC failed. Check the connection and settings. This message appears when a communication error has occurred after setting a network driver in the communication setting window or pressing the CNC Chg button. Only this error message appears in red.

[Messages displayed for status]

Message	Details
Connecting	The line is being connected with CNC. Wait for a while. "■" will be blinking during the execution.
Connect complete	The line has been connected with CNC.
Copying	The data is being copied. The file name to be copied will be displayed under the message. When the file is successfully copied, the file name will be no longer displayed. "■" will be blinking during the execution.
Copy complete	The data has been copied.
Deleting	The data is being deleted. "■" will be blinking during the execution.
Delete complete	The data has been deleted.
Wait	Processing is executed. Wait for a while. During the execution, "■" blinks.
Directory has been created	Directory has been created.
USB drive has been stopped	USB drive has failed to stop. USB drive can be removed.

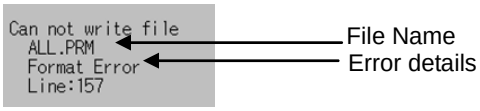
[Displayed messages (for errors)]

Message	Error details
Unable to exec : PLC built in CNC running	Stop PLC running in the CNC and try again.
USB drive has been stopped.	USB drive has failed to stop. USB drive cannot be removed.
Unable to exec : Program running	Automatic operation is being executed. Stop the operation and try again.
Out of memory	Memory capacity was exceeded.Ensure sufficient free space by deleting machining programs and try again.
The same name exists.	Attempting to create a directory is failed as the same name exists. Input a different name.
In sampling	Cannot input or output as it is in sampling. Stop sampling and try again.
File not found	The specified file does not exist in the copy source.Check the settings and try again.
Timeout	Communication with CNC failed. Check the connection.
Can't create a directory	A directory cannot be created. One of the followings is the reason. - The device is set to "CNC". - Cannot write in a memory card. - When outputting the batch file related to the cycle monitor, a serial number 99 exists (exmaple: "SAMP09012399" directory exists for January 23rd, 2009) so that a directory cannot be created. Transit to the different directory or delete the directory.
Directory illegal	The specified directory is illegal.Check the settings and try again. Check the restrictions about directory name and file name on "3.17 Restrictions".
Data protect	The data is protected.Check the functions such as data protection key, edit lock and read only, and then try again.
No. of registration over	The number of registered programs was exceeded. Delete unnecessary machining programs and try again.
Pass is too long	Pass is too long. Check the directory/file name.
File is not specified	[Execute] is pressed without any file specified. Specify a file and try again.
Some error found in file system	File system error occurred. Format the NC memory.

Message	Error details
Can not write file	Could not copy the file to its destination. Remove the status that does not allow the data to be written. This error also appears when copying a input protected file from memory card to CNC. When a parameter file is input into CNC and this message is displayed, the error details are also displayed. (Note)
Can not read out file	Could not read the file in its source.Remove the status (such as read protect) that does not allow the data to be read.
Filename illegal	The file name is illegal. Try again. Check the restrictions about directory name and file name on "3.17 Restrictions".
Memory card does not exist	Card does not exist. Check whether memory card is inserted. Check the access switch.
Error	Any other error occurred. Turn off the power supply, check the data in the memory card and connection status, then try again.

(Note) About error detail
When a parameter file is input into CNC and "Can not write file" error occurs, the error details are displayed.
A line number of memory card is displayed with "Line", and a parameter number is displayed with "#".

[Example]



[File name]

File name	Name
ALL.PRM	File for all parameters
SAMPLING.PRM	Cycle monitor parameter file
PLCSAMP.CTF	Cycle monitor data collection setting file (*)
PLCSAMP.MMG	Cycle monitor drawing setting file (*)

(*) Parameter number will not be displayed. Only line number is displayed.

[Error details]

Message	Error details
Format Error	Format is incorrect.
Range Over	Value is incorrect.
In sampling	Data cannot write as it is in sampling. In this situation, a parameter number and a line number will not be displayed.
Superposition waveform cannot be set.	"Superposition" parameter for cycle monitor is set to "NO" when inputting the second waveform or it is judged that the superposition is not possible compared with the parameter of the first waveform. Check the parameter settings, or input data.

[Messages displayed for confirmation windows]

Message	**Details**
Start copying.	Displayed in the copy confirmation window.
Start deleting.	Displayed in the delete confirmation window.
Start deleting all programs.	Displayed in the delete confirmation window when ALL.PRG is selected.
OK?	Displayed in the copy confirmation window and the delete confirmation window.
The same file exists. Do you want to overwrite?	Displayed in the overwrite confirmation window (for a single file).
Restart the CNC.	Displayed in the message window after SRAM.BIN has been copied.
Program No. duplicate Do you want to overwrite?	Displayed in the overwrite confirmation window (for multiple machining programs).
Do you want to stop USB drive?	USB drive enters the stopped state.

6.16.4 Parameters

The following table shows the parameters related to data protection.

No.	Items	
#1135	unt_nm	(Unit name)
#1166	fixpro	(Fixed cycle editing)
#1121	edlk_c	(Edit lock C)
#8105	Edit lock B	

Refer to the Setup Manual for the details of the parameters.

6.16.5 Signals

The following table shows the signals related to data protection.

Abbrev.	Name
KEY1	Data protection key 1
KEY2	Data protection key 2
KEY3	Data protection key 3

Refer to the PLC Interface Manual for the details of the parameters.

6.17 Restrictions

(1) Following directory names or file names cannot be used.
- Character string including Kanji, Hiragana, Katakana, 1-byte Katakana.
- COM1 TO COM9, LPT1 TO LPT9, AUX, CON, NUL, PRN, CLOCK$
- Directory name beginning with G1
- Directory or file name beginning with "." (period) or "/"
- Directory or file name ending with "." (period) or "/"
- Directory or file name is "." (period) or ".." (two periods)

(2) Number of characters that can be used for full path name and full path name is up to 78. Up to 75 can be used when the file exists in root directory.

(3) There are 20 layers directory in directory hierarchy.

7. Data Backup and Restoration

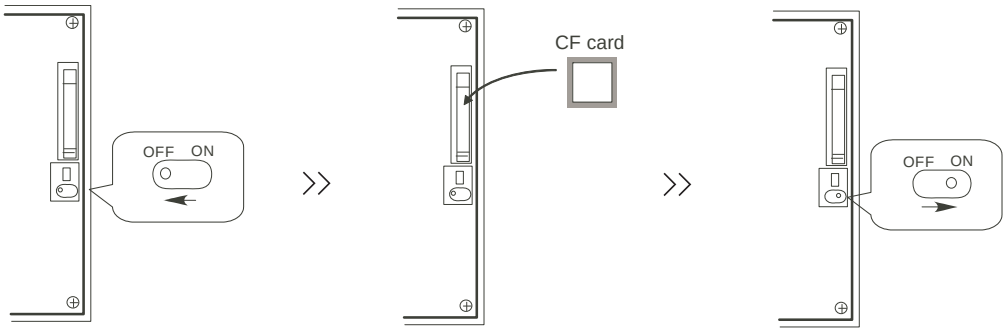
7.1 GOT Data Backup and Reinstallation

Standard functions (standard monitor OS), communication driver, BootOS and projects in GOT can be backed up in batch to the CF card inserted in GOT.
The data backed up in batch can be reinstalled to GOT in easy operation.

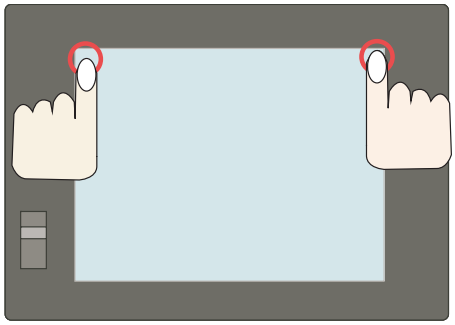
(Note 1) See below for the procedure when OS data and project data (drawing data) are stored in "C Drive". If OS data and project data (drawing data) are installed in "A Drive (CF card)", backup data from the CF card.
(Note 2) When using GOT1000 series GT16, USB memory is also available for backup and reinstallation.

7.1.1 Backup procedures

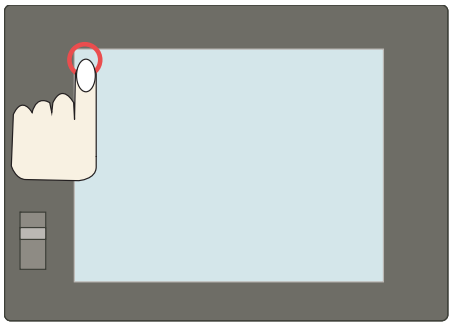
(1) Set "OFF" the CF Card access switch.
(2) Insert the CF Card into the card slot.
(3) Set "ON" the CF Card access switch.



(4) Open the GOT Utility screen.
For GT15: Press the right and left top corners together.

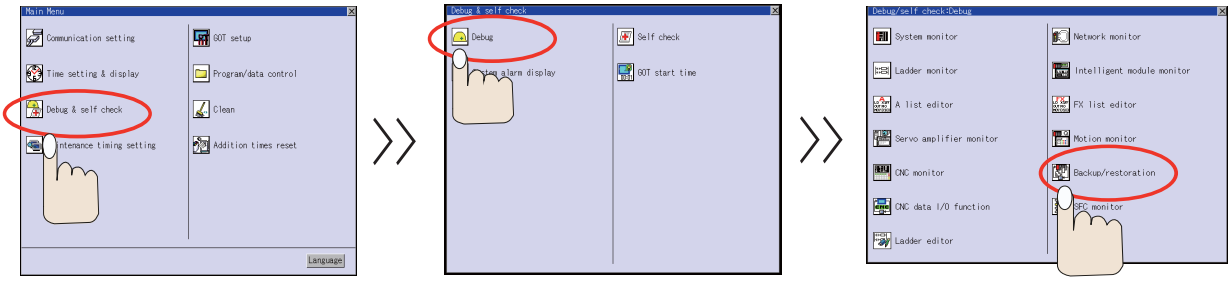


For GT16: Press the left top corner.

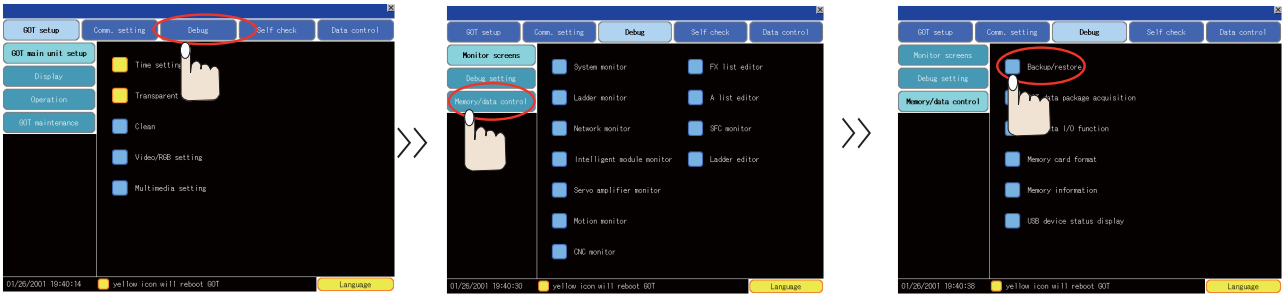


(5) Select the [Backup / Restoration] menu.

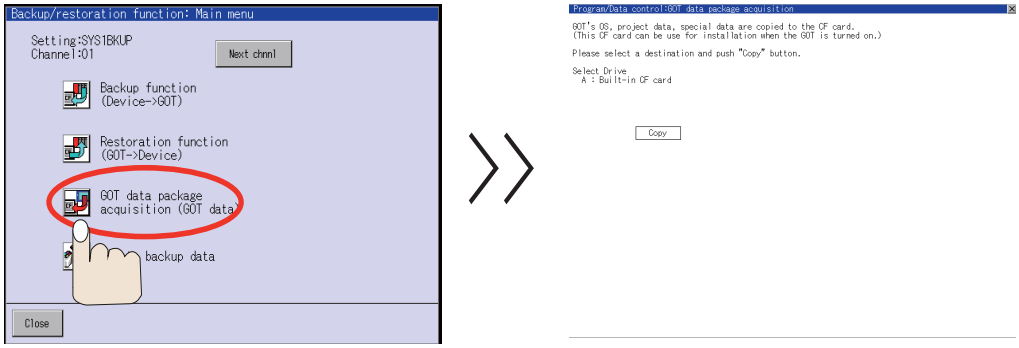
For GT15: [Debug & self check] - [Debug] - [Backup / Restoration]



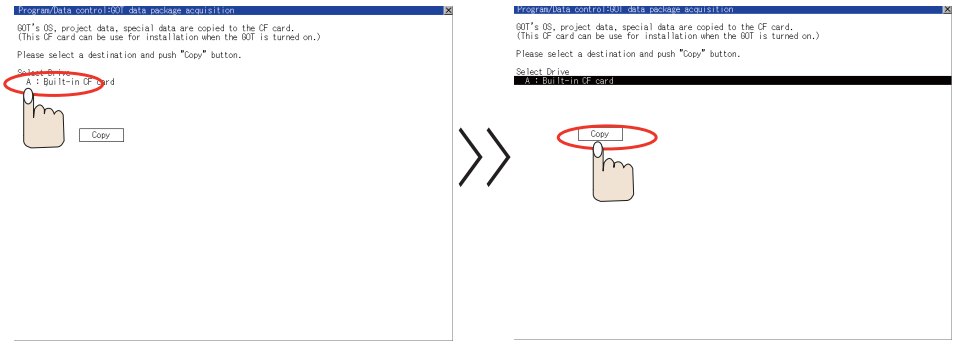
For GT16: [Debug] - [Memory / Data control] - [Backup / Restoration function]



(6) Select [Program/data control] -> [GOT data package acquisition] to display the [Program/Data control: GOT data package acquisition] screen.



(7) Select "A: Built-in CF card". Then select "Copy".



(Note) To backup the data to USB memory (GT16 only), select "E: USB Drive".

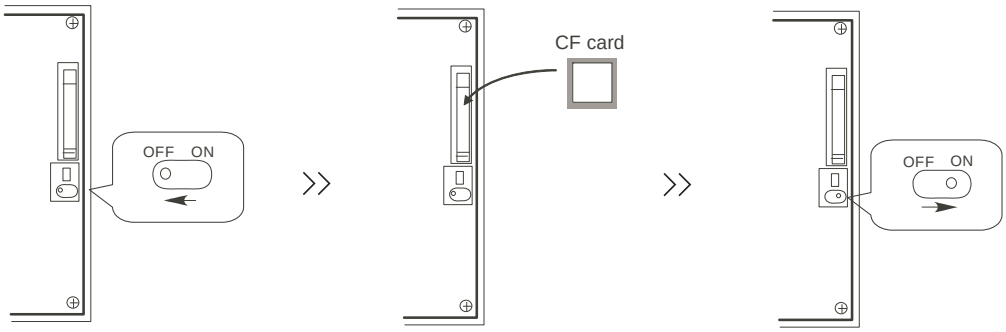
(8) The confirmation window will appear. Select "OK".

(9) When the data has been backed up, confirmation window will appear. Select "OK" to complete.

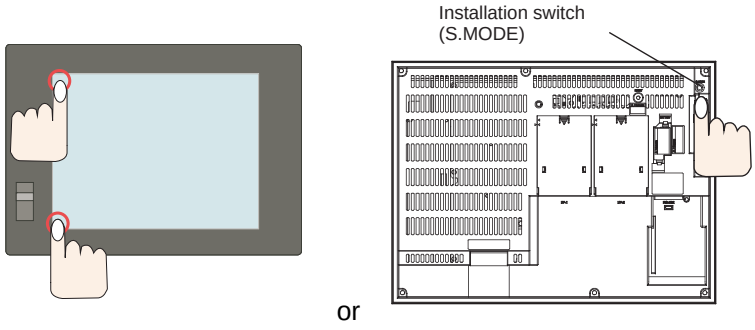
(Note) For details, refer to "GT15 User's Manual"(SH(NA)-080528ENG), Section 13 or "GT16 User's Manual"(SH(NA)-080778ENG), Section 12.

7.1.2 Reinstallation Procedures

(1) Turn the GOT OFF and set "OFF" the CF card access switch on the back of GOT.
Insert the CF card, which contains the data such as standard functions (standard monitor OS), in the slot. Then set "ON" the CF card access switch.



(2) Turn the GOT power ON. Keep touching the GOT screen's left top and bottom during turning ON.
When using GT1595-X or GT16, keep pressing the installation switch (S.MODE switch) on the back of the GOT during turning the GOT ON.



Boot OS and the standard monitor OS are installed to built-in flash memory.

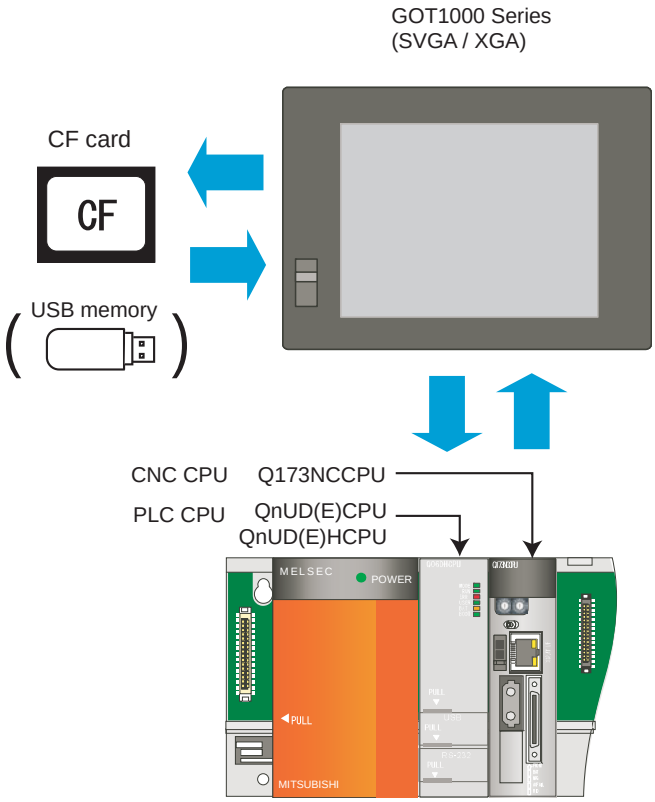
(3) GOT will automatically restart after the completion of installation. (If the standard monitor OS has already been installed, press the button to restart.)

(4) Confirm that the GOT is successfully restarted, and set the GOT's CF card access switch to "OFF". Confirm that the CF card access LED is OFF, and remove the CF card from the GOT's CF card interface.

(Note) For details, refer to the "GT Designer2 Version2 Basic Operation/Data Transfer Manual" (SH(NA)-080529), Section 8.

7.2 PLC/CNC CPU Data Backup and Restoration

Backup / Restoration function is installed on GOT1000 series.
All of PLC/CNC CPU data that needs to backup is saved into the CF card on GOT1000 in an easy operation.
All backup data can be restored to each CPU module one by one.
USB memory(Max.2GB) can be used on GOT1000 series GT16.



(Note 1) Backup/restoration is not executed on latch device of PLC CPU. If the data requires to be restored when CPU is replaced, assign the data to file registers.

(Note 2) Editing CNC data
The backed up CNC data includes machining programs, parameters, and others. Those data are related one another.
Therefore, when any of the data is changed with a text editor and others, the GOT cannot restore the backed-up CNC data to the CNC.
When editing the setting data, use the CNC data I/O function.
(For the CNC data I/O function, refer to "6. CNC Data Input/Output".)

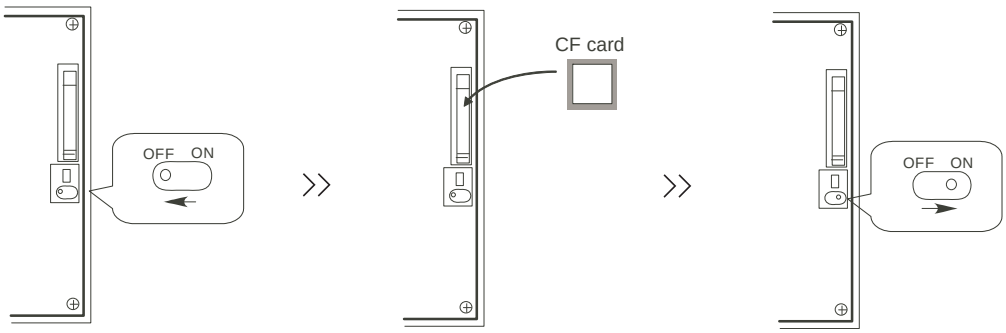
(Note 4) Refer to "Setting the saving destination drive for backup data" for destination drive for backup data.

7.2.1 Data Backup

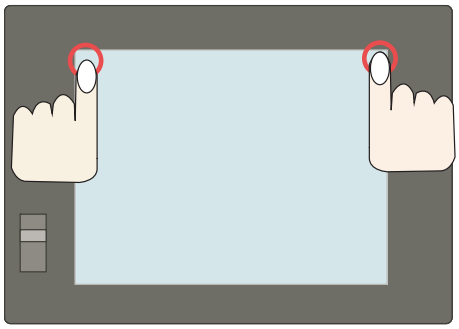
7.2.1.1 Backup procedures

The following shows the backup procedures.

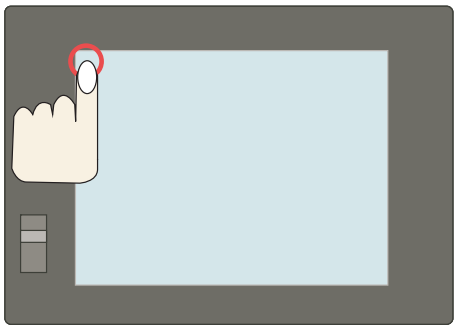
(1) Set "OFF" the CF Card access switch.
(2) Insert the CF Card into the card slot.
(3) Set "ON" the CF Card access switch.



(4) Open the GOT Utility screen. The utility call key(s) is set as follows at factory shipment.
For GT15: Press the right and left top corners together.



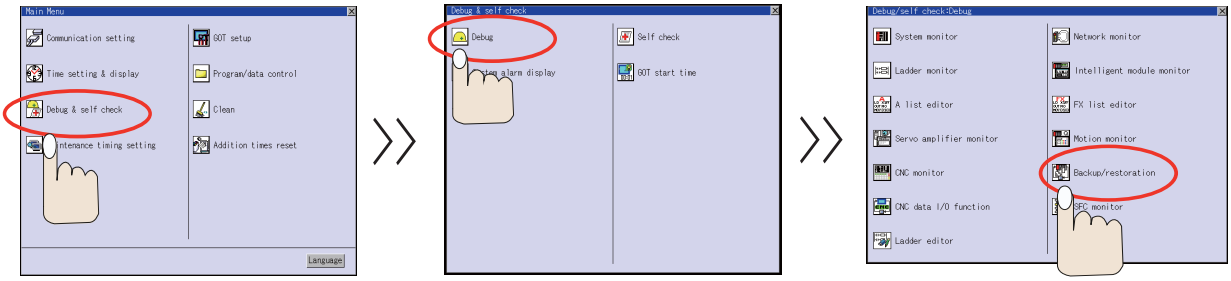
For GT16: Press the left top corner.



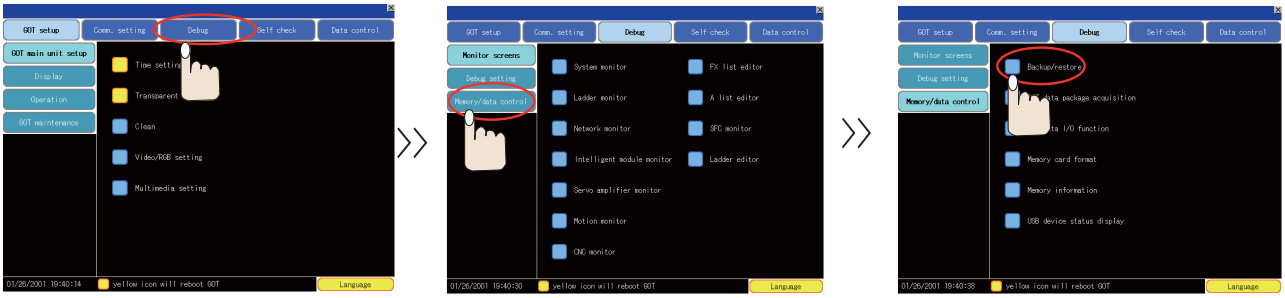
(Note) The utility call key(s) can be arranged on the GT Designer2's "GOT Setup" screen. For details, refer to the "GT Designer2 Version2 Screen Design Manual" (SH(NA)-080530), Section 3.

(5) Select the [Backup / Restoration] menu.

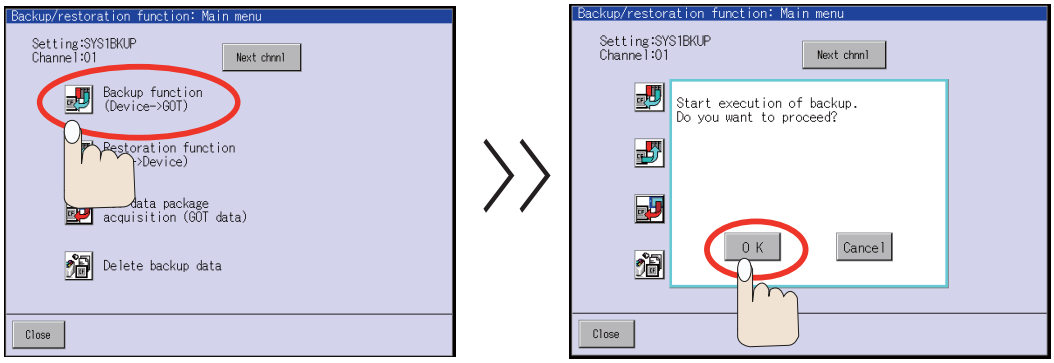
For GT15: [Debug & self check] - [Debug] - [Backup / Restoration]



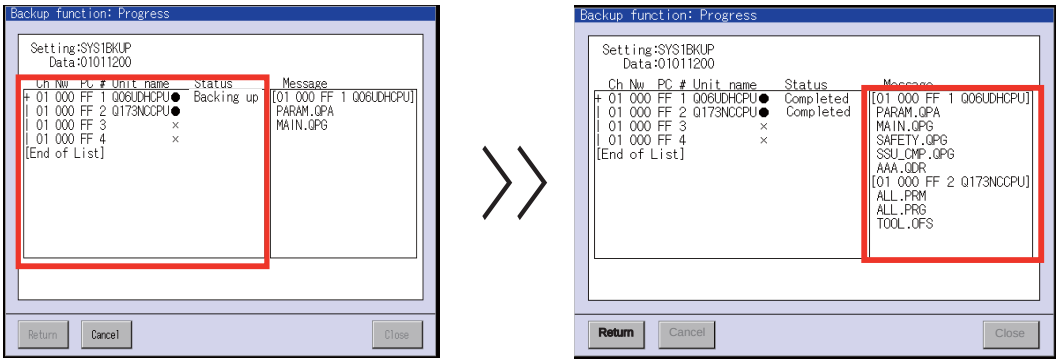
For GT16: [Debug] - [Memory / Data control] - [Backup / Restoration function]



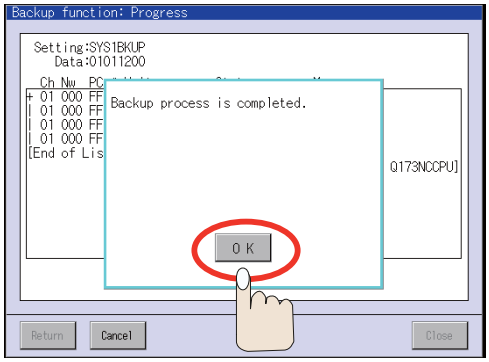
(6) Select [Backup function (Device->GOT)]. Then the confirmation window appears.
Click "OK" to start the backup.



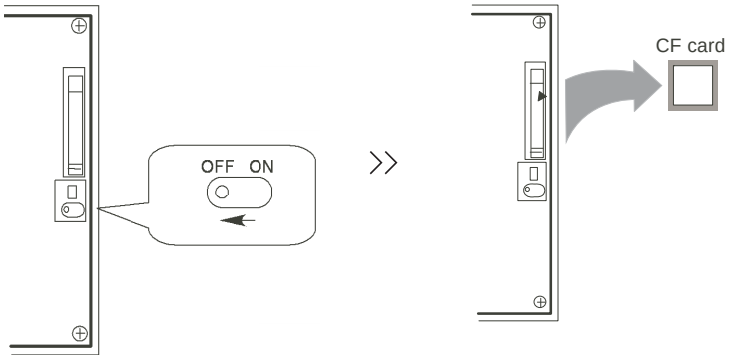
(7) CPU modules are selected and displayed automatically.
Backup files are made. The file names are displayed automatically.



(8) The completion window appears. Select "OK".

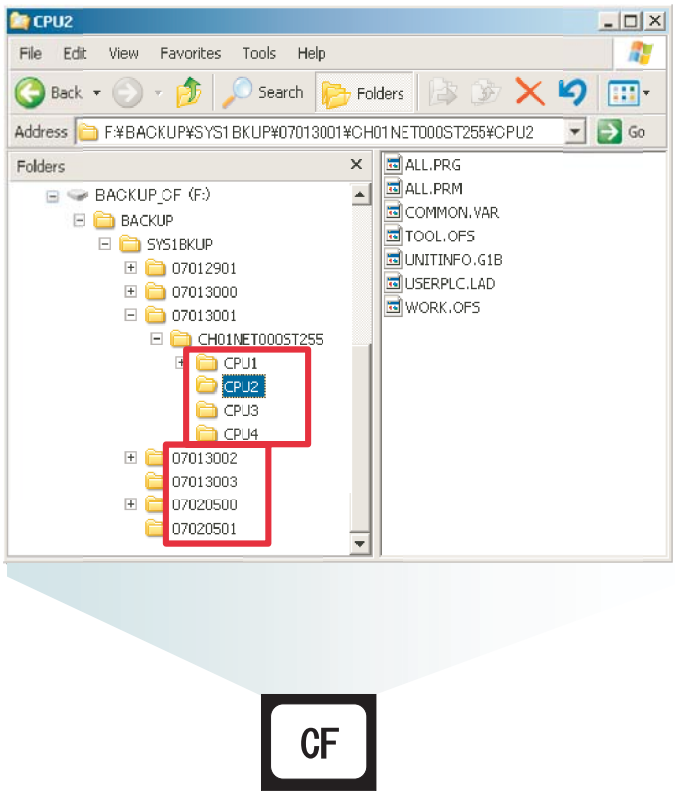


(9) Set "OFF" the Compact Flash card access switch, and then take out the card.



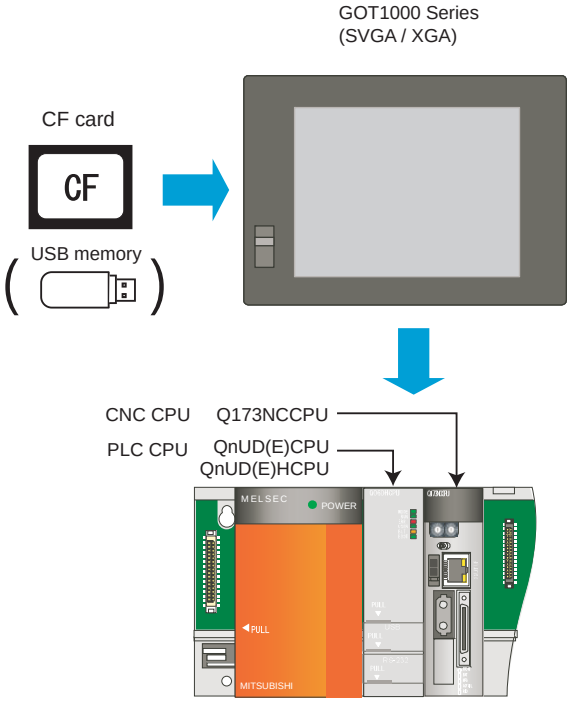
7.2.1.2 Backup Files Structure

Each backup makes a folder in a CF card, and data is saved.
Each folder name contains the backup date and order No.
And, sub folders are made for each CPU module.



7.2.2 Data Restoration

Backup / Restoration function is installed on GOT1000 series.
The backup data can be restored to each CPU module one by one in easy operation.
USB memory can be used on GOT1000 series GT16.



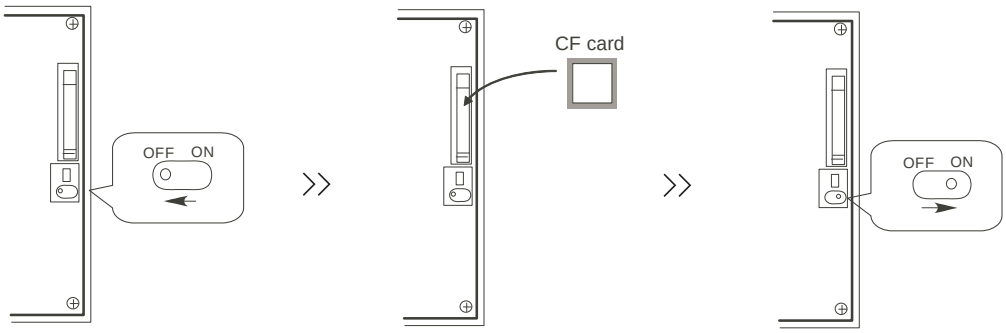
To exchange the CNC CPU unit, initialize the internal data. Refer to "C70 Setup Manual" for how to initialize.

(Note) The zero point of rotary axis will disappear after replacing the CNC CPU unit. Repeat the zero point initialization.

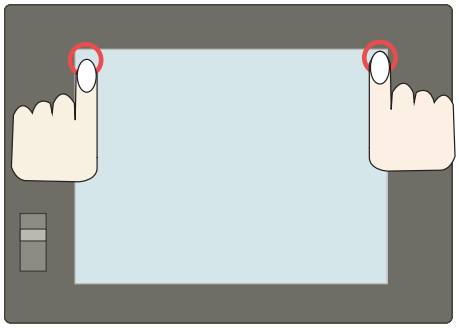
7.2.2.1 Restoration procedures

The following shows the restoration procedures.

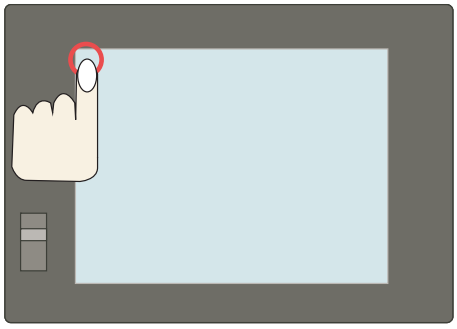
(1) Set "OFF" the CF Card access switch.
(2) Insert the CF Card that has backup data into the card slot.
(3) Set "ON" the CF Card access switch.



(4) Open the GOT Utility screen. The utility call key(s) is set as follows at factory shipment.
For GT15: Press the right and left top corners together.



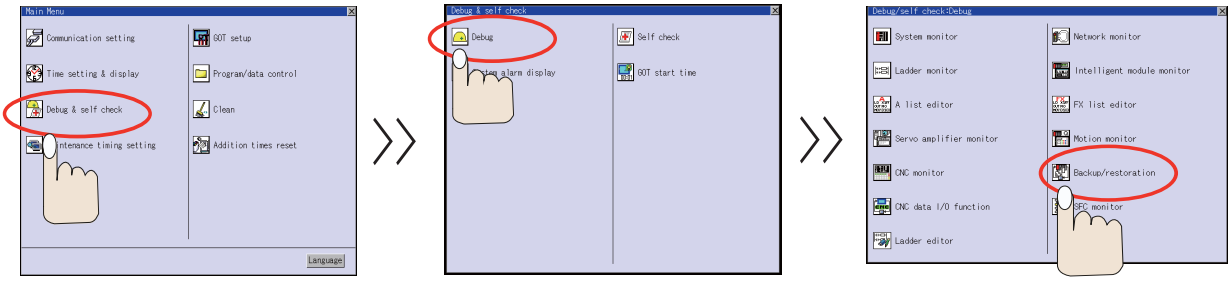
For GT16: Press the left top corner.



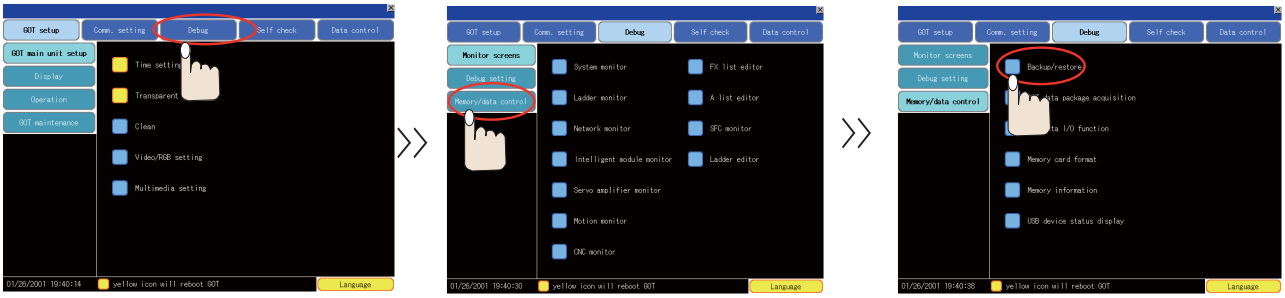
(Note) The utility call key(s) can be arranged on the GT Designer2's "GOT Setup" screen. For details, refer to the "GT Designer2 Version2 Screen Design Manual" (SH(NA)-080530), Section 3.

(5) Select the [Backup / Restoration] menu.

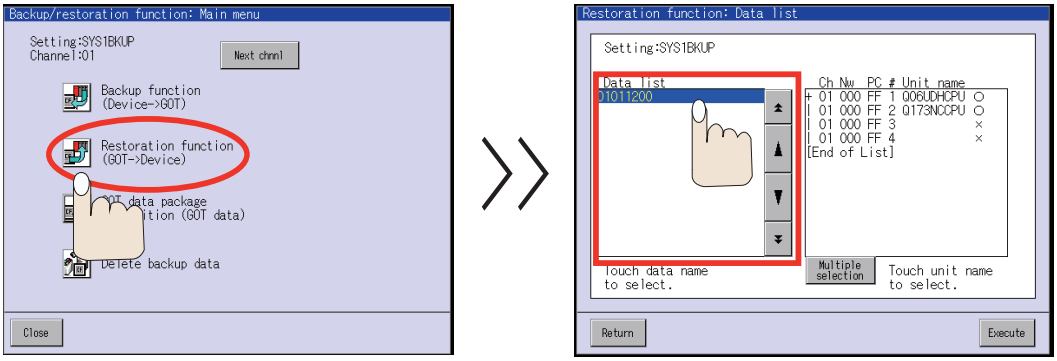
For GT15: [Debug & self check] - [Debug] - [Backup / Restoration]



For GT16: [Debug] - [Memory / Data control] - [Backup / Restoration function]

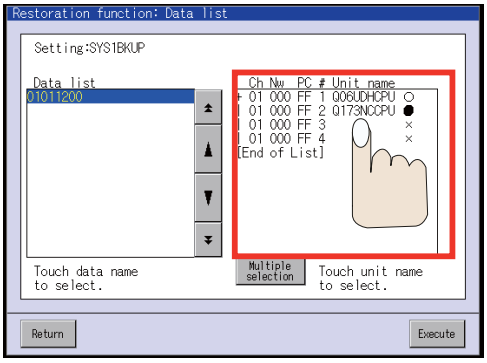


(6) When [Restoration function(Device->GOT)] is selected, backup files in the CF card are displayed in the "Data list" screen. Select the files for the restoration.

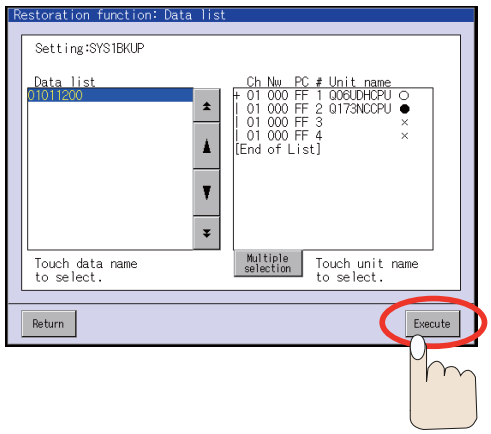


(Note) Data name displayed in the "Date list" screen contains backup date and the number of backup.
Example: Backup data on 2009/04/05 is displayed as follow.
09040500 (1st backup)
09040501 (2nd backup)
Refer to "Backup Files Structure".

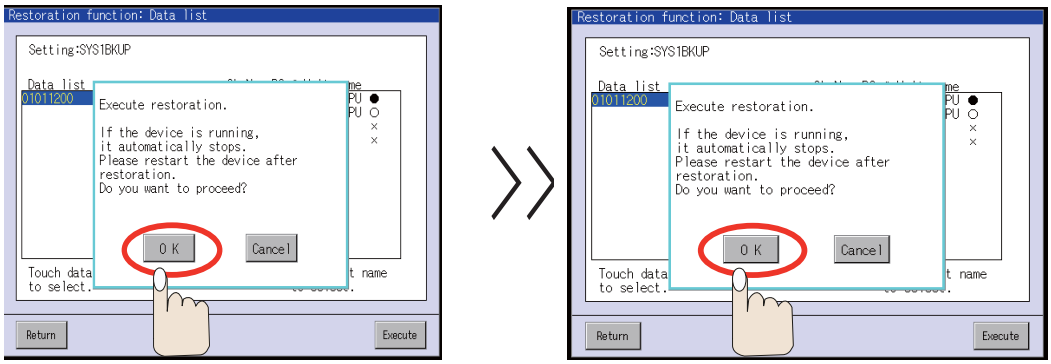
(7) CPU modules are also displayed. Select the CPU modules to be restored.
Selected modules are shown with "?".



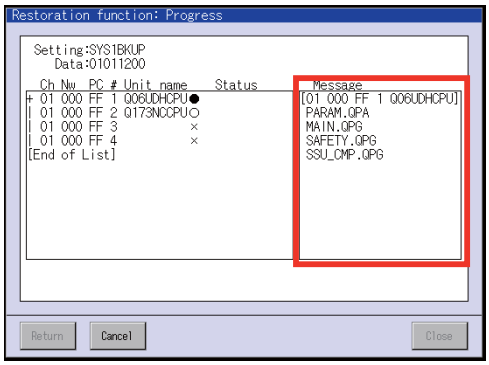
(8) Select "Execute".



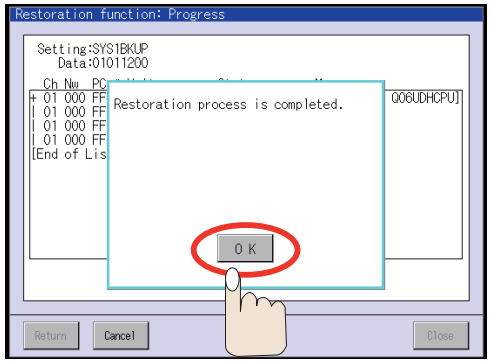
(9) The confirmation window appears. Click "OK" to start the restoration.



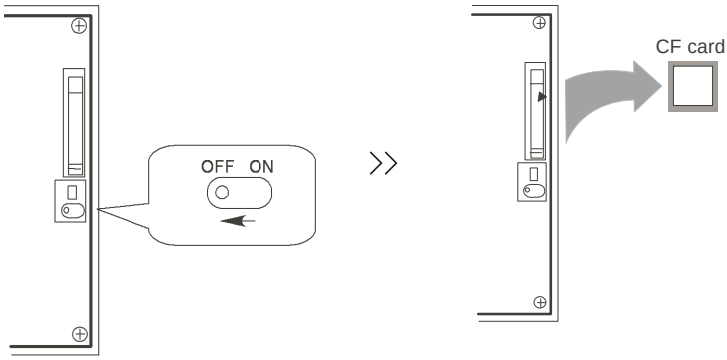
(10) The “Progress” screen appears and displays the restored files.



(11) The completion window appears after all the selected data have been restored. Select "OK". When "OK" is selected, all the CPU modules are reset automatically.



(12) Set "OFF" the Compact Flash card access switch, and then take out the card.



IV. APPENDIXES

Appendix 1. Registering and Editing Fixed Cycle Programs

The fixed cycle subprogram can be input, output and edited.

⚠ **CAUTION**

⃠ Do not change the fixed cycle program without prior consent from the machine maker.

Appendix 1.1 Parameter for Fixed Cycle Operation

The fixed cycle subprogram data is input, output and edited with the DATA IN/OUT screen and EDIT screen in the same manner as the normal user-created machining program. However, a parameter must be set beforehand. Set parameter "1166 fixpro" on the Basic Specification Parameter screen to "1" before starting inputting, outputting and editing operations. When this parameter is valid, the DATA IN/OUT screen and EDIT screen will become dedicated for the fixed cycle control subprogram. Only the fixed cycle programs will appear in the program list. Thus, when done working with the fixed cycle program, set this parameter to "0".

(Note) Parameter "#1166 fixpro" will be set to 0 when the power is turned OFF.

Appendix 1.2 Inputting a Fixed Cycle Program

The fixed cycle program is input on the DATA IN/OUT screen. Confirm that the parameter "#1166 fixpro" for fixed cycle operations is valid.
The operation methods are the same as the user machining programs.
It is handier to input the data in succession.
After registering, confirm that the program has been correctly registered in the program list and EDIT screen.

Appendix 1.3 Outputting a Fixed Cycle Program

The fixed cycle program is output on the DATA IN/OUT screen. Confirm that the parameter "#1166 fixpro" for fixed cycle operations is valid.
The operation methods are the same as the user machining programs. The registered fixed cycle program can be output one by one, or all programs output as a batch.
To output all programs in a batch, or to input the programs, all programs can be input in succession with one operation.
After outputting, always compare the programs to confirm that there are no data output mistakes.

Appendix 1.4 Deleting a Fixed Cycle Program

The fixed cycle program is deleted on the DATA IN/OUT screen. Confirm that the parameter "#1166 fixpro" for fixed cycle operations is valid.
The operation methods are the same as the user machining programs.

Appendix 1.5 Standard Fixed Cycle Subprogram

(1) Lathe specifications

G37	Automatic tool length measurement
O370	AUTO-TLM

```
G31Z#5F#3
IF[ROUND[ABS[#2-[##10*#11-#12]]]GT#8]GOTO
1
IF[ROUND[##10*#11-#12]EQ#4]GOTO1
##9=##10-#12/#11-#2/#11+##9
#3003=#1
N2
M99
N1#3901=126
```

G74	End face cutoff pattern cycle
O740	FACE-PATC

```
G.1
IF[ABS[#2]GT0]GOTO10
#14=1
N10#13=#3
IF[#15NE0]GOTO11
#13=#3-#5
N11#16=0
DO1
#10=0
#11=#4
DO2
#10=#10+#4
IF[ABS[#10]GE[ABS[#1]]]GOTO1
G01X#11
G00X#6
#11=#4-#6
END2
N1G01X#1-#10+#11
IF[#15EQ0]GOTO20
IF[#16EQ0]GOTO21
N20G00Y#5
N21#16=1
G00X-#1
IF[#14]GOTO3
#12=#12+#3
IF[ABS[#12]LT[ABS[#2]]]GOTO2
#14=1
#13=#2-#12+#13
N2G00Y#13
#13=#3-#5
END1
N3G00Y-#2-#5
M99
```

G75	Straight cutting pattern cycle
O750	STRAIGHT-PATC

```
G.1
IF[ABS[#1]GT0]GOTO10
#14=1
N10#13=#4
IF[#15NE0]GOTO11
#13=#4-#5
N11#16=0
DO1
#10=0
#11=#3
DO2
#10=#10+#3
IF[ABS[#10]GE[ABS[#2]]]GOTO1
G01Y#11
G00Y#6
#11=#3-#6
END2
N1G01Y#2-#10+#11
IF[#15EQ0]GOTO20
IF[#16EQ0]GOTO21
N20G00X#5
N21#16=1
G00Y-#2
IF[#14]GOTO3
#12=#12+#4
IF[ABS[#12]LT[ABS[#1]]]GOTO2
#14=1
#13=#1-#12+#13
N2G00X#13
#13=#4-#5
END1
N3G00X-#1-#5
M99
```

G76	Compound thread cutting cycle
O760	THREAD-PATC

```
G.1
#12=1
#13=#9
IF[ABS[#13]GE[ABS[#8]]]GOTO1
#16=1
#13=#8
N1#11=#13
IF[ABS[#11]LT[ABS[#4-#5]]]GOTO2
#11=#4-#5
#14=1
N2#17=#11
#10=ROUND[[#11+#5]*#7]
IF[[#10XOR#1]GE0]GOTO20
#10=-#10
N20G00X#10
#20=#10
DO1
#15=ROUND[#10*#3/#1]
N90#40=90#41=#5001#42=#5002
G00Y#2+#3-#4-#15+#11
G33X#1-#10Y-#3+#15M96.101P1000D3
G00Y-#2+#4-#11M97.101
IF[#14GT0]GOTO3
IF[#16GT0]GOTO7
#12=#12+1
#13=ROUND[#9*SQRT[#12]]
IF[ABS[#13-#11]GE[ABS[#8]]]GOTO8
#16=1
N7#13=#11+#8
N8#11=#13
IF[ABS[#11]LT[ABS[#4-#5]]]GOTO9
#11=#4-#5
#14=1
N9#10=ROUND[[#17-#11]*#7]
IF[[#10XOR#1]GE0]GOTO6
#10=-#10
N6#10=#10+#20
G00X-#1+#10
N12END1
N3IF[ABS[#6]LT1]GOTO5
#14=0
#13=0
DO2
IF[#14GT0]GOTO5
#13=#13+#6
IF[ABS[#13]LT[ABS[#5]]]GOTO4
#13=#5
#14=1
N4G00X#10-#1
N91#40=91#41=#5001#42=#5002
G00Y#2+#3-#4+#13-#15+#11
G33X#1-#10Y-#3+#15M96.101P1000D3
G00Y-#2+#4-#13-#11M97.101
END2
N5G00X-#1
M99
```

G76.1	2-part system simultaneous compound thread cutting cycle
O761	THREAD-PATC-2SYS.

```
G.1
N761!L10
#12=1
#13=#9
IF[ABS[#13]GE[ABS[#8]]]GOTO1
#16=1
#13=#8
N1#11=#13
IF[ABS[#11]LT[ABS[#4-#5]]]GOTO2
#11=#4-#5
#14=1
N2#17=#11
#18=ROUND[[#4-#11-#5]*#7]
IF[[#18XOR#1]GE0]GOTO10
#18=-#18
N10#19=#18
#10=ROUND[[#11+#5]*#7]
IF[[#10XOR#1]GE0]GOTO20
#10=-#10
N20G00X#10
#20=#10
DO1
#15=ROUND[#10*#3/#1]
N90#40=90#41=#5001#42=#5002
G00Y#2+#3-#4-#15+#11
!L11
G33X#1-#10-#18Y-#3+#15M96.101P1000D3
G00Y-#2+#4-#11M97.101
!L12
IF[#14GT0]GOTO3
IF[#16GT0]GOTO7
#12=#12+1
#13=ROUND[#9*SQRT[#12]]
IF[ABS[#13-#11]GE[ABS[#8]]]GOTO8
#16=1
N7#13=#11+#8
N8#11=#13
IF[ABS[#11]LT[ABS[#4-#5]]]GOTO9
#11=#4-#5
#14=1
N9#10=ROUND[[#17-#11]*#7]
IF[[#10XOR#1]GE0]GOTO6
#10=-#10
N6#10=#10+#20
G00X-#1+#10+#18
IF[#14LT0]GOTO11
#18=0
GOTO12
N11#18=#19-#10+#20
N12END1
N3IF[ABS[#6]LT1]GOTO5
#14=0
#13=0
DO2
IF[#14GT0]GOTO5
#13=#13+#6
IF[ABS[#13]LT[ABS[#5]]]GOTO4
#13=#5
#14=1
N4G00X#10-#1
N91#40=91#41=#5001#42=#5002
G00Y#2+#3-#4+#13-#15+#11
!L11
G33X#1-#10Y-#3+#15M96.101P1000D3
G00Y-#2+#4-#13-#11M97.101
!L12
END2
N5G00X-#1
M99
```

G76.2	2-part system simultaneous compound thread cutting cycle
O762	THREAD-PATC-2SYS.

```
G.1
N762!L10
#12=1
#13=#9
IF[ABS[#13]GE[ABS[#8]]]GOTO1
#16=1
#13=#8
N1#11=#13
IF[ABS[#11]LT[ABS[#4-#5]]]GOTO2
#11=#4-#5
#14=1
N2#17=#11
#18=ROUND[[#4-#11-#5]*#7]
IF[[#18XOR#1]GE0]GOTO10
#18=-#18
N10#19=#18
#10=ROUND[[#11+#5]*#7]
IF[[#10XOR#1]GE0]GOTO20
#10=-#10
N20IF[#27NE1]GOTO21
G00X#10
N21#20=#10
#28=1
DO1
#15=ROUND[#10*#3/#1]
#29=#28MOD2
IF[[#27EQ1]AND[#29EQ0]]GOTO22
IF[[#27EQ2]AND[#29EQ1]]GOTO22
N90#40=90#41=#5001#42=#5002
G00Y#2+#3-#4-#15+#11
!L11
G33X#1-#10-#18Y-#3+#15M96.101P1000D3
G00Y-#2+#4-#11M97.101
#21=#18
!L12
N22IF[#14GT0]GOTO3
IF[#16GT0]GOTO7
#12=#12+1
#13=ROUND[#9*SQRT[#12]]
IF[ABS[#13-#11]GE[ABS[#8]]]GOTO8
#16=1
N7#13=#11+#8
N8#11=#13
IF[ABS[#11]LT[ABS[#4-#5]]]GOTO9
#11=#4-#5
#14=1
N9#10=ROUND[[#17-#11]*#7]
IF[[#10XOR#1]GE0]GOTO6
#10=-#10
N6#10=#10+#20
IF[[#27EQ1]AND[#29EQ1]]GOTO24
IF[[#27EQ2]AND[#29EQ0]]GOTO24
IF[[#27EQ2]AND[#28EQ1]]GOTO23
G00X-#1+#10+#21
GOTO24
N23G00X#10
N24IF[#14LT1]GOTO11
#18=0
GOTO12
N11#18=#19-#10+#20
N12#28=#28+1
END1
N3IF[ABS[#6]LT1]GOTO5
#14=0
#13=0
DO2
IF[#14GT0]GOTO5
#13=#13+#6
IF[ABS[#13]LT[ABS[#5]]]GOTO4
#13=#5
#14=1
N4#29=#28MOD2
IF[[#27EQ1]AND[#29EQ1]]GOTO25
IF[[#27EQ2]AND[#29EQ0]]GOTO25
G00X#10-#1+#21
#21=0
N91#40=91#41=#5001#42=#5002
G00Y#2+#3-#4+#13-#15+#11
!L11
G33X#1-#10Y-#3+#15M96.101P1000D3
G00Y-#2+#4-#13-#11M97.101
!L12
N25#28=#28+1
END2
N5G00X-#1
M99
```

G77	Straight cutting cycle
O770	STRAIGHT-TURNING

```
G.1
IF[[#1EQ0]OR[#2EQ0]]GOTO1
Y#2+#7
G1X#1Y-#7
Y-#2
G0X-#1
N1M99
```

G78	Thread cutting cycle
O780	THREAD-CUTTING

```
G.1
IF[[#1EQ0]OR[#2EQ0]]GOTO1
N90#40=90#41=#5001#42=#5002
Y#2+#7
G33X#1Y-#7F#9E#10M96.101P1000D3
G0Y-#2M97.101
X-#1
N1M99
```

G79	End face cutoff cycle
O790	FACE-CUTTING

```
G.1
IF[[#1EQ0]OR[#2EQ0]]GOTO1
X#1+#7
G1X-#7Y#2
X-#1
G0Y-#2
N1M99
```

G83 G87	Deep hole drilling cycle B
O830	DRILL-CYCLE-B

```
G.1
IF[#30]GOTO2
M#24
#29=#11#28=0
Z#2
#2=##5#3003=#8OR1
DO1
#28=#28-#11#26=-#28-#29
Z#26
IF[ABS[#28]GE[ABS[#3]]]GOTO1
G1Z#29
G0Z#28
#29=#11+#14
END1
N1G1Z#3-#26
G4P#4
#3003=#8
G0Z-#3-#2
IF[#24EQ#0]GOTO2
M#24+1
G4P#21
N2M99
```

G83.1 G87.1	Deep hole drilling cycle A
O831	DRILL-CYCLE-A

```
G.1
IF[#30]GOTO2
M#24
#29=0#28=#11
Z#2
#2=##5#3003=#8OR1
DO1
#29=#29+#11
IF[ABS[#29]GE[ABS[#3]]]GOTO1
G1Z#28
G0Z-#14
#28=#11+#14
END1
N1G1Z#3-#29+#28
G4P#4
#3003=#8
G0Z-#3-#2
IF[#24EQ#0]GOTO2
M#24+1
G4P#21
N2M99
```

G83.2	Deep hole drilling cycle 2
O832	DEEP-DRILL-CYCLE-2

```
G.1
IF[#30]GOTO3
#3003=#8OR1
#29=#12#28=0#26=0
G0Z#2
IF[#12NE#0]GOTO1
IF[#11EQ#0]GOTO2
N1#28=#28-#12#26=-#28-#29
IF[ABS[#28]GE[ABS[#3]]]GOTO2
G1Z#12
G4P#4
G0Z#28-#2
G4P#13
#29=#11+#15
DO1
#28=#28-#11#26=-#28-#29
G0Z#26+#2
IF[ABS[#28]GE[ABS[#3]]]GOTO2
G1Z#29
G4P#4
G0Z#28-#2
G4P#13
END1
N2G1Z#3-#26
G4P#4
#3003=#8
G0Z-#3-#2
N3M99
```

G84 G88	Tap cycle
O840	TAP-CYCLE

```
G.1
IF[#30]GOTO2
M#24
Z#2
#2=##5#3003=#8OR1#3004=#9OR3
G1Z#3
G4P#4
M#6
#3900=1
G1Z-#3
#3004=#9
M#7
#3003=#8
IF[#24EQ#0]GOTO1
M#24+1
G4P#21
N1G0Z-#2
N2M99
```

G85 G89	Boring cycle
O850	BORING-CYCLE

```
G.1
IF[#30]GOTO2
M#24
Z#2
#2=##5#3003=#8OR1
G1Z#3
G4P#4
#3003=#8
Z-#3F#23
F#22
IF[#24EQ#0]GOTO1
M#24+1
G4P#21
N1G0Z-#2
N2M99
```

(2) Machining center specifications

G12 G13	Circular cut
O120	CIRCLE-CUTTING

```
#42=#40-#41
G#39X#42Y0I#42/2
X0Y0I-#42
X-#42Y0I-#42/2
M99
```

G34	Bolt hole circle
O340	BOLT-HOLE-CIRCLE

```
#45=0
WHILE[#45LT[ABS[#43]]]DO1
#46=360*#45/#43+#42
#47=RND[#41*COS[#46]]#48=RND[#41*SIN[#4
6]]
X#39+#47Y#40+#48
#39=-#47#40=-#48#45=#45+1
END1
M99
```

G35	Line at angle
O350	LINE-AT-ANGLE

```
#45=0
WHILE[#45LT#43]DO1
#46=#45*#41
#47=RND[#46*COS[#42]]#48=RND[#46*SIN[#4
2]]
X#39+#47Y#40+#48
#39=-#47#40=-#48#45=#45+1
END1
M99
```

G36	Arc
O360	ARC

```
#45=0
WHILE[#45LT#43]DO1
#46=#45*#44+#42
#47=RND[#41*COS[#46]]#48=RND[#41*SIN[#4
6]]
X#39+#47Y#40+#48
#39=-#47#40=-#48#45=#45+1
END1
M99
```

G37	Automatic tool length measurement
O370	AUTO-TLM

```
G31Z#5F#3
IF[ROUND[ABS[#2-[##10*#11-#12]]]GT#8]GOT
O1
IF[ROUND[##10*#11-#12]EQ#4]GOTO1
##9=##10-#12/#11-#2/#11+##9
#3003=#1
N2
M99
N1#3901=126
```

G37.1	Grid
O379	GRID

```
#45=0#47=#39#48=#40
WHILE[#45LT#43]DO1
#46=1
X#47Y#48
WHILE[#46LT#44]DO2
X#41
#46=#46+1
END2
#47=0#48=#42#41=-#41#45=#45+1
END1
M99
```

G81	Drill, spot drill
O810	DRILL

```
G.1
IF[#30]GOTO1
Z#2G#6H#7
#2=##5#3003=#8OR1
G1Z#3
#3003=#8
G0Z-#3-#2,I#23
N1M99
```

G82	Drill, counter boring
O820	COUNTER-BORING

```
G.1
IF[#30]GOTO1
Z#2G#6H#7
#2=##5#3003=#8OR1
G1Z#3
G4P#4
#3003=#8
G0Z-#3-#2,I#23
N1M99
```

G83	Deep hole drill cycle
O830	DEEP-DRILL

```
G.1
IF[#30]GOTO2
#29=#11#28=0
Z#2G#6H#7
#2=##5#3003=#8OR1
DO1
#28=#28-#11#26=-#28-#29
Z#26
IF[ABS[#28]GE[ABS[#3]]]GOTO1
G1Z#29
G0Z#28
#29=#11+#14
END1
N1G1Z#3-#26
#3003=#8
G0Z-#3-#2,I#23
N2M99
```

G84	Tap cycle
O840	TAP-CYCLE

```
G.1
IF[#30]GOTO9
Z#2G#6H#7
#2=##5#3003=#8OR1#3004=#9OR3
IF[#11]GOTO1
GOTO2
N1
IF[#14]GOTO5
N2G1Z#3
GOTO7
N5
#29=0#28=#11
DO1
#29=#29+#11
IF[ABS[#29]GE[ABS[#3]]]GOTO6
G1Z#28
M4
G1Z-#14
M3
#28=#11+#14
END1
N6G1Z#3-#29+#28
N7G4P#4
M4
#3900=1
G1Z-#3
#3004=#9
G4P#4
M3
#3003=#8
G0Z-#2,I#23
N9M99
```

G85	Boring 1
O850	BORING-1

```
G.1
IF[#30]GOTO1
Z#2G#6H#7
#2=##5#3003=#8OR1
G1Z#3
#3003=#8
Z-#3
G0Z-#2,I#23
N1M99
```

G86	Boring 2
O860	BORING-2

```
G.1
IF[#30]GOTO1
Z#2G#6H#7
#2=##5#3003=#8OR1
G1Z#3
G4P#4
M5
G0Z-#3-#2
#3003=#8
M3
N1M99
```

G87	Back boring
O870	BACK-BORING

```
G.1
IF[#30]GOTO1
#3003=#8OR1
M19
X#12Y#13
#3003=#8
Z#2G#6H#7
#3003=#8OR1
G1X-#12Y-#13
#3003=#8
M3
#3003=#8OR1
Z#3
M19
G0X#12Y#13
Z-#2-#3
#3003=#8
X-#12Y-#13
M3
N1M99
```

G88	Boring 3
O880	BORING-3

```
G.1
IF[#30]GOTO1
Z#2G#6H#7
#2=##5#3003=#8OR1
G1Z#3
G4P#4
#3003=#8
M5
#3003=#8OR1
G0Z-#3-#2
#3003=#8
M3
N1M99
```

G89	Boring 4
O890	BORING-4

```
G.1
IF[#30]GOTO1
Z#2G#6H#7
#2=##5#3003=#8OR1
G1Z#3
G4P#4
#3003=#8
Z-#3
G0Z-#2,I#23
N1M99
```

G73	Step cycle
O831	STEP-CYCLE

```
G.1
IF[#30]GOTO2
#29=0#28=#11
Z#2G#6H#7
#2=##5#3003=#8OR1
DO1
#29=#29+#11
IF[ABS[#29]GE[ABS[#3]]]GOTO1
G1Z#28
G4P#4
G0Z-#14
#28=#11+#14
END1
N1G1Z#3-#29+#28
G4P#4
#3003=#8
G0Z-#3-#2,I#23
N2M99
```

G74	Reverse tap cycle
O841	COUNTER-TAP-CYCLE

```
G.1
IF[#30]GOTO9
Z#2G#6H#7
#2=##5#3003=#8OR1#3004=#9OR3
IF[#11]GOTO1
GOTO2
N1
IF[#14]GOTO5
N2G1Z#3
GOTO7
N5
#29=0#28=#11
DO1
#29=#29+#11
IF[ABS[#29]GE[ABS[#3]]]GOTO6
G1Z#28
M3
G1Z-#14
M4
#28=#11+#14
END1
N6G1Z#3-#29+#28
N7G4P#4
M3
#3900=1
G1Z-#3
#3004=#9
G4P#4
M4
#3003=#8
G0Z-#2,I#23
N9M99
```

G76	Fine boring
O861	FINE-BORING

```
G.1
IF[#30]GOTO1
Z#2G#6H#7
#2=##5#3003=#8OR1
G1Z#3
M19
X#12Y#13
G0Z-#3-#2
#3003=#8
X-#12Y-#13
M3
N1M99
```

Appendix 2. List of Function Codes

Function code ISO	Significance in control unit	Screen display	Setting and display unit key in	Save in memory	Function in control unit
0 to 9	Significant	Displayed	Key in enabled	Saved	Value data
A to Z	Significant	Displayed	Key in enabled	Saved	Address
+	Significant	Displayed	Key in enabled	Saved	Sign, variable operator (+)
–	Significant	Displayed	Key in enabled	Saved	Sign, variable operator (–)
·	Significant	Displayed	Key in enabled	Saved	Decimal point
,	Significant	Displayed	Key in enabled	Saved	
/	Significant	Displayed	Key in enabled	Saved	Block delete (optional block skip) variable operator (,)
%	Significant	Displayed (%)	Key in disabled (automatic insertion)	Saved	End of record
LF/NL	Significant	Displayed (;)	Key in enabled ;/EOB	Saved	End of block
(	Significant	Displayed	Key in enabled ;/EOB	Saved	Control out (comment start)
)	Significant	Displayed	Key in enabled ;/EOB	Saved	Control in (commend end)
:	Significant	Displayed	Key in disabled	Saved	Program No. address (Substitute for 0)
#	Significant	Displayed	Key in enabled	Saved	Variable No.
*	Significant	Displayed	Key in enabled	Saved	Variable operator (×)
=	Significant	Displayed	Key in enabled	Saved	Variable definition
[	Significant	Displayed	Key in enabled	Saved	Variable operator
]	Significant	Displayed	Key in enabled	Saved	Variable operator
BS	Insignificant	Blank	Key in disabled	Saved	
HT	Insignificant	Blank	Key in disabled	Saved	
SP	Insignificant	Blank	Key in enabled	Saved	
CR	Insignificant	Blank	Key in disabled	Saved	
DEL	Insignificant	Do not display	Key in disabled	Do not save	
NULL	Insignificant	Do not display	Key in disabled	Do not save	
Other than the above	Insignificant	(Note 2)	Key in disabled	Saved	

(Note 1) Codes not shown above (except comments) will cause an error during operation.
(Note 2) The internally saved characters (including blank) that correspond to the command codes are displayed. Note that @ is not displayed.

Appendix 3. List of Command Value Ranges

Item	Command value range
Minimum input setting unit	0.001
Maximum stroke (value on machine coordinate system)	±99999.999mm
Maximum command value	±99999.999mm
Rapid traverse rate	1 to 1000000mm/min
Cutting feed rate	1 to 1000000mm/min
2nd zero point offset (value on machine coordinate system)	±99999.999mm
Tool offset amount (tool dimension)	±999.999mm
Tool offset amount (wear)	±99.999mm
Handle feed amount	0.001mm/P
Soft limit range (value on machine coordinate system)	±99999.999mm
Dwell time	0 to 99999.999s
Backlash compensation amount	0 to ±511 pulses
Screw lead	0.0001 to 99.999999mm
Synchronous feed	0.001 to 99.999mm/rev

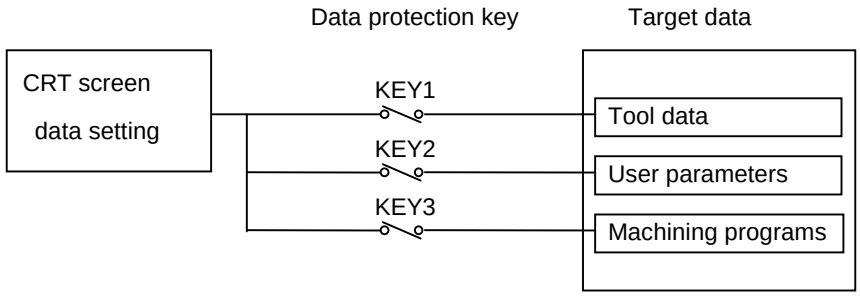
Appendix 4. Data Protection

Appendix 4.1 Data Protection Key

Data protection keys can inhibit data from being set or erased. There are three types of data protection keys as shown below (the key names depend on the machine manufacturers. For the details, refer to the manuals issued by individual machine manufacturers).

1) KEY 1: Protection of all tool data and coordinate system values preset by origin setting
2) KEY 2: Protection of user parameters and common variables
3) KEY 3: Protection of work programs

The data protection keys protect data when they are turned off.



(1) Protection of tool data (KEY 1)
When KEY 1 is off, the operation items listed in Table 1 are inhibited.

Table 1 Data protection by KEY 1

No.	Operation	Screen
1	Origin setting	MONITOR/POSITION
2	Setting/erasing of tool nose wear compensation	TOOL/COMP TOOL TIP OFFSET
3	Setting/erasing of tool length compensation	TOOL/COMP TOOL DATA
4	Setting/erasing of nose-R compensation, wear compensation and tool nose point	TOOL/COMP NOSE-R
5	Setting/erasing of tool offset data	TOOL/COMP TOOL OFFSET
6	Setting/erasing of work coordinate offset data	PARAM/WORK OFFSET

(Note) When key 1 is off:

- You can't set the origin by pressing the CAN C.B key on the POSITION/COORDINATE screen. Only the message "DATA PROTECT" will be displayed.
- No data is input by pressing any key other than the MENU key on the screens corresponding to numbers 2 to 6 in Table 1. Instead, it will display the message "DATA PROTECT". Manual numeric command also cannot be carried out on the TOOL screen.

(2) Protection of user parameters and common variables (KEY 2)
When KEY 2 is turned off, the operation items listed in Table 2 are inhibited.

Table 2 Data protection by KEY 2

No.	Operation	Screen
1	Machining parameter setting	PARAM/PROCESS
2	Control parameter on/off	PARAM/CONTROL
3	Axis parameter setting	PARAM/AXIS
4	Barrier data	PARAM/Barrier data
5	Common variable setting	MONITOR/COMMON VARIABLE

(Note) When KEY 2 is off, pressing any key other than the MENU key on the screens corresponding to numbers 1 to 5 in Table 2 does not enter any data, but displays message "DATA PROTECT".

(3) Protection of machining program (KEY 3)
When KEY 3 is turned off, the operation items listed in Table 3 are inhibited.

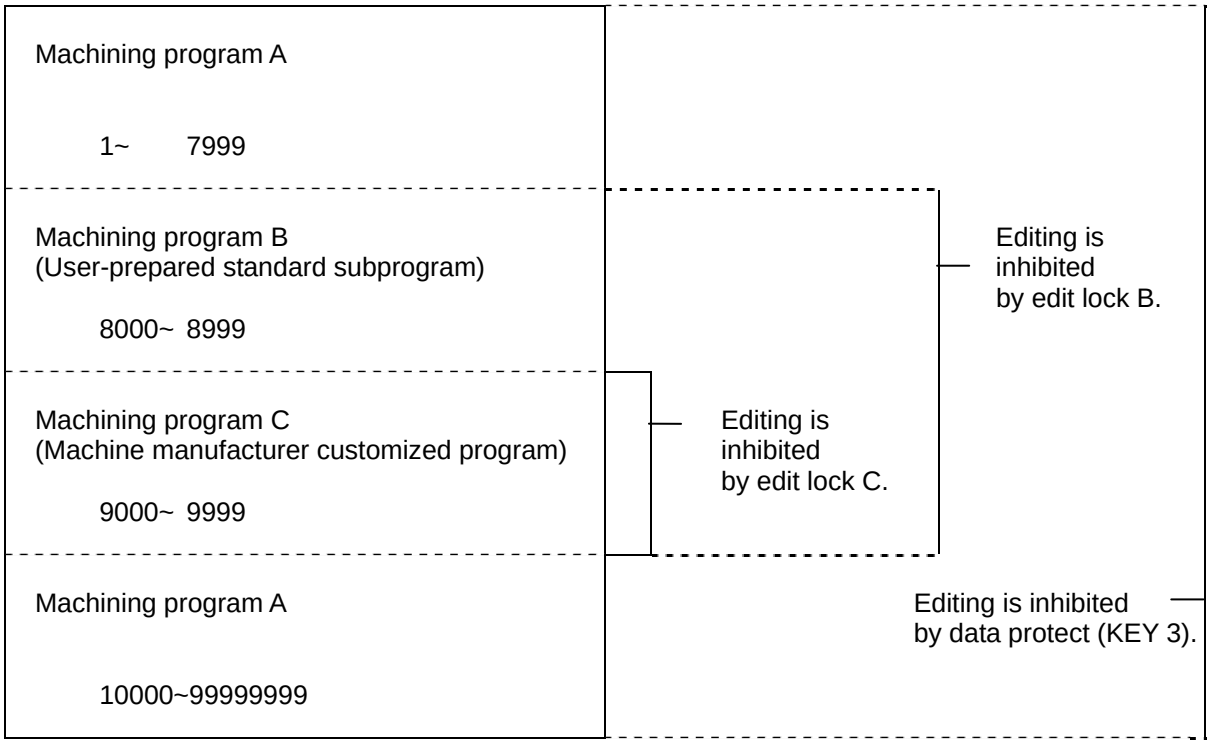
Table 3 Data protection by KEY 3

No.	Operation	Screen	Extended operation menu
1	Storing MDI data in memory	MDI	MDI ENTRY
2	Editing machining program	EDIT	—
3	Creating machining program	EDIT	PROGRAM
4	Setting comments of stored program	EDIT	FILE
5	Erasing machining program (single, group, all)	IN/OUT/ERASE	—
6	Setting comment of stored program	IN/OUT/FILE	—
7	Copying, condensing, and merging a machining program, and changing its number	IN/OUT/COPY	—
8	Modifying the buffer of machining program	MONITOR/ COORDINATE	—

(Note) When KEY 3 is off,
- Pressing any key other than the MENU key on the screens corresponding to numbers 1 to 7 in the above table or the extended operation menu screen causes message "DATA PROTECT" to be displayed; no data can be input.
- When an attempt is made to perform the operation of number 8 in Table 3, pressing the Tthe cursor key causes message "DATA PROTECT" to be displayed without the operation coming into effect.

Appendix 4.2 Edit Lock B, C

The edit lock function B or C inhibits machining program B or C from being edited or erased when these programs require to be protected.



Setting an edit lock affects the following operations on the EDIT/MDI and IN/OUT screens.

○ : Enabled × : Disabled

Screen		Operation	Edit lock B			Edit lock C		
			Work program			Work program		
			A	B	C	A	B	C
EDIT/ MDI	SEARCH	Data search	○	○	×	○	○	×
		Edit	○	×	×	○	○	×
	PROGRAM	Edit	○	×	×	○	○	×
	MDI ENTRY	MDI entry	○	×	×	○	○	×
IN/ OUT	COPY	Copy	○	×	×	○	○	×
		Condense	○	×	×	○	○	×
		Merge	○	×	×	○	○	×
		Program number change	○	×	×	○	○	×
	ERASE	Erase	○	×	×	○	○	×
	COMMENT	Comment setting	○	×	×	○	○	×
MONI-TOR	PROGRAM	Buffer	○	×	×	○	○	×

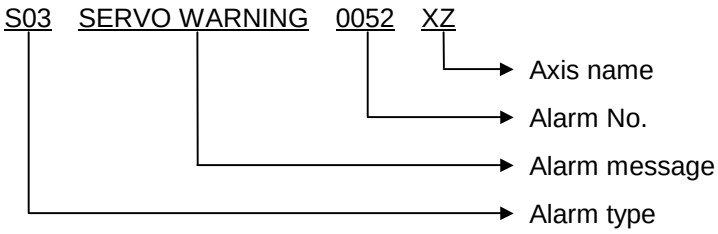
An attempt to perform any of the locked operations causes error message "E15 EDIT LOCK B" or "E16 EDIT LOCK C" to be displayed.

Appendix 5. Table of Conversion Codes for Error Code Output

With this function, the CNC alarms and errors which are normally output to CNC screen are partially coded and output to PLC I/F device. Thus, the contents of alarms and errors can be confirmed without CNC screen.

Appendix 5.1 Code Conversion Specifications

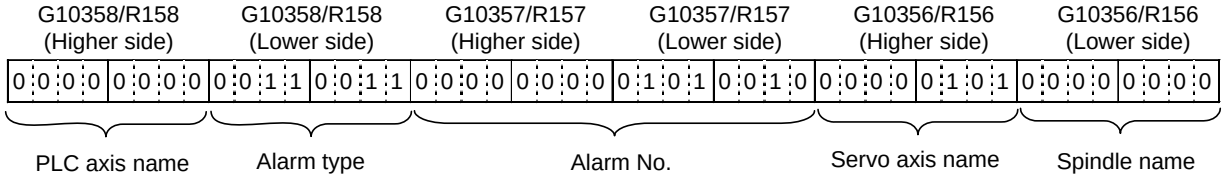
The following output will be made when an alarm occurs.
The message displayed in CNC screen is not converted.
(Example) When servo alarm S03 occurs



The output is as shown above on the CNC screen. However, this can be coded and output to PLC I/F device as shown below using this function.

Alarm type ……	Converted into a 2-digit numeral code. (Refer to the section 5.2.)
Alarm message ……	Not coded, and not output.
Alarm No. ……	The No. is output as HEX.
Axis name ……	Error occurrence axis is expressed as a bit, and the bit of servo and spindle are output separately. The head digit of the alarm without axis name will be “0”.

The output to PLC I/F device is as follows.



	bit							
	7	6	5	4	3	2	1	0
1st axis	0	0	0	0	0	0	0	1
2nd axis	0	0	0	0	0	0	1	0
3rd axis	0	0	0	0	0	1	0	0
4th axis	0	0	0	0	1	0	0	0
5th axis	0	0	0	1	0	0	0	0
6th axis	0	0	1	0	0	0	0	0
7th axis	0	1	0	0	0	0	0	0
8th axis	1	0	0	0	0	0	0	0

As for the servo axis name and spindle name, the bit corresponding to the No. of axis in which the alarm occurs is turned on.
The max servo axis Nos. in the 1st part system are 16, and the max spindle Nos. in the 1st part system are 7. The spindle alarm is output to the 1st part system.
The PLC axis name is output to the 1st part system.

Axis name is added for the messages such as some of M01, S01 to S52, and Z70 to Z73.
The following 48 bits are used as the output PLC I/F devices.

Appendix 5.2 Code Table

The alarm type is coded as the contents of following code table.

Alarm code list

Alarm		Details	Alarm type	Axis name	Priority
Multi-CPU error	A01	MULTI CPU ERROR	A1	Not added	1
System alarm	Z70	ABS. ILLEGAL	55	Added	2
	Z71	DETECTOR ERROR	56		
	Z72	COMPARE ERROR	57		
	Z73	ABS. WARNING	58		
Servo/spindle alarm	S01	SERVO ALARM: PR	31	Added	3
	S02	INIT PARAM ERR	32		
	S03	SERVO ALARM: NR	33		
	S04	SERVO ALARM: AR	36		
MCP alarm	Y02	System alarm	41	Some are added	4
	Y03	Amp. Unequipped	42		
	Y06	mcp_no setting error	49		
	Y07	Too many axes connected	43		
	Y09	Too many axisno connected	4A		
	Y11	Node detect error	4B		
	Y20	Safety observation error	4C		
Emergency stop	EMG	EMERGENC	01	Not added	5
Program error	Pxxx	(Program error)	71	Not added	6
	P990	PREPRO S/W ERR	61		
Servo/spindle warning	S51	PARAMETER ERROR	34	Added	7
	S52	SERVO WARNING	35		
MCP warning	Y51	Parameter error	45		8
	Y21	Safety observation warning	4E		
	Y90	No spindle signal	47		
System warning	Z30	ETHERNET ERROR	51	Not added	9
	Z52	BATTERY FAULT	52		
	Z53	TEMP_OVER	53		
	Z55	RIO NOT CONNECT	54		
	Z59	TIME CONSTANT	59		
Operation error	M01	OPERATION ERROR	11	Some are added	10
Stop code	T01	CAN'T CYCLE ST	21	Not added	11
	T02	FEED HOLD	22		
	T03	BLOCK STOP	23		
	T10	FIN WAIT	26		
Illegal PLC	U10	Built-in PLC alarm	91	Not added	12

The message at emergency stop is displayed in CNC screen shown below.
(Example) EMG EMERGENCY STOP PLC

When the emergency stop occurs, the message is coded and an alarm No. is output.

Emergency stop code list

Error message		Details	Alarm No.
Emergency stop (EMG)	EXIN	External emergency stop	0000
	PLC	Built-in PLC emergency stop	0001
	SRV	Servo drive unit not ready	0002
	STOP	PLC not running	0003
	SPIN	Spindle drive unit not ready	0004
	PC_H	PLC high-speed process error	0005
	PARA	Door open II fixed device setting illegal	0006
	STP2	Built-in PLC not running	0007
	LAD	Built-in PLC illegal code	0010
	MULT	Q and Qr bus alarm	0013
	IPWD	Illegal power down	0014
	CVIN	PS external emergency stop	0015
	MCT	Contactor shutoff test	0016
	SUIN	Emergency stop in the safety circuit	0017

Appendix 5.3 Restrictions

(1) If the MCP alarm "Y02 SYSTEM ALARM" occurs, part of the data will not be coded. Confirm the data on the display unit.
(Example) Y02 SYSTEM ALARM 0051 0104
In this case, only "Y02" and "0051" are coded and output to the PLC I/F device.

(2) If an alarm without an alarm No. occurs, "0" will be set as the alarm No. and output to the PLC I/F device.

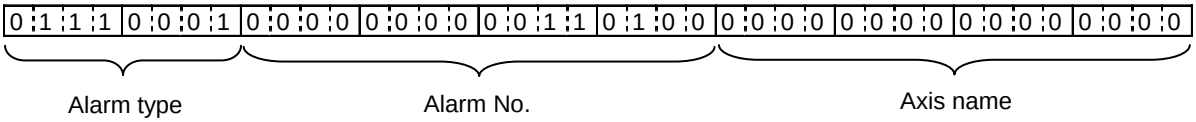
(3) If an alarm that does not have an axis name occurs, "0" will be set as the axis name and output to the PLC I/F device.

(4) Alarms not shown in the code table are not output to the PLC I/F device.

(5) If multiple alarms occur simultaneously, only the alarm with the highest order of priority in the code table will be output.

(6) The "OOO" section of the program error "POOO" is output to the alarm No. are$_a$.
(Example) "P34 G-CODE ERROR 0 0"

The code output becomes 71003400, and the output to PLC I/F devices is as follows.



Appendix 6. Operation Messages on Setting Display Unit

If a setting operation error occurs only any of the setting and display unit's screens, the error No. E○○ and a message indicating the details will appear.

Appendix 6.1 Operation Errors

Δ: Message requiring resetting and restarting
×: Message requiring restarting after canceling error conditions

(The bold characters are the messages displayed on the screen.)

Error No.	Error message		Details
E01	**SETTING ERROR**	Δ	• The setting data is incorrect. An alphabetic character was set when only number can be set, etc. • Data was input without setting number (#).
			(Word editing) • Even though no retrieval data was set, menu key "Word ↓ " /"Word ↑ ", or "STR. ↓ "/"STR. ↑ " was pressed. • Even though no data is stored in edit buffers, menu key "Replace" was pressed. • One of the following characters was entered as the first character of the retrieval data and edit buffers: 0 to 9, ".", " " (space), "+", "−", "=", "*", "[", and "]".
			• When the incremental detection system was used, the parameter (#0 INIT SET) was set on the absolute position setting screen.
			• The data input for the standard parameter setting or during execution of formatting is not "Y" or "N".
			• A value from 4 to 10 was specified for "#1043 lang". • Even though no language data exists, its output and comparison were attempted. Check the numbers (O253 and O254) of the language data to be output.
E02	**DATA OVER**	Δ	• The setting data exceeded the setting range.
			• The compensation data specification exceeded the range when inputting the tool offset data on tape, so that block could not be input. Press the INPUT key again while the input screen is displayed, and the input will continue from the next block.
			• When work coordinate offsets are measured, the calculation results given by pressing the CALC key are exceeding the specified range. Correctly specify the tool length or the abrasion data of cutting edges used for the calculation.
			• When there was no option, 2 or more was specified for "#1043 lang". Otherwise, an option was added and 16 or more was specified for "#1043 lang".
E03	**No. NOT FOUND**	Δ	• The corresponding setting No. (#) was not found. This error occurs if a setting No. not found on the screen was set and input, or if a variable No. not found in the specifications was set and input for the common variables. • When the tool length was measured manually, a nonexisting tool wear compensation number was specified and the sensor was turned on. Specify the R register of the offset number correctly.

Error No.	Error message		Details
E04	**DEV. NOT READY**	×	• The input/output u/nit power is not ON. • The cable is disconnected. • Setting of the transfer speed (baud rate) does not agree.
E05	**NOT ACCEPTABLE**	×	• The PLC timer cannot be set from the screen when the program is valid. (When machine parameter bit selection #6449 bit 1 is set to 1.) • The PLC counter cannot be set from the screen when the program is valid. (When machine parameter bit selection #6449 bit 0 is set to 1.)
			• The tool registration data setting is prohibited. • Setting from the tool life management screen is prohibited.
			• Absolute position setting screen's "#1 ORIGIN" and "#2 ZERO" cannot be set when "#0 INIT SET" is invalid.
			• The INPUT key was pressed to perform search for the program that is in background edit status on the word edit screen. • The menu keys (Replace and Insert) on the word edit screen were manipulated when a running program is displayed (PDISP signal: ON). • An attempt was made to set MDI data in an MDI setting lock state (the MDI setting lock parameter is specified with 0 and a non-MDI mode is valid).
			• Language data in display selection status was entered. Change the display selection status once before entering the data. (#1043 lang)
			• When the manual value command protection (#1228 aux12 /bit7) function is valid, the manual command operation (M, S, and T keys) is performed in POSITION screen.
E06	**NO SPEC**	×	• The menu key for a function not in the specifications was pressed. • A parameter not in the specifications was set. • A language that was not added as an option was selected. (#1043 lang)
E07	**RESET END**	Δ	• The input/output operations were forcibly stopped by reset, etc. (including EMG).
E08	**PHYSICAL ERR**	×	• The input/output parameter setting or input/output unit side setting was incorrect.
E09	**TIME OUT**	×	• The input/output unit parameter "TIME-OUT TIME" setting was too short. • There is no EOB code in the machining program.

Error No.	Error message		Details
E10	**MEMORY OVER**	×	• The program cannot be written because the memory capacity is exceeded. This error occurs when the MDI data setting on the MDI screen exceeds 500 characters, or when saving MDl, editing or making a program on the edit screen, input on the data input/output screen, program copy, etc.
E11	**PROG. No. DUPLI**	Δ	• When registering a machining program in the memory, a program with the same No. as the designated program No. was found in the memory. Refer to the program file to find a program No. not being used, and reset the program No. This error occurs during MDI registration in the MDI screen or during creation of a program in the edit screen.
E12	**FILE ENTRY OVER**	×	• When registering a machining program in the memory, the No. of programs determined in the specifications is exceeded, preventing registration. This error occurs during MDI registration in the MDI screen, creation of a program in the edit screen, data input in the data input/output screen, and program copy.
E13	**NB NOT FOUND**	Δ	• The block with the designated sequence No. or block No. does not exist in the designated program.
E14	**PROG. NOT FOUND**	Δ	• The designated program is not found in the memory. • The corresponding program No. was not found with search of tape memory during graphic check.
E15	**EDIT LOCK B**	×	• An operation (edit, input/output, buffer correction, etc.) inhibited for machining program B and C was attempted.
E16	**EDIT LOCK C**	×	• An operation (edit, input/output, buffer correction, etc.) inhibited for machining program B was attempted.
E17	**PARITY H ERR**	×	• A parity H error was detected during data input, etc. Check the paper tape or input device. This error may occur if the paper tape is dirtied with oil, etc.
E18	**PARITY V ERR**	×	• A parity V error was detected during data input. Check the paper tape to see whether the number of characters in the significant information section of a block is odd. Also check the state (cable wiring, noise measures, etc.) of the connected equipment.
E20	**OVER RUN ERR**	×	• The control method using the DC codes, etc., for the input/output operation is incorrect. Check the settings of the input/output unit parameters, and the settings on the input/output unit side, and reset if necessary.
E21	**PROGRAM RUNNING**	×	• Deletion of a machining program was attempted during operation. • Search was attempted during operation. • Change of data such as parameters was attempted during operation. • Start of graphic check was attempted during operation.
E22	**CODE CHANGE ERR**	×	• There was an illegal code on the paper tape.

Error No.	Error message		Details
E24	**PLC RUN**	×	• Data input/output or comparison was attempted when the built-in PLC was not stopped. (Measures) • Stop the built-in PLC. (Set the right rotary switch 2 of CNC CPU module to "1".)
E25	**DATA MEMORY ERR**	×	• When inputting the tool offset data onto tape, an offset type exceeding the specifications range was designated, and that block could not be input. If the input key is pressed again in the input screen, the input will continue from the next block.
E26	**NO CHARACTERS**	Δ	• The designated character string was not found from the block displayed on the screen to the end of the program when searching with data search in the edit screen. Press the input key again, and the search will start at the head of the program.
E35	**COMPARE ERROR**	×	• An inconsistency was found in the paper tape and memory data during comparison.
E50 **E51** **E52** **E53** **E54** **E55** **E56**	**FILE ERROR** **FILE OPEN ERROR** **FILE CLOSE ERR** **FILE SEEK ERR** **FILE READ ERR** **FILE DELETE ERR** **FILE INSERT ERR**	×	• If one of these errors occurs, the editing or input/output operations cannot be continued. Contact the service center. As for E50, a classification No. will display at the end of the message. Inform the service center of this No. as well.
E60	**IOP ERR □□**	×	A classification number is displayed after the message for E60. Refer to the section shown in parentheses, and remedy the problem. E60 lOP ERROR – 4 (E09 TIME OUT) E60 lOP ERROR – 5 (E08 PHYSICAL ERR) E60 lOP ERROR – 7 (E07 RESET END) E60 lOP ERROR – 10 (E04 DEV. NOT READY) E60 lOP ERROR – 15 (E17 PARITY H ERR) E60 lOP ERROR – 16 (E18 PARITY V ERR) E60 lOP ERROR – 17 (E20 OVER RUN ERROR) E60 lOP ERROR – 18 (E22 CODE CHANGE ERROR) E60 IOP ERROR – 20 (framing and H/W errors)

Error No.	Error message		Details
E60	**IOP ERR ☐☐**	×	• Setting for the bit length is incorrect. (Baud rate, stop bit, and character length) Check the setting of the I/O device system and its parameters and set it again. • Check the situations of the connected devices (cable wiring and noise measures).
E62	**I/O PARAM ERR**	Δ	• The “EIA code” data set for I/O parameter is of an unusable code. The unusable codes are those used as the EIA standard codes and the even hole codes.
E64	**PROGRAM No. ERR**	Δ	• The same No. as the program No. designated for program copy was found in the memory. • During tape input, the first character of the machining program block is the program No. address "O" or "L".
E65	**PROG. No. DUPLI**	Δ	• During tape input, the same No. as the specified program was found in the memory.
E66	**NO PROG. NUMBER**	Δ	• During tape input, the program No. was not found on the paper tape, and a program No. was not designated on the screen’s data setting area. Set the program No., and input again.
E69	**PROG. CHECK MODE**	×	• Search (operation search) was attempted during program check (continuous or step). Retry search after the program check is completed, or after resetting the program search
E70	**TOOL No. DUPLI**	Δ	• A tool No. already registered was newly registered on the tool life management screen.
E71	**TOOL ENTRY OVER**	×	• Registration of data exceeding the max. No. of registerable tools was attempted on the tool life management screen. • When inputting the tool offset data onto tape, a compensation number exceeding the specifications range was specified, and that block could not be input. If the input key is pressed again in the input screen, the input will continue from the next block.
E76	**TOOL No. ERROR**	×	• The offset No. to be used for workpiece coordinate system offset data measurement was invalid. Restart from tool selection. (Correctly specify the R register that contains the offset number.)
E77	**AXIS NO REF-RET**	×	• Zero point return has not been completed for the axis being measured. Return the axis to the zero point.
E78	**AX UNMATCH (TLM)**	×	• During movement of two or more axes, the sensor turned on and the tool length was measured. Keep off from the sensor and perform the measurement for one axis at a time.
E79	**NO REF-RTN (TLM)**	×	• The sensor turned on for an axis that has not completed dog-type reference point return, and the tool length was measured. Return the axis to measure to the reference point.
E84	**CAN’T IN/OUT**	×	• Parameters were input in the setup parameter lock state. Refer to the instruction manual issued by the machine maker. • When using the high-speed program server function, the parameter "#1925 EtherNet" parameter was set to "0".

Error No.	Error message		Details
E86	INPUT DATA ERR	×	• When inputting the tool offset data, the data format was not correct, so that block could not be input. If the input key is pressed again in the input screen, the input will continue from the next block. • When data is read from parameter tape, its format is incorrect.
E87	NOT EDIT PROG.	×	• Playback edit was executed for a fixed cycle subprogram. Playback edit of a fixed cycle subprogram is not possible.
E88	CAN'T ADD BLOCK	×	• Playback edit cannot be executed unless the block being edited with playback is displayed to the end (EOB) on the left side of the machining program display area. Press the cursor key [↓], and display the whole block to the end. Then, input the data.
E91	MODE ERROR (PBK)	×	• G90 was set when "PLAYBACK G90" was off. • G91 was set when "PLAYBACK G90" was on.
E110	ATA NOT READY	Δ	• ATA not mounted error when that appears when ATA Card screen is opened Mount the ATA card.
E111	FILE DUPLICATE	Δ	• The name of the file to be output was found in the ATA card. • New file name to be renamed already exists.
E112	ATA I/F ERROR	×	• The designated file is damaged or the ATA card is damaged. Designate another file or replace the ATA card.
E113	MEMORY OVER	×	• The ATA card capacity was filled when outputting to the ATA card. Delete files from the ATA card, or use another ATA card and output the data again.
E114	CAN'T ERASE	×	• A directory containing subdirectories or files was required to be deleted. Delete the file and all subdirectories before deleting the directory.
E115	FILE NOT FOUND	Δ	• The file to be input, compared, deleted or renamed was not found in the ATA card. Designate a file saved in the ATA card.
E117	DIR NOT FOUND	×	• The set directory was not found on the disk. Check whether an intermediate directory was designated when designating an absolute path.
E190	FORE EDITING	×	• An attempt was made to perform background search for the program that is in foreground search status. (Word editing)
E191	NOT COM. SEARCH	×	• Operation search was attempted in the tape mode.
E200	ADJUST ERROR	×	• The hardware status can't be read correctly, so automatic adjustment was not possible. • Check the remote I/O unit. • A Z55 RIO NOT CONNECT error occurred. • Adjust manually. • Unit defect (replace unit)

Error No.	Error message		Details
E201	**UNIT NOT EQUIP**	×	• The analog output unit is not mounted. • Confirm the remote I/O unit. • Prepare a unit having analog output. • Check the connection (power and signal wires) • Unit defect (replace unit)
E301	**CONNECT ERROR**	×	• The socket connection failed during Ethernet communication. • If the host address is illegal or the port No. is incorrect, set correctly.
E302	**LOGIN ERROR**	×	• Log in failed during Ethernet communication. • Check the user name and password.
E311	**DOWNLOAD ERR**	×	• Reading of a file on the host side failed during Ethernet communication.
E312	**UPLOAD ERROR**	×	• Writing of a file on the host side failed during Ethernet communication.
E313	**NO FILE**	×	• The file designated with the host reception (host → IC) operation was not found on the host side during Ethernet communication. • The file designated with the host transmission (IC → host) operation was not found in the IC card during Ethernet communication.
E314	**FILE DUPLICATE**	×	• The file name to be registered with the host reception (host → IC) operation was found in the IC card during Ethernet communication. • The file name to be registered with the host transmission (IC → host) operation was found in the host during Ethernet communication.
E315	**FILE WRITE ERR**	×	• Writing to the IC card failed during Ethernet communication.
E316	**FILE READ ERR**	×	• Reading of an IC cared file failed during Ethernet communication.
E317	**MEMORY OVER**	×	• The IC card memory is full. • The NC memory is full.
E318	**OVER FLOW ERR**	×	• There are too many files in the host directory.
E319	**DIRECTORY ERR**	×	• Movement of the directory failed.

Appendix 6.2 Operator Messages

The following messages indicate the status of the setting and display functions, and are not operation errors. They are mainly used to show that operation is normal, and serve as guides for the following operations. There is no classification by numbers.

(1) Search and operation related

Message	Message details
SEARCH EXECUTION	• Search is being executed normally.
SEARCH COMPLETE	• Search was completed normally.

(2) MDI/editing related

Message	Message details
MDI NO SETTING	• Only display of MDI data (no execution)
MDI SETTING COMPLET	• The MDI data setting has been completed (execution is now possible).
MDI ENTRY COMPLETE	• The MDI data was saved in the memory with the specified program No.
MDI RUNNING	• The NC is operating with an MDI program, and the MDI data cannot be corrected.
PUSH KEY SEARCH/PROG	• Status in which no programs to be edited have been called on the editing screen. To edit, press the SEARCH or PROGRAM edit key.
EDITING	• The details of a program are being edited on the screen. Press INPUT to write the data in the memory.
EDIT (REP.)	• The program can be edited on the screen in the replace mode.
EDIT (INS.)	• The program can be edited on the screen in the insert mode.
EDIT (CAN'T INSERT)	• The data can be inserted no more because of the shortage of space area while editing the program on the screen with insert mode. Cancel the insertion-impossible state by moving a cursor and release some data area.
EDIT (CAN'T REPLACE)	• The data can be replaced no more since the cursor reached the end of editing area while editing the program on the screen with replace mode. Cancel the replace-impossible state by cursor, DEL key, or C•B/CANCEL.

Message	Message details
IGNORE CHANGE (Y/N)	• The keys to change the display such as NEXT/BACK page keys, menu change key, screen change key, or scroll keys, were pressed while editing a program on the screen. To cancel the changed contents, select “Y”. To leave the changes as they are, select “N”.
PROGRAM RUNNING	• A machining program to be edited is currently being run with memory operation, and cannot be edited.
DELETE?	• Waiting for a key entry (whether to delete the program) in word edit status (when the background search menu is selected)
BACK GROUND EDITING	• Background edit mode
EDIT POSSIBLE	• Editing can be performed in foreground edit mode.
EDIT IMPOSSIBLE	• Editing cannot be performed in foreground edit mode. • This state also occurs during feed hold or fixed cycle mode (single-block stop).
WORD SEARCH FIN	• The word matching the search data was searched on word editing.

(3) Data input/output related

Message	Message details
DATA IN EXECUTION	• Data is being read without error from the external memory.
DATA WRITING	• Data has been entered normally and the input data is being written to the ROM.
DATA IN COMPLETE	• Data has been stored without error.
COMPARE EXECUTION	• Comparison is being executed without error.
COMPARE COMPLETE	• Comparison has completed without error.
DATA OUT EXECUTION	• Data is being output to the external memory without error.
DATA OUT COMPLETE	• Data has been output without error.
ERASE EXECUTION	• Data is being erased without error.
ERASE COMPLETE	• Data has been erased without error.
COPY EXECUTION	• Data is being copied without error.
COPY COMPLETE	• Data has been copied without error.
CONDENSE EXECUTION	• The machining program is being condensed without error.
CONDENSE COMPLETE	• The machining program has been condensed without error.
MERGE EXECUTION	• The machining program is being merged without error.
MERGE COMPLETE	• The machining program has been merged without error.
No. CHANGE EXECUTION	• The machining program No. is being changed without error.
No. CHANGE COMPLETE	• The machining program No. has been changed without error.
CHANGE DIR COMPLETE	• The directories in the ATA card were changed correctly.
CREATE? Y/N	• Confirmation for creating a directory when the designated directory was not found in the external memory. [Y] [INPUT] : A directory will be created. [N] [INPUT] : A directory will not be created.
OVERWRITE? Y/N	• The number of the program to be input was found in the NC memory, or the name of the file to be output was found in the external memory. [Y] [INPUT] : Overwrites. [N] [INPUT] : Does not overwrite.
ERASE? Y/N	• Confirmation for deleting files and directories in the external memory. [Y] [INPUT] : Deletes. [N] [INPUT] : Does not delete.
RENAME COMPLETE	• The name of the file in the external memory was renamed correctly.

(4) Others

Message	Message details
DATA PROTECTING	• The data protection key is valid, and the various data cannot be set or erased, etc.
BASE PARA. SET? (Y/N)	• Waiting for the key input of standard parameter setting (Y/N).
BASE PARA. EXECUTION	• The standard parameters are being set.
FORMAT? (Y/N)	• Waiting for the key input of execute format (Y/N).
FORMAT EXECUTION	• Formatting is being executed.
SETUP COMPLETE	• The simple setup has been completed.
NON SETUP	• Completed without executing simple setup. (When "N" has been set for either "Standard parameter setting? (Y/N)" or "Execute format? (Y/N)".)
CONFIRM OPE? (Y/N)	• Confirmation for erasing operating time or alarm history.

Appendix 6.3 Dialog Error Messages on Input/Output Screen

The following error messages will appear if an illegal operation is carried out on the Data I/O screen. Operations can be continued by pressing the "OK" button.

The error messages are listed below.

Function	Target	Error message	Cause of error
Copy	NC	NC SYSTEM NO. ILLEGAL	The NC No. Mxx's xx is not within the designated range.
		DESIGNATED FILE NAME ALREADY EXIST	The copy destination file already exists.
		DIRECTRY ERROR	The copy destination and copy source directories were not found.
		FILE NOT FOUND	The copy source file was not found.
		FILE NAME FORMAT ILLEGAL	The file name format is illegal. (The extension is wrong, etc.)
		CAN'T READ OUT FILE	The copy source file cannot be read.
		CAN'T WRITE FILE	The copy destination file cannot be written.
		UNABLE TO COPY : PROGRAM RUNNING	Automatic starting, MDI execution or history file sampling is in progress.
		UNABLE TO COPY : PLC RUNNING	Cannot copy because the built-in PLC is running.
		SOME ERROR FOUND IN FILE SYSTEM	There is an error in the file system.
		NO. OF REGISTRATION OVER	The number of registerable programs was exceeded.
		OUT OF MEMORY	The storage capacity was exceeded.
		DATA PROTECT	The data is protected.
		FILE NAME TOO LONG	The file name is too long. (Longer than 256 characters)
		Can't write file because other data in/out function is accessing this NC. Please try again later.	Another input/output function is being executed via the network.
	Memory card	DESIGNATED FILE NAME ALREADY EXIST	There is already a file in the PC.
		DIRECTRY ERROR	The directory was not found.
		FILE NOT FOUND	The file was not found.
		CAN'T OPEN FILE	Opening failed when reading files on PC.
		CAN'T CREATE FILE	Opening failed when writing files on PC.
		TRANSFER ERROR : SOURCE FILE	An error occurred when reading files from PC.
		TRANSFER ERROR : DESTINATION FILE	An error occurred when writing files to PC.
		DRIVE NOT EXIST	The designated drive was not found in the PC.
		Can't read out file because other data in/out function is accessing this NC. Please try again later.	Another input/output function is being executed via the network.

Function	Target	Error message	Cause of error
Delete	NC	CAN'T DELETE FILE	The file is a parameter file, etc., and cannot be deleted.
		DIRECTRY ERROR	The directory does not exist or was not designated.
		FILE NOT FOUND	The file was not found.
		FILE NAME FORMAT ILLEGAL	The file name format is illegal.
		UNABLE TO DELETE : PROGRAM RUNNING	The file cannot be deleted. (The file is running.)
		FILE-SYSTEM ERROR. PLEASE DO SETUP PROCESS	There is an error in the file system.
		DATA PROTECT	The data is protected.
		UNABLE TO DELETE : PLC RUNNING	Cannot delete because the built-in PLC is running.
		Can't write file because other data in/out function is accessing this NC. Please try again later.	Another input/output function is being executed via the network.
	Memory card	CAN'T DELETE FILE	A file that does not exist on the PC was required to be deleted.
		DIRECTRY ERROR	The designated directory is not in the PC.
		FILE NOT FOUND	The designated file does not exist.
		DRIVE NOT EXIST	The designated drive was not found in the PC.
		DIRECTRY ERROR	The directory was not found.
		FILE NOT FOUND	The file was not found.
		CAN'T OPEN FILE	The directory could not be opened.
		TRANSFER ERROR : SOURCE FILE	Calling of the file information failed.
		DRIVE NOT EXIST	The drive does not exist.

Appendix 7. Explanation of Alarms

Appendix 7.1 Operation Errors (M)

(Note) "M01"alarms are displayed as "M01 Operation error" with the error number. Error number is four digit number displayed after error name(such as 0001). "M01" alarms are listed in ascending order in this manual.

M01 Dog overrun 0001

Details

When returning to the reference position, the near-point detection limit switch did not stop over the dog, but overran the dog.

Remedy

- Increase the length of the near-point dog.
- Reduce the reference position return speed.

M01 Some ax does not pass Z phase 0002

Details

One of the axes did not pass the Z-phase during the initial reference position return after the power was turned ON.

Remedy

- Move the detector one rotation or more in the opposite direction of the reference position, and repeat reference position return.

M01 R-pnt direction illegal 0003

Details

When manually returning to the reference position, the return direction differs from the axis movement direction selected with the AXIS SELECTION key.

Remedy

- The selection of the AXIS SELECTION key's +/- direction is incorrect. The error is canceled by feeding the axis in the correct direction.

M01 External interlock axis exists 0004

Details

The external interlock function has activated (the input signal is "OFF") and one of the axes has entered the interlock state.

Remedy

- As the interlock function has activated, release it before resuming operation.
- Correct the sequence on the machine side.
- Check for any broken wires in the "interlock" signal line.

M01 Internal interlock axis exists 0005

Details

The internal interlock state has been entered.
The absolute position detector axis has been removed.
A command for the manual/automatic simultaneous valid axis was issued from the automatic mode.
The manual speed command was issued while the "tool length measurement 1" signal is ON.

Remedy

- The servo OFF function is valid, so release it first.
- An axis that can be removed has been issued, so perform the correct operations.
- The command is issued in the same direction as the direction where manual skip turned ON, so perform the correct operations.
- During the manual/automatic simultaneous mode, the axis commanded in the automatic mode became the manual operation axis. Turn OFF the "manual/automatic valid" signal for the commanded axis.
- Turn ON the power again, and perform absolute position initialization.
- Turn OFF the "tool length measurement 1" signal to start the program by the manual speed command.

M01 H/W stroke end axis exists 0006

Details

The stroke end function has activated (the input signal is "OFF") and one of the axes is in the stroke end status.

Remedy

- Move the machine manually.
- Check for any broken wires in the "stroke end" signal line.
- Check for any limit switch failure.

M01 S/W stroke end axis exists 0007

Details

The stored stroke limit I, II, IIB or IB function has activated.

Remedy

- Move the machine manually.
- Correct any setting error of the parameters for the stored stroke limit.

M01 Chuck/tailstock stroke end ax 0008

Details

The chuck/tail-stock barrier function turned ON, and an axis entered the stroke end state.

Remedy

- Reset the alarm with reset, and move the machine in the reverse direction.

M01 Ref point return No. invalid 0009

Details

2nd reference position return was performed before 1st reference position return has been completed.

Remedy

- Execute 1st reference position return.

M01 Sensor signal illegal ON 0019

Details

The sensor signal was already ON when the "tool length measurement 1" signal was validated.
The sensor signal turned ON when there was no axis movement after the "tool length measurement 1" signal was validated.
The sensor signal turned ON at a position within 100 μm from the final entry start position.

Remedy

- Disable the "tool length measurement 1" signal and move the axis in a safe direction.
- Disabling the sensor signal also clears the operation alarm.

(Note) When the "tool length measurement 1" signal is disabled, the axis can be moved in either direction. Pay attention to the movement direction.

M01 Ref point retract invalid 0020

Details

Reference position retract was performed while the coordinates had not been established.

Remedy

- Execute reference position return.

M01 Tool ofs invld after R-pnt 0021

Details

Reference position return had been performed during the tool escape and return, which invalidated the tool compensation amount after the reference position return.

Remedy

- The error is cleared if the operation mode is changed to other than reference position return before the axis performs reference position return.
- The error is cleared when the tool return is completed.
- The error is cleared if reset 1 is input or the emergency stop button is pushed.

M01 R-pnt ret invld at abs pos alm 0024

Details

A reference position return signal was enabled during an absolute position detection alarm.

Remedy

- Reset the absolute position detection alarm, and then perform the reference position return.

M01 R-pnt ret invld at zero pt ini 0025

Details

A reference position return signal was input during zero point initialization of the absolute position detection system.

Remedy

- Complete the zero point initialization, and then perform reference position return.

M01 High-accuracy skip disabled 0028

Details

The drive unit's hardware or software does not conform to the high-accuracy skip.

Remedy

- The software or hardware does not conform to the function. Contact service center.

M01 Hi-ac skip coord retrieval err 0029

Details

Failed to retrieve the skip coordinate value from the drive unit.

Remedy

- Check the wiring.
- Check the parameters.

M01 Now skip on 0030

Details

The "skip input" signal remains enabled when the operation has shifted from skip retract to measurement.

Remedy

- Increase the skip retract amount.

M01 No skip 0031

Details

Even though the 1st skip was to the correct position, the 2nd skip could not be found.

Remedy

- Check whether the measurement target has moved.

M01 Movement prohibited during tool retract 0035

Details

An axis movement was attempted from the tool retract position.
The movement was attempted by a manual command in the tool return.

Remedy

It is not allowed to move an axis arbitrarily from the tool retract position. Take the following steps to move the axis.
- Cancel the program execution by reset.
- Use the tool escape and retract function and let the axis escape from the tool retract position.

The interruption by a manual command is not allowed in a tool return. Take the following step to move the axis.
- Cancel the program execution by reset.

M01 Chopping axis R-pnt incomplete 0050

Details

Chopping mode has been entered while the chopping axis has not completed reference position return.
All axes interlock has been applied.

Remedy

- Reset the NC or disable the "chopping" signal, and then carry out the reference position return.

M01 Synchronous error excessive 0051

Details

The synchronization error of the primary and secondary axes exceeded the allowable value under synchronous control. A deviation exceeding the synchronization error limit value was found with the synchronization deviation detection.

Remedy

- Select the correction mode and move one of the axes in the direction in which the errors are reduced.
- Increase "#2024 synerr(allowable value)" or set "0" to disable error check.
- When using simple C-axis synchronous control, set "0" for "synchronous control operation method".

M01 No spindle select signal 0053

Details

Synchronous tapping command was issued when the "spindle selection" signals for all spindles were OFF in the multiple-spindle control II.

Remedy

- Turn ON the "spindle selection" signal for the tapping spindle before performing the synchronous tapping command.

M01 No spindle serial connection 0054

Details

Synchronous tapping command was issued in the multiple-spindle control II, while the spindle with the "spindle selection" signal ON was not serially connected.

Remedy

- Make sure the "spindle selection" signal for the spindle is ON.
- Consider the machine construction when issuing the command.

M01 Spindle fwd/rvs run para err 0055

Details

Asynchronous tapping command was issued when M code of the spindle forward/reverse run command, set by "#3028 sprcmm", was one of the followings in the multiple-spindle control II.

- M0, M1, M2, M30, M98, M99, or M198
- M code No. that commands to enable/disable the "macro interrupt" signal

Remedy

- Correct the "#3028 sprcmm (Tap cycle spindle forward run/reverse run M command)" setting.

M01 Tap pitch/thread number error 0056

Details

The command for the pitch or the number of threads is not correct in the synchronous tapping command of the multiple-spindle control II.
The pitch is too small for the spindle rotation speed.
Thread number is too large for the spindle rotation speed.

Remedy

- Correct the pitch, number of threads or rotation speed of the tapping spindle.

M01 Wait for tap retract 0057

Details

The axis travel command is interlocked in the part system where the "Tap retract possible" signal is ON.

Remedy

- If tap retract is necessary, perform it before issuing an axis travel command.
- If tap retract is not necessary, cancel the tap retract enabled state.

M01 Handle ratio too large 0060

Details

- The handle ratio is too large for the handle feed clamp speed.
(The handle feed clamp speed changes according to the rapid traverse rate, external feedrate, maximum speed outside the soft limit range and etc. (or external deceleration speed when external deceleration is valid))

Remedy

- Change the settings of the handle feed clamp speed or the handle ratio.

M01 R-pos offset value illegal 0065

Details

At the start of reference position initial setting, "#2034 rfpofs (Distance-coded reference position detection offset) is not set to "0".

Remedy

- Set "#2034 rfpofs" to "0", then turn the power ON again to perform the reference position initial setting.

M01 R-pos scan distance exceeded 0066

Details

Reference position could not be established within the maximum scan distance.

Remedy

- Check the scale to see if it has dirt or damage.
- Check if the servo drive unit supports this function.

M01 Hypothetical axis high-accuracy control: Non-interpolation error 0090

Details

High-accuracy control was commanded in hypothetical axis command mode in the hypothetical linear axis control. Otherwise, non-interpolation mode is selected.

Remedy

- Correct "#1086 G0Intp (G00 non-interpolation)" and "#1205 G0bdcc (Acceleration and deceleration before G0 interpolation)" settings.

M01 Hypothetical linear axis control: Commanded axis illegal 0091

Details

Under hypothetical linear axis control, a command was issued to an actual axis on hypothetical plane in hypothetical axis command mode or issued to a hypothetical axis in actual axis command mode.

Remedy

- Change the commanded axis or command mode.

M01 Hypothetical axis movable range exceeded 0092

Details

A hypothetical axis is being moved outside the movable range.

Remedy

- Correct the following parameter setting:
  "#12015 v_leng (Hypothetical axis tool length)",
  "#12016 v_ori (Hypothetical axis machine zero point)",
  "#12020 r_lim+ (Actual axis movable range (+))",
  "#12021 r_lim- (Actual axis movable range (-))"

M01 No operation mode 0101

Details

No operation mode

Remedy

- Check for any broken wires in the input mode signal line.
- Check for any failure of the MODE SELECT switch.
- Correct the sequence program.

M01 Cutting override zero 0102

Details

The "cutting feed override" switch on the machine operation panel or the "rapid traverse override" switch is set to"0".
The override was set to "0" during a single block stop.

Remedy

- Set the "cutting feed override" switch or the "rapid traverse override" switch to a value other than "0" to clear the error.
- If the "cutting feed override" switch or the "rapid traverse override" switch has been set to a value other than "0", check for any short circuit in the signal line.
- Correct the sequence program.
- When using the cutting feedrate override method selection or the rapid traverse override method selection, check if the override ratio is not zero.

M01 External feed rate zero 0103

Details

MANUAL FEEDRATE switch on the machine operation panel is set to "0" when the machine is in the JOG or automatic dry run mode.
"Manual feedrate B" is set to "0" during the JOG mode when manual feedrate B is valid.
"Each axis manual feedrate B" is set to "0" during the JOG mode when each axis manual feedrate B is valid.

Remedy

- Set the MANUAL FEEDRATE switch to a value other than "0" to release the error.
- If the MANUAL FEEDRATE switch has been set to a value other than "0" check for any short circuit in the signal line.
- Correct the sequence program.
- Correct the external deceleration parameters as follows:
  When "#1239 set11/bit6" is set to "0", set a non-zero value in "#1216 extdcc".
  When "#1239 set11/bit6" is set to "1", set a non-zero value in "#2086 exdcax1" or "#2161 exdcax2" - "#2165 exdcax6" referring to the value set in the external deceleration speed selection signal.

M01 F 1-digit feed rate zero 0104

Details

The F1-digit feedrate has been set to "0" when the F1-digit feed command was executed.

Remedy

- Set the F1-digit feedrate (from "#1185 spd_F1 (F1 digit feedrate F1)" to "#1189 spd_F5 (F1 digit feedrate F5)").

M01 Spindle stop 0105

Details

The spindle stopped during the synchronous feed/thread cutting command.

Remedy

- Rotate the spindle.
- If the workpiece is not being cut, start dry run.
- Check for any broken wire in the spindle encoder cable.
- Check the connections for the spindle encoder connectors.
- Check the spindle encoder pulse.
- Correct the program. (commands and addresses)

M01 Handle feed ax No. illegal 0106

Details

The axis, designated at handle feed, is out of specifications.
No axis has been selected for handle feed.

Remedy

- Check for any broken wires in the handle feed axis selection signal line.
- Correct the sequence program.
- Check the number of axes in the specifications.

M01 Spindle rotation speed over 0107

Details

Spindle rotation speed exceeded the axis clamp speed during the thread cutting command.

Remedy

- Lower the commanded rotation speed.

M01 Fixed pnt mode feed ax illegal 0108

Details

The axis, designated in the manual arbitrary feed, is out of specifications.
The feedrate in manual arbitrary feed mode is illegal.

Remedy

- Check for any broken wires in the axis selection signal line or the feedrate line for the manual arbitrary feed mode.
- Check the specifications for the manual arbitrary feed mode.

M01 Block start interlock 0109

Details

An interlock signal has been input to lock the block start.

Remedy

- Correct the sequence program.

M01 Cutting block start interlock 0110

Details

An interlock signal has been input to lock the cutting block start.

Remedy

- Correct the sequence program.

M01 Restart switch ON 0111

Details

Restart switch has been turned ON and manual mode has been selected before the restart search is completed.

Remedy

- Search the block to restart.
- Turn the restart switch OFF.

M01 Program check mode 0112

Details

The automatic start button was pressed during program check or in program check mode.

Remedy

- Press the reset button to cancel the program check mode.

M01 Auto start in buffer correct 0113

Details

The automatic start button was pressed during buffer correction.

Remedy

- Press the automatic start button after the buffer correction is completed.

M01 In reset process 0115

Details

The automatic start button was pressed during resetting or tape rewinding.

Remedy

- When rewinding the tape, wait for the winding to end, or press the reset button to stop the winding, and then press the automatic start button.
- During resetting, wait for the resetting to end, and then press the automatic start button.

M01 Playback not possible 0117

Details

The playback switch was turned ON during editing.

Remedy

- Cancel the editing by pressing the input or previous screen key before turning ON the playback switch.

M01 Turn stop in normal line cntrl 0118

Details

The turning angle at the block joint exceeded the limit during normal line control.
In normal line control type I:
"#1523 C_feed (Normal line control axis turning speed)" has not been set.
In normal line control type II:
When turning in the inside of the arc, the set value for "#8041 C-rot. R" is larger than the arc radius.

Remedy

- Correct the program.
- Correct the "#1523 C_feed (Normal line control axis turning speed)" setting.
- Correct the "#8041 C rot. R" setting.

M01 Reverse run impossible 0119

Details

Either of the following conditions occurred:

- there is no block to run backward.
- eight blocks has been continued without any travel command.

Remedy

- Execute forward run to clear the alarm.
- Reset to clear the alarm.

M01 In synchronous correction mode 0120

Details

The synchronous correction mode switch was pressed in non-handle mode.

Remedy

- Select the handle or manual arbitrary feed mode.
- Turn OFF the correction mode switch.

M01 No synchronous control option 0121

Details

The synchronous control operation method was set (with R2589) while no synchronous control option was provided.

Remedy

- Set "0" for "synchronous control operation method".

M01 Computer link B not possible 0123

Details

Cycle start was attempted before resetting was completed.
Computer link B operation was attempted at the 2nd or further part system in a multi-part system.

Remedy

- Perform the cycle start after resetting has been completed.
- Set "#8109 HOST LINK" to "0" and then set to "1" before performing the cycle start.
- Computer link B operation cannot be performed at the 2nd or further part system in a multi-part system.

M01 X/Z axes simultaneous prohibit 0124

Details

The basic axis corresponding to the inclined axis was started simultaneously in the manual mode while the inclined axis control was valid.

Remedy

- Turn the inclined axis and basic axis start OFF for both axes. (This is also applied for manual/automatic simultaneous start.)
- Disable the basic axis compensation, or command it to axes one by one.

M01 Rapid override zero 0125

Details

The RAPID TRAVERSE OVERRIDE switch on the machine operation panel is set to "0".

Remedy

- Set the RAPID TRAVERSE OVERRIDE switch to a value other than "0" to clear the error.
- If the RAPID TRAVERSE OVERRIDE switch has been set to a value other than "0", check for any short circuit in the signal line.
- Correct the sequence program.

M01 Program restart machine lock 0126

Details

Machine lock was applied on the return axis being manually returned to the restart position.

Remedy

- Cancel the machine lock and resume the operation.

M01 Rot axis parameter error 0127

Details

Orthogonal coordinate axis name does not exist.
Rotary axis name does not exist.
A duplicate name is used for the designated orthogonal coordinate axis.
The number of axes that were selected to change tool length compensation along the tool axis amount exceeds the maximum number of axes.
The designated orthogonal coordinate axis name is the same as the rotary axis name.

Remedy

- Correct the rotary axis configuration parameters.

M01 Restart pos return incomplete 0128

Details

Automatic return was performed with an axis whose return to the restart position was not complete.

Remedy

- Perform restart position return manually.
- Enable "#1302 AutoRP (Automatic return by program restart)" before executing the automatic start.

M01 PLC interruption impossible 0129

Details

After the automatic startup, the "PLC interrupt" signal was turned ON during buffer correction, program restart, arbitrary reverse run, tool escape and return, high-speed high-accuracy control II, or NURBS interpolation.

Remedy

- Turning OFF or resetting the "PLC interrupt" signal will clear the alarm.

M01 Restart posn return disabled 0130

Details

Restart position return was attempted in a mode where the return is disabled.

Remedy

- Correct the program restart position.

M01 Chopping override zero 0150

Details

The override became "0" in the chopping operation.

Remedy

- Correct the setting of "chopping override" (R2503).
- Correct the setting of "rapid traverse override" (R2502).

M01 Command axis chopping axis 0151

Details

A chopping axis movement command was issued from the program during the chopping mode. (This alarm will not occur for the command with the movement amount "0".)
(All axes interlock state will be applied.)

Remedy

- Press the reset button or turn OFF the "chopping" signal. When the "chopping" signal is turned OFF, the axis returns to the reference position and performs the movement command in the program.

M01 Bottom dead center pos. zero 0153

Details

The bottom dead center position is set to the same position as the upper dead center position.

Remedy

- Correct the bottom dead center position.

M01 Chopping disable for handle ax 0154

Details

Chopping has been attempted while the chopping axis is selected as the handle axis.

Remedy

- Select an axis other than the chopping axis as the handle axis, or start chopping after changing the mode to the other mode.

M01 No speed set out of soft limit 0160

Details

The axis, without any maximum speed outside of the soft limit range set, was returned from the outside of the soft limit range.

Remedy

- Correct the "#2021 out_f (Maximum speed outside soft limit range)" setting.
- Correct the soft limit range (with "#2013 OT- (Soft limit I-)" and "#2014 OT+ (Soft limit I+)").

M01 Aux axis changeover error 0166

Details

One of the following attempts was made on an axis that can be switched to NC/auxiliary axis.

- A command was issued to an auxiliary axis from machining program.
- When there were more than one NC axis having a same name, a command was issued to those axes from machining program.
- "NC axis control selection" signal was turned OFF while the NC axis was in motion.
- "NC axis control selection" signal was turned ON while the auxiliary axis was in motion.

Remedy

- Turn ON the "NC axis control selection" signal to set the auxiliary axis to NC axis when issuing a command to the axis from machining program.
- When more than one axis have a same name, let only one of the axes work as an NC axis.
- Do not change the "NC axis control selection" signal while the axis is in motion.

M01 Illegal operation during tool tip control 0170

Details

Illegal operation was attempted during tool tip center control.

Remedy

- Change the operation mode to the previous one and restart.

M01 G114.n command illegal 1005

Details

G114.n has been commanded during the execution of G114.n.
G51.2 has been commanded when G51.2 spindle-spindle polygon machining mode has been already entered at another part system.

Remedy

- Command G113 to cancel the operation.
- Turn ON the "spindle synchronization cancel" signal to cancel the operation.
- Command G50.2 to cancel the operation.
- Turn ON the "spindle-spindle polygon cancel" signal to cancel the operation.

M01 Spindle in-use by synchro tap 1007

Details

The spindle is being used in synchronized tapping.

Remedy

- Cancel the synchronized tapping.

M01 SP-C ax ctrl runs independntly 1026

Details

C axis mode command has been issued for polygon machining spindle.
C axis mode command has been issued for synchronized tapping spindle.
Polygon command has been issued for synchronized tapping spindle.
Spindle is being used as spindle/C axis.

Remedy

- Cancel the C axis command.
- Cancel the polygon machining command.
- Cancel the C axis with servo OFF.

M01 Synchronization mismatch 1030

Details

Different M codes were each commanded as synchronization M code in each of the two part systems.
Synchronization with the "!" code was commanded in another part system during M code synchronization.
Synchronization with the M code was commanded in another part system during synchronization with the "!" code.

Remedy

- Correct the program so that the M codes match.
- Correct the program so that the same synchronization codes are commanded.

M01 Multiple C axes select invalid 1031

Details

The "C axis selection" signal has been changed when the multiple C axes selection is not available.
The selected axis by the "C axis selection" signal cannot be controlled for the multiple C axes selection.

Remedy

- Correct the parameter settings and program.

M01 Tap retract Sp select illegal 1032

Details

Tap retract has been executed with a different spindle selected. Cutting feed is in wait state until synchronization is completed.

Remedy

- Select the spindle for which tap cycle was halted before turning ON the "tap retract" signal.

M01 Sp-Sp polygon cut interlock 1033

Details

Cutting feed is in wait state until synchronization is completed.

Remedy

- Wait for the synchronization to end.

M01 Mixed sync ctrl prmtr illegal 1034

Details

There is a mistake in the settings of mixed control axis parameters (crsax [1] to [8]).
Mixed control was attempted within one and the same part system.
Any of the parameter settings is disabling mixed control.

Remedy

- Correct the parameter settings for the mixed control (cross axis control).

M01 Mixed sync ctrl disable modal 1035

Details

Mixed control (cross axis control) was commanded for a part system in which the mixed control (cross axis control) is disabled as shown below.

- During nose R compensation mode
- During pole coordinate interpolation mode
- During cylindrical interpolation mode
- During balance cut mode
- During fixed cycle machining mode
- During facing turret mirror image
- During constant surface speed control mode
- During hobbing mode
- During axis name switch

Remedy

- Correct the program.

M01 Synchro ctrl setting disable 1036

Details

"Synchronous control operation method" was set (with R2589) when the mode was not the C axis mode.
"Synchronous control operation method" was set (with R2589) in the zero point not set state.
Mirror image is disabled.
External mirror image or parameter mirror image was commanded during facing turret mirror image.

Remedy

- Set the contents of the R2589 register to "0".
- Correct the program and parameters.

M01 Synchro start/cancel disable 1037

Details

Synchronous control start/cancel command was issued when the start/cancel is disabled.

Remedy

- Correct the program and parameters.

M01 Move cmnd invld to synchro ax 1038

Details

A travel command was issued to a synchronous axis in synchronous control.

Remedy

- Correct the program.

M01 External spindle speed clamp speed zero 1039

Details

External spindle speed clamp signal has been turned ON while the clamp speed has not been set.

Remedy

- Set the external spindle speed clamp feedrate parameter.
- Turn OFF the external spindle speed clamp signal.

M01 No spindle speed clamp 1043

Details

The constant surface speed command (G96) was issued to the spindle which is not selected for the spindle speed clamp command (G92/G50) under Multiple spindle control II.

Remedy

Press the reset key and carry out the remedy below.
- Select the spindle before commanding G92/G50.

M01 Sp synchro phase calc illegal 1106

Details

Spindle synchronization phase alignment command was issued while the "phase shift calculation request" signal was ON.

Remedy

- Correct the program.
- Correct the sequence program.

M01 NC/PLC axis switch illegal 1250

Details

The following operation was performed to an axis which can be switched over between NC axis and PLC axis.
- PLC axis switchover signal was turned ON or OFF when it was prohibited to switch over the axis.

Remedy

Make sure the axis switchover status signal is OFF and change the ON/OFF of the axis switchover signal.

M80 POSITION ERROR

Details

An axis position is illegal.
An alarm is displayed (AL4 is output) and a block stop is applied on the machining program. When the block stop is not allowed in the thread cutting cycle and the like, the stop is applied at the next position where allowed.

Remedy

Carry out reset. Then confirm that the system starts the operation. If the alarm is displayed again, turn ON the emergency stop switch and turn the NC power OFF and ON.

M90 Parameter set mode

Details

The lock for setup parameters has been released. Setting the setup parameters is enabled while automatic start is disabled.

Remedy

Refer to the manual issued by the machine tool builder.

M91 INVALID MEASUR. 0002

Details

Data is over the range
The measurement result exceeds the tool data setting range.

Remedy

Correct the settings of "#2015 tlml- (Negative direction sensor of tool setter)" and "#2016 tlml+ (Positive direction sensor of tool setter or TLM standard length)".

M91 INVALID MEASUR. 0003

Details

No corresponding No.
No measurement tool No. has been set.
The registered No. is out of the specifications.

Remedy

Correct the measurement tool No.

M91 INVALID MEASUR. 0045

Details

Measurement axis illegal
Sensor has been turned ON while two or more axes are moving.

Remedy

Move a single axis when the sensor is contacted.

M91 INVALID MEASUR. 0046

Details

Measurement axis has not returned to reference position
Reference position return has not been executed on a measurement axis in an incremental system.

Remedy

Carry out the reference position return on the measurement axis before measuring the tool.

M91 INVALID MEASUR. 0089

Details

Sensor signal illegal ON
Sensor has already been ON when TLM mode is turned ON. The travel amount was so small that the tool contacted the sensor.

Remedy

All axes are interlocked when this alarm has occurred.
Turn the TLM mode OFF or use the interlock cancel signal to move the tool off the sensor.
Ensure at least 0.1mm for the movement to the sensor.

M91 INVALID MEASUR. 9000

Details

Speed at contact is below minimum
The tool has contacted the sensor at the lower speed than set in "#1508 TLM_Fmin (Minimum speed toward tool setter)".

Remedy

Correct the feed rate to move the tool to the sensor.

M91 INVALID MEASUR. 9001

Details

Speed at contact is over maximum
The tool has contacted the sensor at the higher speed than set in "#1509 TLM_Fmax (Maximum speed toward tool setter)".

Remedy

Correct the feed rate to move the tool to the sensor.

M91 INVALID MEASUR. 9002

Details

Change of compensation No. or sub-side selection during measurement
A compensation No. or sub-side valid signal state has been changed while a sensor signal is ON or a compensation amount is being written.

Remedy

Carry out the measurement again.

M91 INVALID MEASUR. 9003

Details

Error on response timing of sensor signal
A compensation No. has been changed at the same time as a sensor's response.

Remedy

Carry out the measurement again.

M92 IGNORE INT.LOCK

Details

Manual tool length measurement Interlock temporally canceled
"M01 Operation error 0005" and "M01 Operation error 0019", which occur at manual tool length measurement, are temporally canceled. When a tool has contacted a sensor and "M01 Operation error 0019" has occurred, tool escape is enabled by temporarily turning ON the interlock cancel request. This alarm notifies that the interlock is disabled in the meantime.

Remedy

After carrying out the tool escape from the sensor, turn OFF the interlock temporary cancel signal for manual tool length measurement.

Appendix 7.2 Stop Codes (T)

T01 Cycle start prohibit

Automatic start is not available in stop state.

T02 Feed hold

Feed hold is actuated during automatic operation for some reason.

T03 Block stop

Block stop is actuated during automatic operation for some reason.

(Note1) "T01"stop codes are displayed as "T01 Cycle start prohibit" with the error number.Error number is four digit number displayed after error name(start from 0101). "T01"stop codes are listed in ascending order in this manual.

(Note2) "T02"stop codes are displayed as "T02 Feed hold" with the error number.Error number is four digit number displayed after error name(start from 0201). "T02"stop codes are listed in ascending order in this manual.

(Note3) "T03"stop codes are displayed as "T03 Block stop" with the error number.Error number is four digit number displayed after error name(start from 0001). "T03"stop codes are listed in ascending order in this manual.

T01 Axis in motion 0101

Details

Automatic start is not possible as one of the axes is moving.

Remedy

- Try automatic start again after all axes have stopped.

T01 NC not ready 0102

Details

Automatic start is not possible as the NC is not ready.

Remedy

- Another alarm has occurred. Check the details and remedy.

T01 Reset signal ON 0103

Details

Automatic start is not possible as the "reset" signal has been input.

Remedy

- Turn OFF the "reset" signal.
- Check for any failure of the reset switch which has caused the switch's continuous ON.
- Correct the sequence program.

T01 Auto operation pause signal ON 0104

Details

The feed hold switch on the machine operation panel is ON (valid).

Remedy

- Correct the feed hold switch setting.
- The feed hold switch is B contact switch.
- Fix any broken wires in the feed hold signal line.
- Correct the sequence program.

T01 H/W stroke end axis exists 0105

Details

Automatic start is not possible as one of the axes is at the stroke end.

Remedy

- Manually move any axis whose end is at the stroke end.
- Check for any broken wires in the stroke end signal line.
- Check for any failure in the stroke end limit switch.

T01 S/W stroke end axis exists 0106

Details

Automatic start is not possible as one of the axes is at the stored stroke limit.

Remedy

- Move the axis manually.
- If the axis's end is not at the stroke end, check the parameters.

T01 No operation mode 0107

Details

The operation mode has not been selected.

Remedy

- Select automatic operation mode.
- Check for any broken wires in the signal line for automatic operation mode (memory, FTP, MDI).

T01 Operation mode duplicated 0108

Details

Two or more automatic operation modes have been selected.

Remedy

- Check for any short circuit in the mode (memory, FTP, MDI) selection signal line.
- Check for any failure in the switch.
- Correct the sequence program.

T01 Operation mode changed 0109

Details

The automatic operation mode has changed to another automatic operation mode.

Remedy

- Return to the original automatic operation mode, and execute automatic start.

T01 Tape search execution 0110

Details

Automatic start is not possible as tape search is being executed.

Remedy

- Wait for the tape search to be completed and then execute the automatic start.

T01 Cycle start prohibit 0111

Details

Automatic start is disabled because restart search is in execution.

Remedy

- Execute automatic start after the restart search is completed.

T01 CNC overheat 0113

Details

Automatic start is not possible because a thermal alarm (Z53 CNC overheat) has occurred.

Remedy

- Temperature of the control unit has exceeded the specified temperature.
- Take appropriate measures to cool the unit.

T01 Cycle st. prohibit(Host comm.) 0115

Details

Automatic start cannot is not possible because the NC is communicating with the host computer.

Remedy

- Wait for the communication with host computer to be ended and then execute the automatic start.

T01 Cycle st prohibit(Battery alm) 0116

Details

Automatic start is not possible because the voltage of the battery in the NC control unit has dropped.

Remedy

- Replace the battery of the NC control unit.
- Contact the service center.

T01 R-pnt offset value not set 0117

Details

Automatic operation is not possible because no reference position offset value has been set.

Remedy

- Perform the reference position initialization setting, then set "#2034 rfpofs(Distance-coded reference position detection offset)".

T01 Cycle start prohibit 0118

Details

Tool retract position signal OFF
The axis was moved from the tool retract position. Tool retract position reached signal is OFF.

Remedy

Once the axis has been moved off the tool retract position, resuming the program is not possible. Cancel the program by reset and then execute it from the start.

T01 In absolute position alarm 0138

Details

A start signal was input during an absolute position detection alarm.

Remedy

- Clear the absolute position detection alarm, and then input the start signal.

T01 In abs posn initial setting 0139

Details

A start signal was input during zero point initialization in the absolute position detection system.

Remedy

- Complete zero point initialization before inputting the start signal.

T01 Start during MDI operation at other part system disable 0141

Details

In multi-part system, a start signal was input for MDI mode while the MDI operation was being carried out in another part system.

Remedy

- End the other part system's operation before starting.

T01 Cycle start prohibit 0142

Details

In manual coordinate system setting
Automatic start is not allowed during the manual coordinate system setting.

Remedy

The system restarts after either of the manual coordinate system setting completion signal or the error end signal has been turned ON.

T01 In manual measurement 0143

Details

Automatic start is disabled because manual measurement is in execution.

Remedy

- Execute automatic start after the manual measurement is completed.

T01 Cycle start prohibit 0180

Details

Automatic start became disabled while servo auto turning is enabled.

Remedy

- Set "#1164 ATS" to "0" when the servo auto turning is not executed.

T01 Cycle start prohibit 0190

Details

Automatic start is not possible because the setting of setup parameters is enabled.

Remedy

- Refer to the manual issued by the machine tool builder.

T01 Cycle start prohibit 0191

Details

Automatic start was attempted while a file was being deleted/written.

Remedy

- Wait for the file to be deleted/written and then execute the automatic start.

T01 Cycle st. prohibit (Term exp'd) 0193

Details

Automatic start is not possible because the valid term has been expired.

Remedy

- Enter the decryption code and turn the power ON again.

T02 H/W stroke end axis exists 0201

Details

An axis is at the stroke end.

Remedy

- Manually move the axis away from the stroke end limit switch.
- Correct the machining program.

T02 S/W stroke end axis exists 0202

Details

An axis is at the stored stroke limit.

Remedy

- Manually move the axis.
- Correct the machining program.

T02 Reset signal ON 0203

Details

The reset has been entered.

Remedy

- The program execution position has returned to the start of the program. Execute automatic operation from the start of the machining program.

T02 Auto operation pause signal ON 0204

Details

The "feed hold" switch is ON.

Remedy

- Press the CYCLE START switch to resume the automatic operation.

T02 Operation mode changed 0205

Details

The operation mode has changed to another mode during automatic operation.

Remedy

- Return to the original automatic operation mode, and press the CYCLE START switch to resume the automatic operation.

T02 Acc/dec time cnst too large 0206

Details

The acceleration and deceleration time constants are too large. (This alarm occurs with the system alarm Z59.)

Remedy

- Set a larger value for "#1206 G1bF(Maximum speed)".
- Set a smaller value for "#1207 G1btL(Time constant)".
- Set a lower cutting speed.

T02 Abs posn detect alarm occurred 0215

Details

An absolute position detection alarm occurred.

Remedy

- Clear the absolute position detection alarm.

T02 Aux axis changeover error 0220

Details

A travel command was issued to an auxiliary axis.

Remedy

- Turn ON the "NC axis control selection" signal and press the CYCLE START switch to restart the automatic operation with.

T03 Single block stop signal ON 0301

Details

The SINGLE BLOCK switch on the machine operation panel is ON.
The SINGLE BLOCK or MACHINE LOCK switch changed.

Remedy

- Press the CYCLE START switch to resume the automatic operation.

T03 Block stop cmnd in user macro 0302

Details

A block stop command was issued in the user macro program.

Remedy

- Press the CYCLE START switch to resume the automatic operation.

T03 Operation mode changed 0303

Details

Automatic mode changed to another automatic mode.

Remedy

- Return to the original automatic operation mode, and press the CYCLE START switch to resume the automatic operation.

T03 MDI completed 0304

Details

MDI operation has ended the last block.

Remedy

- Set the MDI operation again, and press the CYCLE START switch to start the MDl operation.

T03 Block start interlock 0305

Details

The interlock signal, which locks the block start, is ON.

Remedy

- Correct the sequence program.

T03 Cutting blck start interlock 0306

Details

The interlock signal, which locks the block cutting start, is ON.

Remedy

- Correct the sequence program.

T03 Inclined Z offset change 0310

Details

The "inclined axis control: No Z axis compensation" signal has turned ON or OFF during the program operation.

Remedy

- Press the CYCLE START switch to resume the automatic operation.

T03 Aux axis changeover error 0330

Details

The "NC axis control selection" signal was turned OFF while a NC axis was traveling.

Remedy

- Turn the "NC axis control selection" signal ON and press the CYCLE START switch to resume the automatic operation.

T04 Collation stop 0401

Details

Collation stop occurred.

Remedy

- Execute the automatic start to resume the automatic operation.

T10 Fin wait 0200

Details

Waiting for completion of G11
G11 was issued while an axis was moving.

Remedy

Resume the operation after G11 has been completed.

T10 Fin wait 0ooo

Details

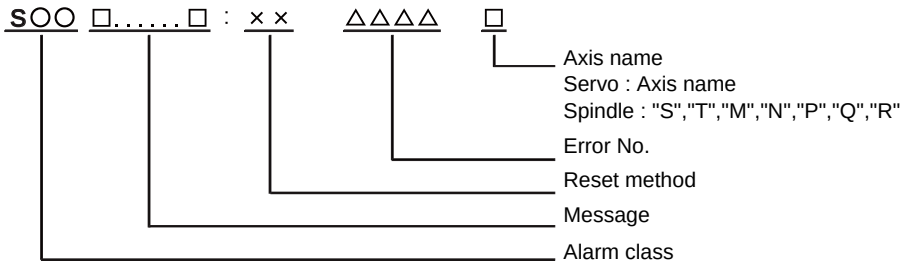
The following Nos. are shown during the operation of the corresponding completion wait factor. The numbers will disappear when the operation is completed.
The completion wait factor is indicated with four digits (in hexadecimal).
Display format of completion wait factor
0__ __ __
(a)(b)(c)
Each of the hexadecimal numbers (a), (b) and (c) indicates the following details.
(a)
bit0: In dwell execution
bit3: Unclamp signal wait (Note 1)
(b)
bit0: Waiting for spindle position to be looped
bit3: Door open (Note 2)
(c)
bit0: Waiting for MSTB completion
bit1: Waiting for rapid traverse deceleration
bit2: Waiting for cutting speed deceleration
bit3: Waiting for spindle orientation to com

(Note 1) This shows the wait state for the unclamp signal's ON/OFF for the index table indexing.

(Note 2) This shows the door open state caused by the door interlock function.

Appendix 7.3 Servo/Spindle Alarms (S)

Appendix 7.3.1 Servo Errors (S01/S03/S04)



Alarm class	Message	Reset method	Resetting methods
S01	Servo alarm	PR	After removing the cause of the alarm, reset the alarm by turning the NC power ON again.
S03	Servo alarm	NR	After removing the cause of the alarm, reset the alarm by inputting the NC RESET key.
S04	Servo alarm	AR	After removing the cause of the alarm, reset the alarm by turning the drive unit power ON again.

Error No. consists of four digits (0010 to). Servo alarms are explained in ascending order of the error No. The four digits on the left part of each alarm indicate the error No.
(Note) For the details of servo alarms, refer to your drive unit's instruction manual.

Drive unit alarms

0010 Insufficient voltage

Details

A drop of bus voltage was detected in main circuit.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0011 Axis selection error

Details

The axis selection rotary switch has been incorrectly set.
- Servo stop method: Initial error
- Spindle stop method: Initial error

0012 Memory error 1

Details

A hardware error was detected during the power ON self-check.
- Servo stop method: Initial error
- Spindle stop method: Initial error

0013 Software processing error 1

Details

An error was detected for the software execution state.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0016 Init mag pole pos detect err

Details

In the built-in motor which uses the absolute position detector, the servo ON has been set before the magnetic pole shift amount is set.
The magnetic pole position, detected in the initial magnetic pole position detection control, is not correctly set.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0017 A/D converter error

Details

A current feedback error was detected.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0018 Motor side dtc: Init commu err

Details

An error was detected in the initial communication with the motor side detector.
- Servo stop method: Initial error
- Spindle stop method: Initial error

0019 Detector commu err in syn cont

Details

An error of the shared detector on the machine side was detected on the secondary axis of the speed command synchronization control.
- Servo stop method: Dynamic stop

001A Machine side dtc: Init comu er

Details

An error was detected in the initial communication with the machine side detector.
- Servo stop method: Initial error
- Spindle stop method: Initial error

001B Machine side dtc: Error 1

Details

An error was detected by the detector connected to the machine side.
The error details are different according to the detector type.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- OSA105, OSA105-ET2, OSA166, OSA166-ET2(MITSUBISHI) Memory alarm
- OSA18() CPU alarm
- MDS-B-HR() Memory error
- AT343, AT543, AT545(Mitsutoyo) Initialization error
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) Initialization error
- MPRZ Series(MHI) Installation accuracy fault
- SR75, SR85, SR77, SR87, RU77(SONY) Laser diode error

[Detector alarm (Spindle drive unit)]
- TS5690, TS5691(MITSUBISHI) Memory error
- MDS-B-HR() Initialization error
- OSA18() CPU error
- EIB Series(HEIDENHAIN) Initialization error
- MPCI scale(MHI) Installation accuracy fault

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

001C Machine side dtc: Error 2

Details

An error was detected by the detector connected to the machine side.
The error details are different according to the detector type.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- OSA105, OSA105-ET2, OSA166, OSA166-ET2(MITSUBISHI) LED alarm
- AT343, AT543, AT545(Mitsutoyo) EEPROM error
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) EEPROM error
- SR75, SR85, SR77, SR87, RU77(SONY) System memory error

[Detector alarm (Spindle drive unit)]
- TS5690, TS5691(MITSUBISHI) Waveform error
- EIB Series(HEIDENHAIN) EEPROM error

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

001D Machine side dtc: Error 3

Details

An error was detected by the detector connected to the machine side.
The error details are different according to the detector type.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- OSA105, OSA105-ET2, OSA166, OSA166-ET2(MITSUBISHI) Data alarm
- OSA18() Data alarm
- MDS-B-HR() Data error
- AT343, AT543, AT545(Mitsutoyo) Photoelectric type, static capacity type data mismatch
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) Relative/ absolute position data mismatch
- MPRZ Series(MHI) Detection position deviance
- SR75, SR85, SR77, SR87, RU77(SONY) Encoder mismatch error

[Detector alarm (Spindle drive unit)]
- MDS-B-HR() Data error
- OSA18() Data error
- MPCI scale(MHI) Detection position deviance

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

001E Machine side dtc: Error 4

Details

An error was detected by the detector connected to the machine side.
The error details are different according to the detector type.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- AT343, AT543, AT545(Mitsutoyo) ROM/RAM error
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) ROM/RAM error
- MPRZ Series(MHI) Scale breaking

[Detector alarm (Spindle drive unit)]
- MPCI scale(MHI) Scale breaking

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

001F Machine side dtc: Commu error

Details

An error was detected in the communication with the machine side detector.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0021 Machine side dtc: No signal

Details

In the machine side detector, ABZ-phase feedback cannot be returned even when the motor moves.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0022 Detector data error

Details

An error was detected in the feedback data from the position detector.
- Servo stop method: Dynamic stop

0023 Excessive speed error

Details

The state that there is a difference between the actual speed and command speed continued for longer than the excessive speed deviation timer setting.
- Spindle stop method: Coast to a stop

0024 Grounding

Details

The motor power cable is in contact with FG (Frame Ground).
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0025 Absolute position data lost

Details

The absolute position data was lost in the detector.
- Servo stop method: Initial error

0026 Unused axis error

Details

In the multi-axis drive unit, there is an axis set to free, and the other axis detected a power module error.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0027 Machine side dtc: Error 5

Details

An error was detected by the detector connected to the machine side.
The error details are different according to the detector type.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- MDS-B-HR() Scale not connected
- AT343, AT543, AT545(Mitsutoyo) CPU error
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) CPU error
- MPRZ Series(MHI) Absolute value detection fault

[Detector alarm (Spindle drive unit)]
- MDS-B-HR() Connection error
- EIB Series(HEIDENHAIN) CPU error

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

0028 Machine side dtc: Error 6

Details

An error was detected by the detector connected to the machine side.
The error details are different according to the detector type.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- AT343, AT543, AT545(Mitsutoyo) Photoelectric type overspeed
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) Overspeed
- SR75, SR85, SR77, SR87, RU77(SONY) Over speed

[Detector alarm (Spindle drive unit)]
- TS5690, TS5691(MITSUBISHI) Overspeed
- EIB Series(HEIDENHAIN) Overspeed

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

0029 Machine side dtc: Error 7

Details

An error was detected by the detector connected to the machine side.
The error details are different according to the detector type.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- AT343, AT543, AT545(Mitsutoyo) Static capacity type error
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) Absolute position data error
- MPRZ Series(MHI) Gain fault
- SR75, SR85, SR77, SR87, RU77(SONY) Absolute position data error

[Detector alarm (Spindle drive unit)]
- MPCI scale(MHI) Gain fault

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

002A Machine side dtc: Error 8

Details

An error was detected by the detector connected to the machine side.
The error details are different according to the detector type.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- AT343, AT543, AT545(Mitsutoyo) Photoelectric type error
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) Relative position data error
- MPRZ Series(MHI) Phase fault
- SR75, SR85, SR77, SR87, RU77(SONY) Relative position data error

[Detector alarm (Spindle drive unit)]
- TS5690, TS5691(MITSUBISHI) Relative position data error
- EIB Series(HEIDENHAIN) Relative position data error
- MPCI scale(MHI) Phase fault

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

002B Motor side dtc: Error 1

Details

An error was detected by the detector connected to the motor side.
The error details are different according to the detector type.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- OSA105, OSA105-ET2, OSA166, OSA166-ET2(MITSUBISHI) Memory alarm
- OSA18() CPU alarm
- MDS-B-HR() Memory error
- AT343, AT543, AT545(Mitsutoyo) Initialization error
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) Initialization error
- MPRZ Series(MHI) Installation accuracy fault
- SR75, SR85, SR77, SR87, RU77(SONY) Laser diode error

[Detector alarm (Spindle drive unit)]
- TS5690, TS5691(MITSUBISHI) Memory error
- MDS-B-HR() Initialization error
- OSA18() CPU error
- EIB Series(HEIDENHAIN) Initialization error
- MPCI scale(MHI) Installation accuracy fault

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

002C Motor side dtc: Error 2

Details

An error was detected by the detector connected to the motor side.
The error details are different according to the detector type.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- OSA105, OSA105-ET2, OSA166, OSA166-ET2(MITSUBISHI) LED alarm
- AT343, AT543, AT545(Mitsutoyo) EEPROM error
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) EEPROM error
- SR75, SR85, SR77, SR87, RU77(SONY) System memory error

[Detector alarm (Spindle drive unit)]
- TS5690, TS5691(MITSUBISHI) Waveform error
- EIB Series(HEIDENHAIN) EEPROM error

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

002D Motor side dtc: Error 3

Details

An error was detected by the detector connected to the motor side.
The error details are different according to the detector type.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- OSA105, OSA105-ET2, OSA166, OSA166-ET2(MITSUBISHI) Data alarm
- OSA18() Data alarm
- MDS-B-HR() Data error
- AT343, AT543, AT545(Mitsutoyo) Photoelectric type, static capacity type data mismatch
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) Relative/ absolute position data mismatch
- MPRZ Series(MHI) Detection position deviance
- SR75, SR85, SR77, SR87, RU77(SONY) Encoder mismatch error

[Detector alarm (Spindle drive unit)]
- MDS-B-HR() Data error
- OSA18() Data error
- MPCI scale(MHI) Detection position deviance

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

002E Motor side dtc: Error 4

Details

An error was detected by the detector connected to the motor side.
The error details are different according to the detector type.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- AT343, AT543, AT545(Mitsutoyo) ROM/RAM error
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) ROM/RAM error
- MPRZ Series(MHI) Scale breaking

[Detector alarm (Spindle drive unit)]
- MPCI scale(MHI) Scale breaking

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

002F Motor side dtc: Commu error

Details

An error was detected in the communication with the motor side detector.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0030 Over regeneration

Details

Over-regeneration level exceeded 100%. The regenerative resistor is overloaded.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0031 Overspeed

Details

The motor speed exceeded the allowable speed.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

0032 Power module overcurrent

Details

The power module detected the overcurrent.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0033 Overvoltage

Details

The bus voltage in main circuit exceeded the allowable value.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0034 NC-DRV commu: CRC error

Details

The data received from the NC was outside the setting range.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

0035 NC command error

Details

The travel command data received from the NC was excessive.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

0036 NC-DRV commu: Commu error

Details

The communication with the NC was interrupted.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

0037 Initial parameter error

Details

An incorrect set value was detected among the parameters send from the NC at the power ON.
In the safety observation function, an error was detected in the relation between the safety speed and safety rotation number in the speed observation mode.
- Servo stop method: Initial error
- Spindle stop method: Initial error

0038 NC-DRV commu: Protocol error 1

Details

An error was detected in the communication frames received from the NC.
Or, removing an axis or changing an axis was performed in the synchronous control.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

0039 NC-DRV commu: Protocol error 2

Details

An error was detected in the axis data received from the NC.
Or, in changing an axis, the parameter setting of the synchronous control was applied when the axis was installed.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

003A Overcurrent

Details

Excessive motor drive current was detected.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

003B Power module overheat

Details

The power module detected an overheat.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

003C Regeneration circuit error

Details

An error was detected in the regenerative transistor or in the regenerative resistor.
- Servo stop method: Dynamic stop

003D Pw sply volt err acc/dec

Details

A motor control error during acceleration/deceleration, due to a power voltage failure, was detected.
- Servo stop method: Dynamic stop

003E Magnet pole pos detect err

Details

The magnetic pole position, detected in the magnetic pole position detection control, is not correctly detected.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0041 Feedback error 3

Details

Either a missed feedback pulse in the motor side detector or an error in the Z-phase was detected in the full closed loop system.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0042 Feedback error 1

Details

Either a missed feedback pulse in the position detection or an error in the Z-phase was detected. Or the distance-coded reference check error exceeded the allowable value when the distance-coded reference scale was used.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0043 Feedback error 2

Details

An excessive difference in feedback was detected between the machine side detector and the motor side detector.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0045 Fan stop

Details

An overheat of the power module was detected during the cooling fan stopping.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0046 Motor overheat

Details

Either the motor or the motor side detector detected an overheat.
Or, the thermistor signal receiving circuit of the linear motor or DD motor was disconnected.
Or, the thermistor signal receiving circuit was short-circuited.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

0048 Motor side dtc: Error 5

Details

An error was detected by the detector connected to the main side.
The error details are different according to the connected detector.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- MDS-B-HR() Scale not connected
- AT343, AT543, AT545(Mitsutoyo) CPU error
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) CPU error
- MPRZ Series(MHI) Absolute value detection fault

[Detector alarm (Spindle drive unit)]
- MDS-B-HR() Connection error
- EIB Series(HEIDENHAIN) CPU error

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

0049 Motor side dtc: Error 6

Details

An error was detected by the detector connected to the main side.
The error details are different according to the connected detector.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- AT343, AT543, AT545(Mitsutoyo) Photoelectric type overspeed
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) Overspeed
- SR75, SR85, SR77, SR87, RU77(SONY) Over speed

[Detector alarm (Spindle drive unit)]
- TS5690, TS5691(MITSUBISHI) Overspeed
- EIB Series(HEIDENHAIN) Overspeed

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

004A Motor side dtc: Error 7

Details

An error was detected by the detector connected to the main side.
The error details are different according to the connected detector.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- AT343, AT543, AT545(Mitsutoyo) Static capacity type error
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) Absolute position data error
- MPRZ Series(MHI) Gain fault
- SR75, SR85, SR77, SR87, RU77(SONY) Absolute position data error

[Detector alarm (Spindle drive unit)]
- MPCI scale(MHI) Gain fault

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

004B Motor side dtc: Error 8

Details

An error was detected by the detector connected to the main side.
The error details are different according to the connected detector.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

[Detector alarm (Servo drive unit)]
- AT343, AT543, AT545(Mitsutoyo) Photoelectric type error
- LC193M, LC493M, RCN223M, RCN227M, RCN727M, RCN827M, EIB Series(HEIDENHAIN) Relative position data error
- MPRZ Series(MHI) Phase fault
- SR75, SR85, SR77, SR87, RU77(SONY) Relative position data error

[Detector alarm (Spindle drive unit)]
- TS5690, TS5691(MITSUBISHI) Relative position data error
- EIB Series(HEIDENHAIN) Relative position data error
- MPCI scale(MHI) Phase fault

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

004C Current err mag pole estim

Details

Current detection failed at the initial magnetic pole estimation.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

004E NC command mode error

Details

An error was detected in the control mode send from the NC.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

004F Instantaneous power interrupt

Details

The control power supply has been shut down for 50ms or more.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

0050 Overload 1

Details

Overload detection level became 100% or more. The motor or the drive unit is overloaded.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

0051 Overload 2

Details

In a servo system, current command of 95% or more of the unit’s max. current was given continuously for 1 second or longer. In a spindle system, current command of 95% or more of the motor’s max. current was given continuously for 1 second or longer.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

0052 Excessive error 1

Details

A position tracking error during servo ON was excessive.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

0053 Excessive error 2

Details

A position tracking error during servo OFF was excessive.
- Servo stop method: Dynamic stop

0054 Excessive error 3

Details

There was no motor current feedback when the alarm "Excessive error 1" was detected.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0056 Commanded speed error

Details

In the C-axis control mode, excessive speed error was detected.
- Spindle stop method: Deceleration stop enabled

0058 Collision detection 1: G0

Details

A disturbance torque exceeded the allowable value in rapid traverse modal (G0).
- Servo stop method: Maximum capacity deceleration stop

0059 Collision detection 1: G1

Details

A disturbance torque exceeded the allowable value in the cutting feed modal (G1).
- Servo stop method: Maximum capacity deceleration stop

005A Collision detection 2

Details

A current command with the maximum drive unit current value was detected.
- Servo stop method: Maximum capacity deceleration stop

005B Sfty obsrvation: Cmd spd err

Details

A commanded speed exceeding the safe speed was detected in the safety observation mode.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

005D Sfty obsrvation: Door stat err

Details

The door state signal input in the NC does not coincide with the door state signal input in the drive unit in the safety observation mode. Otherwise, door open state was detected in normal mode.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

005E Sfty obsrvation: FB speed err

Details

A motor speed exceeding the safe speed was detected in the safety observation mode.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

005F External contactor error

Details

A contact of the external contactor is welding.
- Servo stop method: Deceleration stop enabled
- Spindle stop method: Deceleration stop enabled

0080 Motor side dtc: cable err

Details

The cable type of the motor side detector cable is for rectangular wave signal.
- Servo stop method: Initial error

0081 Machine side dtc: cable err

Details

The cable type of the machine side detector cable does not coincide with the detector type which is set by the parameter.
- Servo stop method: Initial error

0087 Drive unit communication error

Details

The communication frame between drive units was aborted.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

0088 Watchdog

Details

The drive unit does not operate correctly.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

008A Drivers commu data error 1

Details

The communication data 1 between drivers exceeded the tolerable value in the communication between drive units.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

008B Drivers commu data error 2

Details

The communication data 2 between drivers exceeded the tolerable value in the communication between drive units.
- Servo stop method: Dynamic stop
- Spindle stop method: Coast to a stop

Power supply alarms

0061 Pw sply: Pwr module overcurnt

Details

Overcurrent protection function in the power module has started its operation.
- LED display: 1

0062 Pw sply: Frequency error

Details

The input power supply frequency increased above the specification range.
- LED display: 2

0066 Process error

Details

An error occurred in the process cycle.
- LED display: 6

0067 Pw sply: Phase interruption

Details

An open-phase condition was detected in input power supply circuit.
- LED display: 7

0068 Pw sply: Watchdog

Details

The system does not operate correctly.
- LED display: 8

0069 Pw sply: Grounding

Details

The motor power cable is in contact with FG (Frame Ground).
- LED display: 9

006A Pw sply: Ext contactor weld

Details

A contact of the external contactor is welding.
- LED display: A

006B Pw sply: Rush relay welding

Details

An error was detected in the rush circuit.
- LED display: b

006C Pw sply: Main circuit error

Details

An error was detected in charging operation of the main circuit capacitor.
- LED display: C

006D Pw sply: Parameter error

Details

An error was detected in the parameter sent from the drive unit.
- LED display: d

006E Pw sply: Memory error

Details

An error was detected in the internal memory.
An error was detected in the A/D converter.
An error was detected in the unit identification.
- LED display: E

006F Power supply error

Details

No power supply is connected to the drive unit, or a communication error was detected.
- LED display: F

0070 Pw sply: External EMG stop err

Details

A mismatch of the external emergency stop input and NC emergency stop input continued for 30 seconds.
- LED display: G

0071 Pw sply: Instant pwr interrupt

Details

The power was momentarily interrupted.
- LED display: H

0072 Pw sply: Fan stop

Details

A cooling fan built in the power supply unit stopped, and overheat occurred in the power module.
- LED display: I

0073 Pw sply: Over regeneration

Details

Over-regeneration detection level became over 100%. The regenerative resistor is overloaded. This alarm cannot be reset for 15 min from the occurrence to protect the regeneration resistor. Leave the drive system energized for more than 15 min, then turn the power ON to reset the alarm.
- LED display: J

0075 Pw sply: Overvoltage

Details

L+ and L- bus voltage in main circuit exceeded the allowable value. As the voltage between L+ and L- is high immediately after this alarm, another alarm may occur if this alarm is reset in a short time. Wait more than 5 min before resetting so that the voltage drops.
- LED display: L

0076 Pw sply: Ext EMG stop set err

Details

The rotary switch setting of external emergency stop is not correct, or a wrong external emergency stop signal is input.
- LED display: M

0077 Pw sply: Power module overheat

Details

Thermal protection function in the power module has started its operation.
- LED display: n

Appendix 7.3.2 Initial Parameter Errors (S02)

S02 Initial parameter error : PR 2201-2264 (Axis name)

Details

The servo parameter setting data is illegal.
The alarm No. is the No. of the servo parameter where the error occurred.

Remedy

Check the descriptions for the appropriate servo parameters and correct them.

S02 Initial parameter error : PR 2301 (Axis name)

Details

The number of constants to be used in the following functions is too large:

- Electronic gears
- Position loop gain
- Speed feedback conversion

Remedy

Check that all the related parameters are specified correctly.
sv001:PC1, sv002:PC2, sv003:PGN1
sv018:PIT, sv019:RNG1, sv020:RNG2

S02 Initial parameter error : PR 2302 (Axis name)

Details

When high-speed serial incremental detector (OSE104, OSE105) is connected, parameters for absolute position are set to ON.
Set the parameters for absolute position detection to OFF.
To detect an absolute position, replace the incremental specification detector with an absolute position detector.

Remedy

Check that all the related parameters are specified correctly.
sv017:SPEC, sv025:MTYP

S02 Initial parameter error : PR 2303 (Axis name)

Details

No servo option is found.
The closed loop (including the ball screw-
end detector) or dual feedback control is an optional function.

Remedy

Check that all the related parameters are specified correctly.
sv025:MTYP/pen
sv017:SPEC/dfbx

S02 Initial parameter error : PR 2304 (Axis name)

Details

No servo option is found.
The SHG control is an optional function.

Remedy

Check that all the related parameters are specified correctly.
sv057:SHGC
sv058:SHGCsp

S02 Initial parameter error : PR 2305 (Axis name)

Details

No servo option is found.
The adaptive filtering is an optional function.

Remedy

Check that all the related parameters are specified correctly.
sv027:SSF1/aflt

S02 Initial parameter error : PR 13001-13256 (Axis name)

Details

Parameter error
The spindle parameter setting data is illegal.
The alarm No. is the No. of the spindle parameter where the error occurred.

Remedy

Check the descriptions for the appropriate spindle parameters and correct them.
Refer to Alarm No.37 in Spindle Drive Maintenance Manual.

Appendix 7.3.3 Parameter Errors (S51)

S51 Parameter error 2201-2264 (Axis name)

Details

Servo parameter setting data is illegal.
The alarm No. is the No. of the servo parameter where the warning occurred.

Remedy

Check the descriptions for the appropriate servo parameters and correct them.

S51 Parameter error 13001-13256 (Axis name)

Details

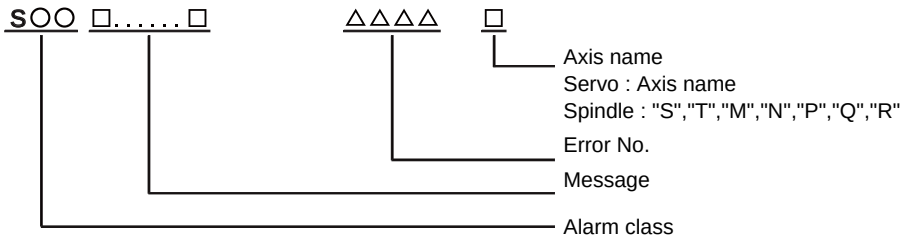
Spindle parameter setting data is illegal.
The alarm No. is the No. of the spindle parameter where the warning occurred.

Remedy

Check the descriptions for the appropriate spindle parameters and correct them.
Refer to Spindle Drive Maintenance Manual.

Appendix 7.3.4 Servo Warnings (S52)

Servo warning is displayed in the following format.



Alarm class	Message
S52	Servo warning

Error No. consists of four digits (0096 to). Servo warnings are explained in ascending order of the error No.
The four digits on the left part of each warning indicate the error No.
(Note) For the details of servo warnings, refer to your drive unit's instruction manual.

Drive unit warnings

0096 Scale feedback error

Details

An excessive difference in feedback amount was detected between the main side detector and the MPI scale in MPI scale absolute position detection system.
- Reset method: Automatically reset once the cause of the warning is removed.

0097 Scale offset error

Details

An error was detected in the offset data that is read at the NC power-ON in MPI scale absolute position detection system.

009B Detec cnv: Mag pole shift warn

Details

The difference between the magnetic pole position after the phase Z has been passed (magnetic pole shift amount:SV028) and the initially detected position is excessive in the built-in motor's incremental control system. The magnetic pole is controlled by the initial detection value.
- Reset method: Automatically reset once the cause of the warning is removed.

009E Abs pos dtc: Rev count error

Details

An error was detected in the revolution counter data of the absolute position detector. The accuracy of absolute position is not guaranteed.
- Reset method: Automatically reset once the cause of the warning is removed.

009F Battery voltage drop

Details

The battery voltage to be supplied to the absolute position detector is dropping.
- Reset method: Automatically reset once the cause of the warning is removed.

00A3 DisCode ref chk: init s-up war

Details

This warning is detected until the axis reaches the reference position during the initial setup of the distance-coded reference check function. This warning turns OFF after the axis has reached the position, thus set the value displayed on the drive monitor to the parameter.
- Reset method: Automatically reset once the cause of the warning is removed.

00A6 Fan stop warning

Details

A cooling fan in the drive unit stopped.
- Reset method: Automatically reset once the cause of the warning is removed.

00E0 Over regeneration warning

Details

Over-regeneration detection level exceeded 80%.
- Reset method: Automatically reset once the cause of the warning is removed.

00E1 Overload warning

Details

A level of 80% of the Overload 1 alarm state was detected.
- Reset method: Automatically reset once the cause of the warning is removed.

00E4 Set parameter warning

Details

An incorrect set value was detected among the parameters send from the NC in the normal operation.
- Reset method: Automatically reset once the cause of the warning is removed.

00E6 Control axis detach warning

Details

A control axis is being detached. (State display)
- Reset method: Automatically reset once the cause of the warning is removed.

00E7 In NC emergency stop state

Details

In NC emergency stop. (State display)
- Stop method: Deceleration stop enabled
- Reset method: Automatically reset once the cause of the warning is removed.

00E8-00EF Power supply warning

Details

The power supply unit detected a warning. The error details are different according to the connected power supply unit.
Refer to "Power supply warning".
- Stop method: - (EA: Deceleration stop enabled)
- Reset method: Automatically reset once the cause of the warning is removed.

Power supply warnings

00E9 Instant pwr interrupt warning

Details

The power was momentarily interrupted.
- LED display: P

00EA In external EMG stop state

Details

External emergency stop signal was input.
- Reset method: Automatically reset once the cause of the warning is removed.
- LED display: q

00EB Pw sply: Over regenerat warn

Details

Over-regeneration detection level exceeded 80%.
- Reset method: Automatically reset once the cause of the warning is removed.
- LED display: r

00EE Power supply: Fan stop warning

Details

A cooling fan built in the power supply unit stopped.
- Reset method: Automatically reset once the cause of the warning is removed.
- LED display: u

Appendix 7.4 MCP Alarms (Y)

(Note1) "Y02" alarms are displayed as "Y02 System alarm"with the error number. Error number is the four digit number displayed displayed after error name(start from 0050). "Y02"alarms are listed in ascending order in this manual.

(Note2) "Y20" alarms are displayed as "Y20 Safety observation alarm"with the error number. Error number is the four digit number displayed displayed after error name(start from 0001). "Y20"alarms are listed in ascending order in this manual.

(Note3) "Y21" warnings are displayed as "Y21 Safety observation warning"with the error number. Error number is the four digit number displayed displayed after error name(start from 0001). "Y21"warnings are listed in ascending order in this manual.

(Note4) "Y51" warnings are displayed as "Y51 Parameter error"with the error number. Error number is the four digit number displayed displayed after error name(start from 0001). "Y51"warnings are listed in ascending order in this manual.

Y02 System alm: Process time over 0050

Details

System alarm: Process time is over.

Remedy

The software or hardware may be damaged.
Contact the service center.

(Note 1) When two or more "Y02 System alarms" occur at the same time, only the alarm which has occurred first is displayed.

(Note 2) Station No. always shows "0" in the alarm details, because C70 has only one communication channel.

Y02 SV commu er: CRC error 1 0051 0000

Details

A communication error has occurred between controller and drive unit.

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more "Y02 System alarms" occur at the same time, only the alarm which has occurred first is displayed.

Y02 SV commu er: CRC error 2 0051 0001

Details

A communication error has occurred between controller and drive unit.

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more "Y02 System alarms" occur at the same time, only the alarm which has occurred first is displayed.

Y02 SV commu er: Recv timing err 0051 0002

Details

A communication error has occurred between controller and drive unit.

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more "Y02 System alarms" occur at the same time, only the alarm which has occurred first is displayed.

Y02 Servo communication error: Data ID error 0051 0x03

Details

A communication error has occurred between CNC and drive unit.
The axis ID transferred from the drive unit has changed after initial communication.
x: Drive unit rotary switch No. (0 or later)

Remedy

- Take measures against noise.

(Note 1) When two or more "Y02 System alarms 0051" alarms occur at the same time, only the alarm which occurs first is displayed. Therefore, the display precedence will be as follows: 0006, 0x04, 0005, 0x20, then 0x03.

(Note 2) Station No. always shows "0" in the alarm details, because C70 has only one communication channel.

(Note 3) If the error is not cleared with the measures above, the drive unit may have a fault. Take a note of the 7-segment LED contents of each drive unit and contact service center.

Y02 Servo communication error : Number of received frames 0051 0x04

Details

A communication error has occurred between CNC and drive unit.
The number of received frames was inconsistent in four consecutive communication cycles.
x: The number of received frames subtracted by one (0 or later)

Remedy

- Check for any duplication of rotary switch settings on drive units connected with other drive units.
- Confirm that the number of axes does not exceed the number designated by CNC.
- Check for any failure of the communication cable connectors.
- Check for any failure of the communication cables.
- Take measures against noise.

(Note 1) When two or more "Y02 System alarms 0051" alarms occur at the same time, only the alarm which occurs first is displayed. Therefore, the display precedence will be as follows: 0006, 0x04, 0005, 0x20, then 0x03.

(Note 2) Station No. always shows "0" in the alarm details, because C70 has only one communication channel.

(Note 3) If the error is not cleared with the measures above, the drive unit may have a fault. Take a note of the 7-segment LED contents of each drive unit and contact service center.

Y02 SV commu er: Commu error 0051 0005

Details

A communication error has occurred between controller and drive unit.
Non-specified communication errors occurred in four consecutive communication cycles.

Remedy

- Check for any failure of the communication cable connectors.
- Check for any failure of the communication cables.
- Take measures against noise.

(Note 1) When two or more "Y02 System alarms 0051" alarms occur at the same time, only the alarm which occurs first is displayed. Therefore, the display precedence will be as follows: 0006, 0x04, 0005, 0x20, then 0x03.

(Note 2) Station No. always shows "0" in the alarm details, because C70 has only one communication channel.

(Note 3) If the error is not cleared with the measures above, the drive unit may have a fault. Take a note of the 7-segment LED contents of each drive unit and contact service center.

Y02 Servo communication error : Connect error 0051 0006

Details

A communication error has occurred between CNC and drive unit.
CRC errors, overrun errors or short frame errors occurred in four consecutive communication cycles.
Otherwise, 250 bytes of "0"/"1" data was received.

Remedy

- Confirm that the rotary switch setting on the drive unit connected with CNC does not duplicate with any other.
- Confirm that the number of axes does not exceed the number designated by CNC.
- Check for any failure of the communication cable connectors.
- Check for any failure of the communication cables.
- Take measures against noise.

(Note 1) When two or more "Y02 System alarms 0051" alarms occur at the same time, only the alarm which occurs first is displayed. Therefore, the display precedence will be as follows: 0006, 0x04, 0005, 0x20, then 0x03.

(Note 2) Station No. always shows "0" in the alarm details, because C70 has only one communication channel.

(Note 3) If the error is not cleared with the measures above, the drive unit may have a fault. Take a note of the 7-segment LED contents of each drive unit and contact service center.

Y02 Control axis No. error: Connection error 0051 0x07

Details

A communication error has occurred between CNC and drive unit.
x: Drive unit rotary switch No. (0 or later)

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors.
- Check for any failure of the communication cables.
- The drive unit may have a fault. Take a note of the 7-segment LED contents of each drive unit and contact service center.
- Update the drive unit software version.

(Note 1) When two or more "Y02 System alarms 0051" alarms occur at the same time, only the alarm which occurs first is displayed. Therefore, the display precedence will be as follows: 0006, 0x04, 0005, 0x20, then 0x03.

(Note 2) Station No. always shows "0" in the alarm details, because C70 has only one communication channel.

(Note 3) If the error is not cleared with the measures above, the drive unit may have a fault. Take a note of the 7-segment LED contents of each drive unit and contact service center.

Y02 Servo communication error : Initial communication error 0051 0x20

Details

A communication error has occurred between CNC and drive unit.
A drive unit stopped due to transition failure from initial communication to runtime.
x: Drive unit rotary switch No. (0 or later)

Remedy

- Confirm that "the spindle drive unit rotary switch No. + 1" does not duplicate with the lower two digits in "#1021 mcp_no (Drive unit I/F channel No. (servo)".
- Confirm that "the servo drive unit rotary switch No. + 1" does not duplicate with the lower two digits in "#3031 smcp_no (Drive unit I/F channel No. (spindle)".

(Note 1) When two or more "Y02 System alarms 0051" alarms occur at the same time, only the alarm which occurs first is displayed. Therefore, the display precedence will be as follows: 0006, 0x04, 0005, 0x20, then 0x03.

(Note 2) Station No. always shows "0" in the alarm details, because C70 has only one communication channel.

(Note 3) If the error is not cleared with the measures above, the drive unit may have a fault. Take a note of the 7-segment LED contents of each drive unit and contact service center.

Y02 SV commu er: Node detect error 0051 xy30

Details

A communication error has occurred between controller and drive unit.
No response from drive unit to the request from NC when setting network configuration.
x: Channel No. (from 0)
y: Station No. (from 0)

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more "Y02 System alarms" occur at the same time, only the alarm which has occurred first is displayed.

Y02 SV commu er: Commu not support 0051 xy31

Details

A communication error has occurred between controller and drive unit.
Drive unit's software version doesn't support the communication mode that the controller requires.
x: Channel No. (from 0)
y: Station No. (from 0)

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more "Y02 System alarms" occur at the same time, only the alarm which has occurred first is displayed.

Y02 System alarm 0052 0001

Details

Transfer to buffer is not properly done in servo communication.

Remedy

Software/ hardware may have a fault.
Contact service center.

(Note 1) When two or more "Y02 System alarms 0051" alarms occur at the same time, only the alarm which occurs first is displayed. Therefore, the display precedence will be as follows: 0006, 0x04, 0005, 0x20, then 0x03.

(Note 2) Station No. always shows "0" in the alarm details, because C70 has only one communication channel.

(Note 3) If the error is not cleared with the measures above, the drive unit may have a fault. Take a note of the 7-segment LED contents of each drive unit and contact service center.

Y03 Amp. Unequipped (Axis name)

Details

The drive unit is not correctly connected.
Alphabet (axis name): Servo axis drive unit not mounted
1 to 8: PLC axis drive unit not mounted
S: No.1 spindle drive unit not mounted
T: No.2 spindle drive unit not mounted
M: No.3 spindle drive unit not mounted
N: No.4 spindle drive unit not mounted
P: No.5 spindle drive unit not mounted
Q: No.6 spindle drive unit not mounted
R: No.7 spindle drive unit not mounted

Remedy

Check the drive unit mounting state.
- Check the end of the cable wiring.
- Check the cable for broken wires.
- Check the connector insertion.
- The drive unit input power has not been ON.
- The drive unit axis No. switch is illegal.
- Turn down the DIP switch on the drive unit of the axis corresponding to the error No. (axis name).
- Check for any duplication of rotary switch settings on a drive unit. The LED will indicate "11" if the drive unit has duplicate setting.
- Correct the "#1002 axisno (Number of axes)" and "#1039 spinno (Number of spindles)" settings.
- Before setup debugging, confirm that the "#2018 no_srv (Operation with no servo control)" is set to "1" and "#3024 sout (Spindle connection)" is set to "0" for any axis to which the drive unit is not connected. (Note that the normal setting is "0" in "#2018 no_srv" and "1" in "#3024 sout".)

(Note 1) This alarm is displayed for each part system. Therefore, the error No. is not displayed unless the alarm occurs in the displayed part system.

(Note 2) When the alarm occurs on a servo axis, the error No. indicates the axis name set in "#1013 axname (Axis name)". When the alarm occurs on a PLC axis or the spindle, the error No. display is fixed by the order of CNC control axes.

Y05 Initial parameter error

Details

There is a problem in the value set for the number of axes or the number of part systems.

Remedy

Correct the value set for the following corresponding parameters:
“#1001 SYS_ON (System validation setup)”,
“#1002 axisno (Number of axes)”,
“#1039 spinno (Number of spindles)”,
“#2187 chgPLCax (PLC axis switchover No.)”, etc.

Y06 mcp_no setting error

Details

- MCP Nos. of the servo/spindle drive units are not continuous.
- There is a duplicate setting.
- Axis No. is out of the setting range.
- Channel No. is out of the setting range.

Remedy

Check the values set for the following parameters.
- "#1021 mcp_no (Drive unit I/F channel No. (servo))"
- "#3031 smcp_no (Drive unit I/F channel No. (spindle))"

(Note 1) "#1021 mcp_no (Drive unit I/F channel No. (servo))" and "#3031 smcp_no (Drive unit I/F channel No. (spindle))" must be continuous.

(Note 2) This alarm is displayed taking precedence over the alarms "Y02 System alarm", "Y07 Too many axes connected" and "Y09 Too many axisno connected".

Y07 Too many axes connected 0000 - Exceeded number of axes

Details

The number of axes connected to CNC has exceeded the maximum number of connectable axes.
The exceeded number of axes per channel is displayed as alarm No.

Remedy

- Correct the "#3024 sout (Spindle connection)" setting for the spindle to which the drive unit is connected.
- Turn up the DIP switch for any unused axis in the multi-axes drive unit.
- Correct the "#1002 axisno (Number of axes)" and "#1039 spinno (Number of spindles)" settings.
- Disconnect the connected axes as many as shown in the alarm No. so that the number of connected axes will not exceed the maximum number of connectable axes.

(Note 1) This alarm occurs when the number of the connected axes exceeded the maximum number of the connectable axes assumed in the system.

(Note 2) This alarm indicates a communication timeout state of drive unit (with "AA" in the drive unit LED display) as the initial communication is not executed.

(Note 3) If this alarm occurs, the alarm "Y03 Message: Drive unit unequipped" will not occur.

(Note 4) This alarm is displayed taking precedence over the alarm "Y02 System alarm" and "Y09 Too many axisno connected".

Y08 Too many drive units connected 0000 - 00FF

Details

The number of drive units connected to each channel exceeds 8.
The exceeded number of drive units per channel is displayed as alarm No.

Remedy

Remove drive units from the channel whose alarm No. is other than "0" for the number displayed as the alarm No. Keep the number of connected drive units to 8 or less.

(Note 1) The drive unit is not counted when all the axes connected to it are invalid.

(Note 2) If this alarm occurs, the alarm "Y03 Message: Drive unit unequipped" will not occur.

(Note 3) The alarm "Y07 Too many axes connected" and "Y09 Too many axisno connected" are displayed taking precedence over this alarm.

Y09 Too many axisno connected Exceeded number of axes

Details

The No. of the axis (drive unit's rotary switch No.) connected to CNC is bigger than the maximum number of connectable axes.

Remedy

Be sure to turn down the rightmost DIP switch on each drive unit.

(Note 1) This alarm occurs when the number of the connected axes exceeded the maximum number of the connectable axes assumed in the system.

(Note 2) If this alarm occurs, drive unit indicates a communication timeout (with "AA" in the LED display) as the initial communication is not executed.

(Note 3) If this alarm occurs, the alarm "Y03 Message: Drive unit unequipped" will not occur.

(Note 4) This alarm is displayed taking precedence over the alarm "Y02 System alarm".

Y11 Node Detect Err 8002-8300 0x00

Details

Drive unit does not respond to the request from CNC when the CNC is turned ON.
Error No. shows the No. of communication phase at which the response stopped.
x: Station No. with the error (0 or later)

Remedy

The communication error may be caused by the drive unit software version that does not correspond to the CNC software version. Check the drive unit software version.
This alarm is canceled after the CNC restarts.
When the alarm is not canceled, write down the alarm No. and the software version of each drive unit, then contact service center.

Y12 No commu. with axis drv unit

Details

Although the high-speed synchronous tapping option is valid, the connected drive unit doesn't support the option.

Remedy

Replace the drive unit with that supports the option.

Y13 No commu. with sp drv unit

Details

Although the high-speed synchronous tapping option is valid, the connected drive unit doesn't support the option.

Remedy

Replace the drive unit with that supports the option.

Y14 VIR.AX AMP EQU. (Axis name)

Details

A drive unit is connected to a hypothetical axis (with "1" in "#2116 v_axis (Hypothetical axis)").

Remedy

- Disconnect the drive unit from the hypothetical axis.
- Correct the "#1021 mcp_no (Drive unit I/F channel No. (servo))" and servo drive unit rotary switch settings.

Y20 Parameter compare error 0001 (Axis name)

Details

The speed monitoring parameter in the NC does not correspond to the parameter transmitted to the drive unit.
The name of the axis with an error is displayed.

Remedy

The NC or the servo drive unit may be damaged.
Contact the service center.

Y20 Sfty obsrvation: Cmd spd err 0002 (Axis name)

Details

The speed exceeding the speed set with the parameter was commanded during the speed monitoring mode.
The name of the axis with an error is displayed.

Remedy

Check the speed monitoring parameter and the sequence program.
Restart the NC.

Y20 Sfty obsrvation: FB pos err 0003 (Axis name)

Details

The commanded position, transmitted to the servo drive unit from NC, is totally different from the feedback position received from the servo drive unit during the speed monitoring mode.
The name of the axis with an error is displayed.

Remedy

The NC or the servo drive unit may be damaged.
Contact the service center.

Y20 Sfty obsrvation: FB speed err 0004 (Axis name)

Details

Actual rotation speed of the motor is exceeding the speed that has been set with speed monitoring parameter during the speed monitoring mode.
The name of the axis with an error is displayed.

Remedy

Correct the speed observation parameter and the sequence program.
Restart the NC.

Y20 Door signal: Input mismatch 0005 Door No.

Details

Door state signals on the NC side and the drive side do not match. It may be caused by the followings:

- Cable disconnection
- Damaged door switch
- Damaged NC or servo drive unit

Remedy

Check the cable.
Check the door switch.
Restart the NC.

Y20 No speed observation mode in door open 0006 Door No.

Details

The door open state was detected when the speed monitoring mode was invalid.
The causes may be same as the ones for 0005 (Door signal: Input mismatch). Also the sequence program may not be correct.

Remedy

Correct the sequence program.
Restart the NC.

Y20 Speed obsv: Para incompatible 0007 (Axis name)

Details

Two speed monitoring parameters are not matched at the rising edge of the "speed monitor mode" signal.
The name of the axis with an error is displayed.

Remedy

Correct the relevant parameters so that the two speed monitoring parameters match.
Restart the NC.

Y20 Contactor welding detected 0008 Contactor No.

Details

Contactor welding was detected.
Some contactors take a while to be shutdown after the servo ready is turned OFF, and the servo ready was turned ON in the meantime.
The contactor showing unusual move will be indicated as a bit.

bit0 : MC_dp1

bit1 : MC_dp2

Remedy

- Make sure that contactor's auxiliary B contact signal is output correctly to the device set on "#1330 MC_dp1(Contactor weld detection device 1)" and "#1331 MC_dp2(Contactor weld detection device 2)".
- If welding, replace the contactor.
- Restart the NC.

Y20 No spec: Safety observation 0009

Details

“#2313 SV113 SSF8/bitF (Servo function selection 8)”, “#13229 SP229 SFNC9/bitF (Spindle function 9)” and “#21125 SSU_num (Number of dual signal modules)” are set for a system with no safety observation option.

Remedy

Disable “#2313 SV113 SSF8/bitF (Servo function selection 8)” and “#13229 SP229 SFNC9/bitF (Spindle function 9)”.
Set “#21125 SSU_num (Number of dual signal modules)” to "0". Restart the NC.

Y20 Safety OBS ERR 0012 (Contactor data)

Details

Contactor's operation is not following the NC's commands.
The contactor showing unusual move will be indicated as a bit.

bit0 : MC_dp1

bit1 : MC_dp2

(Example 1) Operation error only in MC_dp1 : 0001

(Example 2) Operation error in both MC_dp1 and MC_dp2 : 0003

Remedy

- Check if the contactor's auxiliary b contact signal is correctly output to the device set in "#1330 MC_dp1" and "#1331 MC_dp2".
- Check the wiring for contactor shutoff.
- Check for contactor's welding.
- Turn the controller ON again.

Y20 Dual signal: compare error 0020 (Device No.)

Details

The dual signals are not matched between PLC CPU and CNC CPU.
The following factors may cause the error.

- Cable is disconnected.
- Sensor is broken.

Alarm No. shows the device No. which has the error. ("0024" indicates the device No. X24.)
When two or more signals are detected for errors, the No. shows the first detected signal.

Remedy

Check the wiring.

Y20 Dual signal: comparison ladder transfer error 0021 (Data transfer error code)

Details

The request to transfer data from CNC CPU to PLC CPU was not successfully processed.
<Data transfer error code>
4029, 41CF: The user ladder area does not have enough capacity for dual-signal comparison ladder to be written in.
4031: Incorrect device size has been set in PC parameter.
413A: There is a possibility that the different version of dual-signal comparison sequence (SSU_CMP) or partly changed or deleted SSU_CMP was written in the PLC CPU.

Remedy

<Data transfer error code>
4029, 41CF: Check whether the user ladder area has enough capacity. If the area does not enough capacity, decrease the number of steps in the user ladder and then restart the NC.
4031: Make sure that no change has been made to the device allocation settings of PC parameter.
Correct the PC parameter and then restart the NC.
Other than above: Contact the service center.
413A: Restart the NC. When SSU_CMP is stored by GX Developer, reread it from the CNC CPU.

Y20 Dual signal: comparison ladder transfer timeout 0022 (Data request code)

Details

A timeout error occurred at the data transfer request from CNC CPU to PLC CPU.
This error may occur when a file is being accessed by other devices such as GX Developer and GOT.

Remedy

Disconnect the access by other devices and restart the NC.

Y20 Dual signal: comparison ladder compare error 0023

Details

The dual-signal comparison ladder written in PLC CPU is not matched with that in CNC CPU.
A fault of PLC CPU or CNC CPU may cause the error.

Remedy

Contact the service center.

Y20 Dual signal: comparison stop (on PLC) 0024

Details

Dual-signal comparison stopped on PLC CPU.
The following factor may cause the error.
"SSU_CMP" is not entered in the [PLC parameter] - [Program] settings.

Remedy

Correct the settings of PC parameter.

Y20 Dual signal: comparison stop (on NC) 0025

Details

Dual-signal comparison stopped on CNC CPU.

Remedy

Contact the service center.

Y20 Dual signal: output/FB compare error 0026 (Device No.)

Details

The output signal from dual signal module is not matched with the feedback signal.
The following factor may cause the error.

- The dual signal module is not supplied with 24VDC.

(Ex.) When a compare error is detected in Y24/X24 signal, 0024 will be displayed.
When more than one signal are detected, the smallest No. will be dispalyed.

Remedy

Supply 24VDC to the dual signal module.
Make sure that the dual signal output is successfully done, and then restart the NC.
If the error is not cleared by these measures, the dual signal module may have a fault.
Replace the module.

Y20 Dual signal: parameter setting error 0027

Details

The parameter settings are incorrect.
Parameters to check:
"#21125 SSU_num (Number of dual signal modules)"
"#21143 SSU_Dev1 (Dual signal module device1)" to "#21145 SSU_dev3 (Dual signal module device3)"
"#2180(PR) S_DIN Speed observation input door No."
"#3140(PR) S_DINSp Speed observation input door No."

Remedy

Correct the parameter settings.

Y20 Dual signal: parameter compare error 0028

Details

Parameters have not been successfully transferred from CNC CPU to PLC CPU.
Parameters to check:
"#21125 SSU_num (Number of dual signal modules)"
"#21142 SSU_delay (Dual-signal comparison tolerance time)"
"#21143 SSU_Dev1 (Dual signal module device1)" to "#21145 SSU_dev3 (Dual signal module device3)"

Remedy

Contact the service center.

Y20 Dual signal: comparison ladder read error 0029

Details

The controller, when turned ON, cannot read the dual-signal comparison ladder correctly from the FROM of CNC CPU.

Remedy

Contact the service center.

Y20 Safety observation: module mount error 0030 (Unconfirmed module to be mounted)

Details

Some of the dual signal modules in the parameter "#21125 SSU_num (Number of dual signal modules)" are not confirmed to be mounted.
Alarm No. shows the unconfirmed module to be mounted at the bit-level.
("0006" indicates the module No.2 and 3.)

Remedy

Make sure that the dual signal modules entered in the parameter are all mounted.
Make sure that the remote I/O cable is connected.
Check for any overlapped station No. set with rotary switch of dual signal module.

Y20 Safety observation: parameter memory error 0031 (Parameter No.)

Details

- The parameters related to the safety observation function are not consistent with the check data, due to destruction of memory or other reason.
- Any value is set to #2448, #2449, #13248, or #13249 when "#21162(PR) mulstepssc Multi-step speed monitor enabled" is "1".

Remedy

- Correct the parameter settings. - When any value
is set to #2448, #2449, #13248, or #13249 when "#21162(PR) mulstepssc
Multi-step speed monitor enabled" is "1", set "0" to these parameters.

(Note) If the NC system is updated from the one older than Ver. C4 to the one newer than Ver. C4, the error can be cleared by restarting the NC.

Y20 Safety observation: PLC CPU type error 0032

Details

The PLC CPU type of the 1st module doesn't support the safety observation function.

Remedy

Check the PLC CPU type.
Contact the service center.

Y20 Dual-signal comparison sequence error (NC side) 0033

Details

NC side's dual-signal comparison sequence data is overwritten.

Remedy

Rewrite the NC side's dual-signal comparison sequence.

Y20 Dual signal PC parameter illegal 0034 xxyy

Details

PC parameter settings which were written to in the PLC CPU is illegal.

Remedy

Refer to the following error indication "xxyy" to correct the setting.

- xx = 01
  There is a contradiction between the setting of the dual signal module's head XY devices in I/O assignment setting and NC parameters "#21143 SSU_Dev1" - "#21145 SSU_Dev3".
  Check the dual signal module's position, then make the same setting for PC parameters and NC parameters.
  "yy" shows the module No. (yy=01 to 03)

- xx = 02
  In I/O assignment's detail settings, output mode of the dual signal module at error is set to "Hold".
  Set the output mode to "Clear".
  "yy" shows the module No. (yy=01 to 03)

- xx = 03
  The number of device points in device settings is illegal.
  Correct the number of device points.
  "yy" shows the No. corresponding to the device.
  The correspondence between Nos. and devices is as follows:
  00: M
  02: L
  03: F
  04: V
  10: B
  11: SB
  18: D
  24: W
  25: SW
  32: T
  35: C
  38: ST

- xx = 04
  High-speed timer limit is set to other than "10.00".
  Set the high-speed timer limit to "10.00".

Y20 Safety observation PC parameter get error 0035

Details

PLC CPU's PC parameters couldn't be got.

Remedy

Check if the PC parameters are correctly written in the PLC CPU.
Contact service center.

Y20 Safety observation parameter setting lock password data error 0036

Details

The password that is used to lock the safety observation parameter setting is incorrect.

Remedy

Read the password data in text format, then set the data in "#21150 Safety_key (Safety observation parameter password-lock cancel key)".

Y20 Safety observation device information setting parameter error 0037

Details

The setting values of the safety device mounting information 1 to 4 (#21151, #21152, #21157, #21158) do not coincide with the safety device mounting information check 1 to 4 (#21153, #21154, #21159, #21160).

Remedy

Correct the following parameter settings.
- #21151 SC_EQP_1 Safety device mounting information 1
- #21152 SC_EQP_2 Safety device mounting information 2
- #21157 SC_EQP_3 Safety device mounting information 3
- #21158 SC_EQP_4 Safety device mounting information 4
- #21153 SC_EQP_CHK1 Safety device mounting information check 1
- #21154 SC_EQP_CHK2 Safety device mounting information check 2
- #21159 SC_EQP_CHK3 Safety device mounting information check 3
- #21160 SC_EQP_CHK4 Safety device mounting information check 4

Y20 Safety observation device memory check error (PLC side) 0046

Details

An error was detected during device memory check of the PLC.

Remedy

Confirm that the devices used by the dual-signal comparison sequence are not written in the interruption program.
The PLC CPU may be broken.
Contact service center.

Y20 Safety observation device memory check error (NC side) 0047

Details

An error was detected during device memory check of the NC.

Remedy

The NC CPU may be broken.
Contact service center.

Y20 Dual signal output check error at power ON (PLC side) 0048 (Device No.)

Details

The output signal's ON/OFF check of the dual signal module didn't finish normally.

Remedy

The PLC side output transistor may be broken.
The output pin may be connected to 24VDC. Check the wiring of the device to which the error occurred.
The No. of the error module is displayed. Replace the dual signal module.

Y20 Dual signal output check error at power ON (NC side) 0049 (Device No.)

Details

The output signal's ON/OFF check of the dual signal module didn't finish normally.

Remedy

The NC side output transistor may be broken.
The output pin may be connected to 24VDC. Check the wiring of the device to which the error occurred.
The No. of the error module is displayed. Replace the dual signal module.

Y20 Output OFF check error (PLC side) 0050 (Unit No.)

Details

Although the output OFF check function turned OFF the dual signal module's output signals, there is a feedback input signal which is staying ON.

Remedy

The PLC side output transistor may be broken. The No. of the error module is displayed.
Replace the dual signal module.
Carry out the output OFF check to confirm there is no problem.

Y20 Output OFF check error (NC side) 0051 (Unit No.)

Details

Although the output OFF check function turned OFF the dual signal module's output signals, there is a feedback input signal which is staying ON.

Remedy

The NC side output transistor may be broken.
The No. of the error module is displayed.
Replace the dual signal module.
Carry out the output OFF check to confirm there is no problem.

Y21 Speed obsv signal: Speed over 0001 (Axis name)

Details

The speed exceeds the safety speed limit when the "speed monitor mode" signal is ON.
The name of the axis with an error is displayed.

Remedy

Decelerate the speed to reset the warning and start the speed monitor.

Y21 Dual signal: State after error unconfirmed 0020 (Device No.)

Details

- A dual-signal comparison error was detected.
  Signal outputs from both PLC and CNC CPUs have not been confirmed since the power was turned ON again.

The No. of device from which the signal output has not been confirmed is displayed.
When several signals have not been confirmed, the smallest device No. will be displayed.

Remedy

- Remove the cause of the dual-signal comparison error, turn the controller OFF and ON, then turn the signal ON and OFF in the dual-signal comparison state.
- The dual signals with the error can be reset and canceled if not used.

Y51 Parameter G0tL illegal 0001

Details

The time constant has not been set or exceeded the setting range.

Remedy

Correct "#2004 G0tL (G0 time constant (linear))".

Y51 Parameter G1tL illegal 0002

Details

The time constant has not been set or exceeded the setting range.

Remedy

Correct "#2007 G1tL (G1 time constant (linear))".

Y51 Parameter G0t1 illegal 0003

Details

The time constant has not been set or exceeded the setting range.

Remedy

Correct "#2005 G0t1 (G0 time constant (primary delay) / Second-step time constant for soft acceleration/deceleration)".

Y51 Parameter G1t1 illegal 0004

Details

The time constant has not been set or exceeded the setting range.

Remedy

Correct "#2008 G1t1 (G1 time constant (primary delay)/Second-step time constant for soft acceleration/deceleration)".

Y51 Parameter grid space illegal 0009

Details

The grid space is illegal.

Remedy

Correct "#2029 grspc(Grid interval)".

Y51 Parameter stapt1-4 illegal 0012

Details

The time constant has not been set or exceeded the setting range.

Remedy

Correct the parameters from "#3017 stapt1(Tap time constant (Gear: 00))" to "#3020 stapt4(Tap time constant (Gear: 11))".

Y51 Slave axis No. illegal 0014

Details

In the axis synchronization, parameter settings for slave axis have been attempted in different part system from that of master axis.

Remedy

- Correct the "#1068 slavno (Slave axis number)" setting.

Y51 Parameter skip_tL illegal 0015

Details

The time constant has not been set or exceeded the setting range.

Remedy

Correct "#2102 skip_tL (Skip time constant linear)".

Y51 Parameter skip_t1 illegal 0016

Details

The time constant has not been set or exceeded the setting range.

Remedy

Correct "#2103 skip_t1 (Skip time constant primary delay / Second-step time constant for soft acceleration/deceleration)".

Y51 Parameter G0bdcc illegal 0017

Details

"#1205 G0bdcc (Acceleration and deceleration before G0 interpolation)" for the 2nd part system is set to acceleration/deceleration before G0 interpolation.

Remedy

Correct "#1205 G0bdcc (Acceleration and deceleration before G0 interpolation)".

Y51 OMR-II parameter error 0018

Details

An illegal setting was found in the OMR-II-related parameters. OMR-II has been disabled.

Remedy

Correct the related parameter settings.

Y51 PLC indexing stroke length err 0019

Details

"#12804 tleng (Linear axis stroke length)" has not been set or exceeded the setting range while the linear axis equal indexing is enabled for the PLC indexing axis.

Remedy

Correct "#12804 tleng (Linear axis stroke length)".

Y51 No hi-accu acc/dec t-const ext 0020

Details

There is no expansion option for the high-accuracy acceleration/deceleration time constant.

Remedy

Set "#1207 G1btL (Time constant)" to a value within the setting range with no expansion specification for the extended high-accuracy time constant.

Y51 Values of PC1/PC2 too large 0101

Details

The PC1 and PC2 settings for the rotary axis are too large.

Remedy

Correct "#2201 SV001 PC1 (Motor side gear ratio)" and "#2202 SV002 PC2 (Machine side gear ratio)".

Y51 Parameter Error 0201

Details

Hypothetical axis control parameter setting is incorrect.

Remedy

- Correct the following parameter settings:
  "#1017 rot (Rotational axis)",
  "#2116 v_axis (Hypothetical axis)",
  "#2117 V_axno (Hypothetical axis No.)",
  "#12015 v_leng (Hypothetical axis tool length)"

Y51 Spindle/C axis unit illegal 0202

Details

The setting units of the spindle and C axis are different.

Remedy

Match the spindle unit ("#3035 spunit") and C axis part system unit("#1003 iunit").

Y90 No spindle signal 0001-0007

Details

There is an error in the spindle encoder signal.
The data transmission to the drive unit is stopped when this error occurs.

Remedy

Check the spindle encoder's feedback cable and the encoder.

Appendix 7.5 System Alarms (Z)

Z20 Power ON again

Details

- A parameter was set on the CNC monitor screen, which would become valid after turning the power ON again.
- A parameter was set which would become valid after turning the power ON again via EZSocket.
- Parameter data (ALL.PRM) was input.
- Backup data (SRAM.BIN) was input by SRAM backup function.
- Backup data and restore data were restored.
- A parameter was changed by G10 L70 command, which would become valid after turning the power ON again.
- "#1060 SETUP" was set to "1" and the standard parameters were set.

Remedy

Turn the power ON again.

Z30 ETHERNET ERROR 1 000E

Details

Socket open error
A communication error occurred when downloading a program during a program server operation.
000E displayed with an error No. indicates the communication type. (FTP communication type)

Remedy

Check the Ethernet communication path.
Check the system on the server side.

Z30 ETHERNET ERROR 5 000E

Details

Data receive error
A communication error occurred when downloading a program during a program server operation.
000E displayed with an error No. indicates the communication type. (FTP communication type)

Remedy

Check the Ethernet communication path.
Check the system on the server side.

Z30 ETHERNET ERROR 6 000E

Details

Data receive number error
A communication error occurred when downloading a program during a program server operation.
000E displayed with an error No. indicates the communication type. (FTP communication type)

Remedy

Check the Ethernet communication path.
Check the system on the server side.

Z40 Format mismatch

Details

"#1052 MemVal (No. of common variables shared in part system designation)", formatted at "0", was set to "1".

Remedy

- Reset "#1052 MemVal (No. of common variables shared in part system designation)" to "0" or format and restart.

Z51 EEPROM ERROR

Details

The data read from EEPROM does not coincide with the data that has been written into it.

Remedy

- If the same alarm occurs with the same operation, a hardware fault may be the cause. Contact the service center.

Z52 Battery fault 000x

Details

The voltage of the battery in the NC control unit has dropped. (The battery used to save the internal data.)
0001: Battery warning
0002: Battery detecting circuit error
0003: Battery alarm

(Note)The display of "Z52 battery fault 0001" can be removed by resetting. However, the warning state will not be cleared until the battery is replaced.

Remedy

- Replace the battery of the NC control unit.
- Check for any disconnection of the battery cable.
- After fixing the battery's fault, check the machining program.

Z53 CNC overheat

Details

The controller or operation board temperature has risen above the designated value.

(Note)Temperature warning
When an overheat alarm is detected, the alarm is displayed and the overheat signal is output simultaneously. Automatic operation will be continued, while restarting after resetting or stopping with M02/M30 is not possible. (Restarting after block stop or feed hold is possible.)
The alarm will be cleared and the overheat signal will turn OFF when the temperature drops below the specified temperature.
Z53 CNC overheat 000x
[000x]
0001: The temperature in the control unit is high.
0002: The temperature around the communication terminal (setting and display unit) is high.
0003: The temperature in the control unit and around the communication terminal (setting and display unit) is high.

The ambient temperature must be lowered immediately when a "Z53 CNC overheat" alarm occurs. However, if the machining needs to be continued, set "0" for the following parameter. Then the alarm will be invalidated.

Remedy

- Cooling measures are required.
- Turn OFF the controller power, or lower the temperature with a cooler, etc.

Z55 RIO communication stop

Details

An error occurs in the communication between the control unit and remote I/O unit.
Disconnection of a cable
Fault in remote I/O unit
Fault of power supply to remote I/O unit
The alarm and the I/O unit No. is displayed when an error occurs in the communication between the control unit and remote I/O unit.
The remote I/O unit No. is displayed in eight digits. Two digits (in hexadecimal) are used for each board and part system.
[Display format of remote I/O unit No.]
Z55 RIO communication stop __ __ __ __ __ __ __ __
(a) (b) (c) (d)(e) (f) (g) (h)
(a)(b): Remote I/O 2nd part system communication interrupted station
(c)(d): Remote I/O 1st part system communication interrupted station
(e)(f): Remote I/O 3rd part system communication interrupted station
(g)(h): Board connection remote I/O communication interrupted station
(a)(b) indicates the following station in hexadecimal.
bit0: RIO (0th station)
bit1: RIO (first station)
bit2: RIO (second station)
bit3: RIO (third station)
bit4: RIO (fourth station)
bit5: RIO (fifth station)
bit6: RIO (sixth station)
bit7: RIO (seventh station)
This also applies for the remote I/O 1st part system communication interrupted station, remote I/O 3rd part system communication interrupted station and board connection remote I/O communication interrupted station.

Remedy

- Check and replace the cables.
- Replace the remote I/O unit.
- Check the power supply (existence of supply and voltage).
- When not using the safety observation, set "1" in "#21102 add02/bit2 (RIO communication interruption alarm disabled)".

Z57 System warning 0100

Details

The specified capacity cannot be allocated for the buffer memory used for program server operation.

Remedy

A remedy like changing options setting is required. Contact the service center.
(Even this alarm occurs, other functions than program server operation can be used.)

Z58 ROM write not completed

Details

A machine tool builder macro program has not been written to FROM after being registered/ edited/ copied/ condensed/ merged/ the number changed/ deleted.

Remedy

- Write the machine tool builder macro program to FROM.

The program does not need to be written to FROM unless the editing operations and so on need to be valid after the NC power OFF.

Z59 Acc/dec time cnst too large

Details

Acceleration and deceleration time constants are too large.
(This alarm is output at the same time as "T02 0206".)

Remedy

- Set the larger value for "#1206 G1bF(Maximum speed)".
- Set the smaller value for "#1207 G1btL(Time constant)".
- Set the lower feedrate.

Z60 Fieldbus communication error n1 n2 n3 n4

Details

A communication error has occurred on the Fieldbus communication with HN571/HN573/ HN575.
[n1 :Shows state of the master channel (shown in hexadecimal number)]
00 :Offline In initializing
40 :Stop Cutting I/O communication
80 :Clear Resetting output data of each slave by sending 0 data.
C0 :In operation I/O In I/O communication
[n2 :Shows error state (shown in hexadecimal number)]
bit0 :Control error Parameter error
bit1 :Auto clear error Communication with all the slave channels was cut because a communication with one slave channel had an error.
bit2 :Non exchange error Slave channel with communication error is found.
bit3 :Fatal error The communication cannot be continued because severe network failure exists.
bit4 :Event error Short-circuit was found on the network.
bit5 :Not ready CNC communication is not ready.
bit6 :Time out error Time out was detected in communication with each channel.
bit7 :Not used
[n3 :Shows error No. (shown in hexadecimal number)]

- Error in master channel (when remote address with an error is FF (hexadecimal number))
  0 :No error Operating normally
  32 :No USR_INT Damage in HN571. Replace HN571.
  33 :No global data field
  34 :No FDL-task
  35 :No PLC-task
  37 :Master parameter incorrect
  39 :Slave parameter incorrect
  3C :Data offset exceeding allowable set value received. Check the configuration setting.
  3D :Slave data send range overlap
  3E :Slave data receive range overlap
  3F :Not set data hand shake Damage in HN571. Replace HN571.
  40 :RAM range exceeded
  41 :Slave parameter data set illegal
  CA :No segment
  D4 :Data base read illegal Download the configuration data again.
  D5 :Operating system illegal Damage in HN571. Replace HN571.
  DC :Watch dog error
  DD :Hand shake mode No data communication by 0
  DE :Master auto clear mode When setting auto clear mode, the auto clear mode was performed because one slave was not able to connect in run time.
- Error in slave channel (when remote address with an error is other than FF (hexadecimal number))
  Check the configuration of slave channel in which error has occurred. Check if there is any short-circuit in wire to bus.
  2 :Station overflow reported
  3 :Station stopped responding to master command
  9 :No slave required responding data
  11 :No station respond
  12 :No master to logical token ring
  15 :Illegal parameter requested

[n4 : Shows slave No. where communication error has occurred. (shown in hexadecimal number)]
'FF' means an error in master channel.

Z64 Valid term soon to be expired xx

Details

The valid term will be expired in less than a week. Remaining valid term is xx days.

Remedy

- Obtain a decryption code by making a payment. Enter the decryption code.

Z65 Valid term has been expired

Details

The valid term has been expired with no decryption code input.

Remedy

- Obtain a decryption code by making a payment. Enter the decryption code.

Z67 CC-Link communication error

Details

A communication error occurred during CC-Link communication using CC-Link unit.

Remedy

- Refer to "List of Messages" in CC-Link (Master/Slave) Specification manual (BNP-C3039-214).

Z68 CC-Link unconnected

Details

A cable between CC-Link unit and a device is disconnected or broken.

Remedy

- Connect the cable.
- Check for any broken cables.

Z69 External link error 2

Details

A FROM/TO instruction was used while the MELSEC-Q interface expansion module is not installed.

Remedy

Install the MELSEC-Q interface expansion module.

Z69 External link error 3

Details

A negative value was set for an I/O No. in the FROM/TO instruction.

Remedy

Correct the I/O No.

Z69 External link error 4

Details

A negative value was set for transfer size in the FROM/TO instruction.

Remedy

Correct the transfer size.

Z69 External link error 5

Details

The number of FROM/TO instructions within one scan has exceeded 50.

Remedy

Correct the user PLC (ladder sequence) so that the number of FROM/TO instructions per scan is 50 or less.

Z69 External link error 6

Details

The access to the buffer memory by the FROM/TO instruction has exceeded 12K words per scan.

Remedy

Correct the user PLC (ladder sequence) so that the buffer memory access by the FROM/TO instruction won't exceed 12K words per scan. (The total size of FROM/TO is up to 12K words.)

Z69 External link error 7

Details

A FROM/TO instruction was used in high-speed processing.

Remedy

Delete the FROM/TO instruction from high-speed processing.

Z69 External link error 8

Details

The bit device number designated in the FROM/TO instruction is not a multiple of 16.

Remedy

Correct the bit device number designated in the FROM/TO instruction to be a multiple of 16.

Z69 External link error 9

Details

With a FROM/TO instruction, a value out of the address range (negative value, or 0x8000 or over) was set as the head address of the buffer memory.

Remedy

Correct the head address of the buffer memory.

Z69 External link error 10

Details

An alarm occurred in the MELSEC module mounted on the extension base.

Remedy

Check for any disconnection of the MELSEC module and the cables on the extension base. Then turn the CNC's power ON again.

Z69 External link error 11

Details

The I/O No. designated in the FROM/TO instruction is different from the mounted location of the intelligent function module on the extension base (the module's I/O No.).

Remedy

Correct the I/O No. Then turn the CNC's power ON again.

Z89 APLC ERROR 0001

Details

C language module is not adequately stored in NC in APLC release.

Remedy

Resend the C language module.

Z89 APLC ERROR 0002

Details

SRAM size exceeded 16KB in APLC release.

Remedy

Make the SRAM size to 16KB or smaller.

Z89 APLC ERROR 0003

Details

DRAM size exceeded 512KB in APLC release.

Remedy

Downsize the DRAM to 512KB or smaller.

Z89 APLC ERROR 0004

Details

APLC module does not include initialize function aplc_top, dramSizeCheck, sramSizeCheck, or setUserBaseMain.

Remedy

Check if APLC module includes the above functions.

Z89 APLC ERROR 0005

Details

The contents of APLC module in FROM is illegal and cannot be loaded.

Remedy

Check the contents of APLC module.
* Check if it does not include an undefined function."

Z89 APLC ERROR 0006

Details

APLC was activated during APLC invalid mode.

Remedy

Change to APLC valid mode.

Z99 FILE AREA ERROR 0001

Details

The edited data is destroyed probably because it was not properly updated and the backup operation did not complete normally.

Remedy

Format the area again.

Z99 FILE AREA ERROR 0002 (month, date)

Details

There was a high possibility that the edited data was not properly updated so that it was recovered using the backup data.
The machining program is not the newest.

Remedy

Check the machining program you were editing, and edit it again if it is same as the one before being edited.
This alarm will be canceled by turning ON the edit data recovery signal after recovering work is completed.
"Emergency stop DATA" will be canceled by turning the controller ON again.
The data when the recovered data was backuped will be displayed in four digits (month and date) behind the error No., as a rough idea for data recovery.

Z99 FILE AREA ERROR 0003

Details

The edit work just before the power went down may not be reflected.

Remedy

Check the machining program you were editing, and edit it again if it is same as the one before editing.
This alarm will be canceled by turning ON the edit data recovery signal after recovering work is completed.
"Emergency stop DATA" will be canceled by turning the controller ON again.

Z99 FILE AREA ERROR 0004

Details

The compressed data does not fit in the memory.

Remedy

Delete unnecessary data and edit it again.
If the power is turned ON again while the error still appears, the program revert to the status before the error occurred.

Z99 FILE AREA ERROR 0005

Details

Data cannot be uncompressed normally at power ON.

Remedy

Format the area again. Contact the service center.

Z99 FILE AREA ERROR 0006

Details

Memory necessary for edited data or compression cannot be allocated.

Remedy

Contact the service center.

Appendix 7.6 Absolute Position Detection System Alarms (Z7*)

(Note1) "Z70"alarms are displayed as "Z70 Abs data error"with the error number. Error number is the four digit number displayed displayed after error name(start from 0001). "Z70"alarms are listed in ascending order in this manual.

(Note2) "Z71"alarms are displayed as "Z71 Abs encoder failure"with the error number. Error number is the four digit number displayed displayed after error name(start from 0001). "Z71"alarms are listed in ascending order in this manual.

Z70 Abs posn base set incomplete 0001 (Axis name)

Details

Zero point initialization is incomplete. Otherwise, the spindle was removed.

Remedy

Complete zero point initialization.

(Note) To release alarm "Z70 Abs data error", enter the parameter data output when establishing the absolute position and turn ON the power again. For the rotary axis, however, the alarm cannot be released by entering the parameter data.

- Zero point initialization: Required

Z70 Absolute position lost 0002 (Axis name)

Details

The absolute position basic point data saved in the NC has been damaged.

Remedy

Set the parameters. If the basic point data is not restored by setting the parameters, perform zero point initialization.

(Note) To release alarm "Z70 Abs data error", enter the parameter data output when establishing the absolute position and turn ON the power again. For the rotary axis, however, the alarm cannot be released by entering the parameter data.

- Zero point initialization: (Required)

Z70 Abs posn param changed 0003 (Axis name)

Details

Any of the parameters for absolute position detection has been changed.
#1003 iunit
#1016 iout
#1017 rot
#1018 ccw
#1040 M_inch
#2049 type

Remedy

Correct the parameter settings. Then turn the power ON again and perform zero point initialization.

(Note) To release alarm "Z70 Abs data error", enter the parameter data output when establishing the absolute position and turn ON the power again. For the rotary axis, however, the alarm cannot be released by entering the parameter data.

- Zero point initialization: Required

Z70 Abs posn initial set illegal 0004 (Axis name)

Details

The zero point initialization point is not at the grid position.

Remedy

Perform the zero point initialization again.

(Note) To release alarm "Z70 Abs data error", enter the parameter data output when establishing the absolute position and turn ON the power again. For the rotary axis, however, the alarm cannot be released by entering the parameter data.

- Zero point initialization: Required

Z70 Abs posn param restored 0005 (Axis name)

Details

The data has been restored by inputting the parameters during the alarm No.0002.

Remedy

Turn the power ON again to start the operation.

(Note) To release alarm "Z70 Abs data error", enter the parameter data output when establishing the absolute position and turn ON the power again. For the rotary axis, however, the alarm cannot be released by entering the parameter data.

- Zero point initialization: Not required

Z70 Abs data error 0006 (axis name)

Details

Deviation of the servo axis with scale when the power is OFF exceeds the set value in "#2051 check (Check)".

Remedy

Search for the factor which led the deviation of the servo axis at the power OFF.

- Zero point initialization: Not required
- Alarm reset when power is turned OFF: -
- Servo alarm No.: -

Z70 Abs posn data lost 0080 (Axis name)

Details

The absolute position data has been lost. An error of the multi-rotation counter data in the detector and so on may be the cause.

Remedy

Replace the detector and complete zero point initialization.

(Note) To release alarm "Z70 Abs data error", enter the parameter data output when establishing the absolute position and turn ON the power again. For the rotary axis, however, the alarm cannot be released by entering the parameter data.

- Zero point initialization: Required
- Servo alarm No.: (9E)etc.

Z70 Abs posn error(servo alm 25) 0101 (Axis name)

Details

The servo alarm No. 25 was displayed and the power was turned ON again.

Remedy

Perform zero point initialization again.

(Note) To release alarm "Z70 Abs data error", enter the parameter data output when establishing the absolute position and turn ON the power again. For the rotary axis, however, the alarm cannot be released by entering the parameter data.

- Zero point initialization: Required
- Servo alarm No.: -25

Z70 Abs posn error(servo alm E3) 0106 (Axis name)

Details

The servo alarm No. E3 was displayed and the power was turned ON again.

Remedy

Perform zero point initialization again.

(Note) To release alarm "Z70 Abs data error", enter the parameter data output when establishing the absolute position and turn ON the power again. For the rotary axis, however, the alarm cannot be released by entering the parameter data.

- Zero point initialization: Required
- Servo alarm No.: (E3)

Z71 AbsEncoder:Backup voltage drop 0001 (Axis name)

Details

Backup voltage in the absolute position detector dropped.

Remedy

Replace the battery, check the cable connections, and check the detector. Turn the power ON again and perform zero point initialization.
- Zero point initialization: Required
- Alarm reset when power is turned OFF: -(Z70-0101 is displayed after the power is turned ON again.)
- Servo alarm No.: 25

Z71 AbsEncoder: Commu error 0003 (Axis name)

Details

Communication with the absolute position detector has been disabled.

Remedy

Check and replace the cables, card or detector. Turn the power ON again and perform zero point initialization.
- Zero point initialization: (Required) only when the detector has been replaced.
- Alarm reset when power is turned OFF: Reset
- Servo alarm No.: 91

Z71 AbsEncoder: Abs data changed 0004 (Axis name)

Details

Absolute position data has been changed at the absolute position establishment.

Remedy

Check and replace the cables, card or detector. Turn the power ON again and perform zero point initialization.
- Zero point initialization: (Required) only when the detector has been replaced.
- Alarm reset when power is turned OFF: Reset
- Servo alarm No.: 93

Z71 AbsEncoder: Serial data error 0005 (Axis name)

Details

An error of the serial data was found in the absolute position detector.

Remedy

Check and replace the cables, card or detector. Turn the power ON again and perform zero point initialization.
- Zero point initialization: (Required) only when the detector has been replaced.
- Alarm reset when power is turned OFF: Reset
- Servo alarm No.: 92

Z71 AbsEncoder: Abs/inc posn diffr 0006 (Axis name)

Details

Servo alarm E3
Absolute position counter warning

Remedy

Operation is possible until the power is turned OFF.
- Zero point initialization: (Required) after the power is turned ON again.
- Alarm reset when power is turned OFF: Reset (Z70-0106 is displayed after the power is turned ON again.)
- Servo alarm No.: E3

Z71 AbsEncoder: Initial commu er 0007 (Axis name)

Details

Initial communication with the absolute position detector is not possible.

Remedy

Check and replace the cables, card or detector. Turn the power ON again and perform zero point initialization.
- Zero point initialization: (Required) only when the detector has been replaced.
- Alarm reset when power is turned OFF: Reset
- Servo alarm No.: 18

Z73 Battery for abs data fault 0001

Details

Low backup battery
Servo alarm 9F
Low battery voltage

Remedy

This is displayed when the battery voltage is low or the cable has been damaged.
The absolute position initialization is not required.
Even after the servo alarm 9F is canceled, this alarm will continue to be displayed until NC reset is entered.

(Note) When this alarm has occurred, do not turn OFF the drive unit power in order to protect the absolute position data. Replace the battery with the drive unit power ON.

Appendix 7.7 Emergency Stop Alarms (EMG)

When there are several causes for an emergency stop, only one of them will be displayed. The display priority is shown below in descending order.

DATA, SRV, SPIN, PARA, LAD, MULT, IPWD, LINK, MCT, EXIN, CVIN, SUIN, PLC, APLC, STOP, STP2

Refer to Emergency stop cause (G10221/R21) to confirm which causes are detected.

EMG Emergency stop PLC

Details

The built-in PLC has entered the emergency stop state during the sequence process.

Remedy

- Investigate and remove the cause of the built-in PLC emergency stop.

EMG Emergency stop EXIN

Details

The "emergency stop" signal is significant (open).

Remedy

- Cancel the "emergency stop" signal.
- Check for any broken wires.

EMG Emergency stop SRV

Details

An alarm occurred in the servo system causing an emergency stop.

Remedy

- Investigate and remove the cause of the servo alarm.

EMG Emergency stop STOP

Details

The sequence program in PLC is not running.

Remedy

- Check the setting of the toggle switch in front side of the PLC module. Correct it if set to STOP.

EMG Emergency stop SPIN

Details

Spindle drive unit is not mounted.

Remedy

- Cancel the causes of the other emergency stop.
- Check the "emergency stop" signal input in the spindle drive unit.

EMG Emergency stop PARA

Details

Setting of the door open II fixed device is illegal.
Setting of the parameters for dog signal random assignment is illegal.

Remedy

- Correct the "#1155 DOOR_m" and "#1156 DOOR_s" settings. (When the door open II fixed device is not used, set "#1155 DOOR_m" and "#1156 DOOR_s" to "100".)
- Correct the "#2073 zrn_dog (Origin dog Random assignment device)", "#2074 H/W_OT+ (H/W OT+ Random assignment device)", "#2075 H/W_OT- (H/W OT- Random assignment device)" and "#1226 aux10/bit5 (Arbitrary allocation of dog signal)" settings.

EMG Emergency stop LINK

Details

A DeviceNet communication error has occurred.
(Any of the network errors L10, L11 and L12 has occurred.)

Remedy

- Clear the network error.
- Setting "0" in "#21113 Add13/bit0 DeviceNet error monitor" disables the DeviceNet communication error monitoring and clears this alarm.

EMG Emergency stop WAIT

Details

The preparation sequence is not sent from the master station. Otherwise, the contents of the received preparation sequence are inconsistent with those of the parameters, so that the usual sequence cannot be started.

(Note) When "LINK" is also entered for the PLC serial link, refer to "Note" in the section, "LINK".

Remedy

- Check that the CC-Link card switch setting and wiring as well as the external sequencer transmission are normal.
- Check the diagnostic screen for link communication errors.

EMG Emergency stop XTEN

Details

The CC-Link card is operating incorrectly.
Switch/parameter settings for the CC-Link card are incorrect.

Remedy

- Replace the CC-Link card.
- Correct the switch/parameter settings for the CC-Link card.

EMG Emergency stop LAD

Details

The sequence program has an illegal code.

Remedy

- Correct any illegal device Nos. or constants in the sequence program.

EMG Emergency stop CVIN

Details

The "emergency stop" signal for power supply is significant (open) because the external emergency stop function for power supply is enabled.

Remedy

- Cancel the "emergency stop" signal.
- Check for any broken wires.

EMG Emergency stop MCT

Details

The status is one of the following:

- The contactor shutoff test is being executed.
- The output OFF check is being executed.
- The "dual signal check start" signal was not ON when "#21161 SftySgnlChkTrg (Dual signal check-time change)" was set to "1". The output check and contactor welding detection at the power ON have not been completed.

Remedy

- The emergency stop is reset automatically after the contactor shutoff is confirmed.
- If the contactor shutoff is not confirmed within 5 seconds after the "contactor shutoff test" signal has been input, the "contactor welding detected" alarm occurs and the emergency stop status remains.
- Make sure that the contactor's auxiliary B contact signal is correctly output to the device that is set in "#1330 MC_dp1" and "#1331 MC_dp2" (Contactor weld detection device 1 and 2), and then turn the power ON again.
- The emergency stop is reset automatically after the output OFF check is completed.
- If the contactor shutoff is not confirmed within 5 seconds after the "output OFF check" signal has been input, the "output OFF check error" alarm occurs. The dual signal module may be broken when this alarm occurs. Replace the module.
- When "#21161 SftySgnlChkTrg (Dual signal check time change)" is set to "1", turn ON the "dual signal check start" signal after the power ON.

EMG Emergency stop IPWD

Details

The data backup for power failure might not have been executed successfully at the previous power failure.

Remedy

- If this message appears frequently, the power supply may be deteriorated. Contact the service center.

EMG Emergency stop SUIN

Details

The emergency stop input signal (M0) is OFF in the NC/PLC safety circuit.

Remedy

- Check the conditions for turning ON the emergency stop input signal.
- Check for any broken wires.

EMG Emergency stop STP2

Details

Sequence programs stopped in CNC.

Remedy

- Correct the rotary switch 1 (on the right) of the control unit if set to “1”.

EMG Emergency stop MULT

Details

An error related to Q bus or Qr bus occurred.

Remedy

- Refer to the error No. that follows the message “A01 Multi CPU error” to take a remedy.

EMG Emergency stop DATA

Details

An error was detected in the data in NC and the following alarm occurred.
- Z99 FILE AREA ERROR (except for 0004)

Remedy

- Refer to the remedy of "Z99 FILE AREA ERROR".
- This emergency stop will be canceled by resolving the error cause and turning the power ON again.

EMG Emergency stop APLC

Details

Emergency stop status is established during APLC release.

Remedy

Contact the machine tool builder.

Appendix 7.8 Auxiliary Axis Operation Errors (M)

M00 Aux ax R-pnt direction illegal 0003 (Axis No. 1 to 4)

Details

When executing reference position return, the axis was moved in the opposite of the designated direction.

Remedy

- Move the axis in the correct direction.

M00 Aux ax external interlock 0004 (Axis No. 1 to 4)

Details

The axis interlock function is valid.

Remedy

- Cancel the interlock signal

M00 Aux ax internal interlock 0005 (Axis No. 1 to 4)

Details

An interlock was established by the servo OFF function.

Remedy

- Cancel the servo OFF.

M00 Aux ax stored stroke limit 0007 (Axis No. 1 to 4)

Details

The stored stroke limit was reached.

Remedy

- Check the stored stroke limit setting and machine position

M00 Aux ax R ret invld at abs alm 0024 (Axis No. 1 to 4)

Details

Reference position return was executed during an absolute position alarm.

Remedy

- Initialize the absolute position reference point and then fix the absolute position coordinates.

M00 Aux ax R ret invld at ini 0025 (Axis No. 1 to 4)

Details

Reference position return was executed while initializing the absolute position.

Remedy

- Initialize the absolute position reference point and then fix the absolute position coordinates.

M00 Aux ax no operation mode 0101 (Axis No. 1 to 4)

Details

The operation mode is not designated, or the operation mode was changed during axis movement.

Remedy

- Correctly designate the operation mode.

M00 Aux ax feedrate 0 0103 (Axis No. 1 to 4)

Details

The operation parameter's feedrate setting is zero.
The operation parameter feedrate setting is zero.
Or, the override is valid, and the override value is zero.

Remedy

- Set a value other than zero in the feedrate setting or override value.

M00 Aux ax sta No. illegal 0160 (Axis No. 1 to 4)

Details

A station No. exceeding the No. of indexed divisions was designated.

Remedy

- Correctly designate the station No.

M00 Aux ax R-pnt ret incomplete 0161 (Axis No. 1 to 4)

Details

Automatic/manual operation was started before reference position return was executed with the incremental system.

Remedy

- Execute the reference position return.

M00 Aux abs position initializing 0162 (Axis No. 1 to 4)

Details

The start signal was input while initializing the absolute position reference point.

Remedy

- Complete the absolute position reference point initialization.

M00 Aux ax abs position error 0163 (Axis No. 1 to 4)

Details

The start signal was input during an absolute position alarm.

Remedy

- Initialize the absolute position reference point and then fix the absolute position coordinates.

M00 Aux ax arbitrary positioning 0164 (Axis No. 1 to 4)

Details

The manual operation mode was started during the random positioning mode.

Remedy

- Turn the random positioning mode OFF before switching to the manual operation mode.

M00 Aux arbitrary coordinate index sta No. ilgl 0165 (Axis No. 1 to 4)

Details

The commanded station No. was higher than 20 or the number of indexing stations during arbitrary coordinate indexing.

Remedy

- Check the commanded station No. and the parameter "#12801 station" setting.
- Designated station No.0 during arbitrary coordinate indexing.

Appendix 7.9 Built-in PLC Alarms (U)

U10 Illegal PLC 0400 (Sub status)

Details

The instruction being used is not supported by built-in PLC. Otherwise, nothing is designated in the CALL or CJ instruction.

Remedy

Correct the sequence program.

U10 Illegal PLC 1000 (Sub status)

Details

The instruction being used is not supported by built-in PLC.

Remedy

Correct the sequence program.

U10 Illegal PLC 2700 (Sub status)

Details

The instruction, integrated statement or integrated note being used is not supported by built-in PLC.

Remedy

- Correct the sequence program.
- Restart the NC after PC memory format, and then write the corrected sequence program.

U10 Illegal PLC 8001 (Sub status)

Details

The designated value with BIN command is illegal.

Remedy

Confirm the methods for using the BIN function commands.

U10 Illegal PLC 8002 (Sub status)

Details

The designated value with BCD command is illegal.

Remedy

Confirm the methods for using the BCD function commands.

Appendix 7.10 Multi CPU Errors (A)

For alarms which are not explained below, refer to the error code list in "QCPU User's Manual (Hardware Design, Maintenance and Inspection)" (SH(NA)-080483).

A01 MULTI CPU ERROR 1000

Details

Run mode suspended or failure of CPU module.

- Malfunctioning due to noise or other reason
- Hardware fault

Remedy

- Take noise reduction measures.
- Reset and restart the CPU module.
  If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1001

Details

Run mode suspended or failure of CPU module.

- Malfunctioning due to noise or other reason
- Hardware fault

Remedy

- Take noise reduction measures.
- Reset and restart the CPU module.
  If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1002

Details

Run mode suspended or failure of CPU module.

- Malfunctioning due to noise or other reason
- Hardware fault

Remedy

- Take noise reduction measures.
- Reset and restart the CPU module.
  If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1003

Details

Run mode suspended or failure of CPU module.

- Malfunctioning due to noise or other reason
- Hardware fault

Remedy

- Take noise reduction measures.
- Reset and restart the CPU module.
  If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1004

Details

Run mode suspended or failure of CPU module.

- Malfunctioning due to noise or other reason
- Hardware fault

Remedy

- Take noise reduction measures.
- Reset and restart the CPU module.
  If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1005

Details

Run mode suspended or failure of CPU module.

- Malfunctioning due to noise or other reason
- Hardware fault

Remedy

- Take noise reduction measures.
- Reset and restart the CPU module.
  If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1006

Details

Run mode suspended or failure of CPU module.

- Malfunctioning due to noise or other reason
- Hardware fault

Remedy

- Take noise reduction measures.
- Reset and restart the CPU module.
  If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1009

Details

- A failure is detected on the power supply module, CPU module, base unit or extension cable.

Remedy

- Reset and restart the CPU module.
  If the same error is detected again, it is considered that the power supply module, CPU module, base unit or extension cable is faulty. Contact your local service center.

A01 MULTI CPU ERROR 1010

Details

Entire program was executed without the execution of an END instruction.

- When the END instruction is executed it is read as another instruction code, e.g. due to noise.
- The END instruction has been changed to another instruction code somehow.

Remedy

- Take noise reduction measures.
- Reset and restart the CPU module.
  If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1101

Details

- The sequence program storing built-in RAM/program memory in the CPU module is faulty.

Remedy

- This suggests a CPU module hardware fault. Contact your nearest Mitsubishi representative.

A01 MULTI CPU ERROR 1102

Details

- The work area RAM in the CPU module is faulty.

Remedy

- This suggests a CPU module hardware fault. Contact your nearest Mitsubishi representative.

A01 MULTI CPU ERROR 1103

Details

- The device memory in the CPU module is faulty.

Remedy

- This suggests a CPU module hardware fault. Contact your nearest Mitsubishi representative.

A01 MULTI CPU ERROR 1104

Details

- The address RAM in the CPU module is faulty.

Remedy

- This suggests a CPU module hardware fault. Contact your nearest Mitsubishi representative.

A01 MULTI CPU ERROR 1105

Details

- The memory in the CPU module is faulty.
- The CPU shared memory in the CPU module is faulty.

Remedy

- Take noise reduction measures.
- Reset and restart the CPU module.
  If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1150

Details

- The CPU module indicates a fault of memory in the Multi CPUs high-speed communication area.

Remedy

- Take noise reduction measures.
- Reset and restart the CPU module.
- If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1160

Details

- The program memory in the CPU module is faulty.

Remedy

- Take noise reduction measures.
- Format the program memory, write all files to the PLC, then reset the CPU module, and RUN it again.
- If the same error is displayed again, the possible cause is a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1161

Details

- Data in the built-in device memory was overwritten.

Remedy

- Take noise reduction measures.
- If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1162

Details

- A fault of the data in the battery was detected.
  (This error occurs when the automatic formatting is not set.)

Remedy

- Replace the battery of either CPU module or SRAM card.
- Take noise reduction measures.
- If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1200

Details

- The operation circuit for indexing in the CPU module does not operate normally.

Remedy

- This suggests a CPU module hardware fault.
  Contact your local service center.

A01 MULTI CPU ERROR 1201

Details

- The hardware (logic) in the CPU module does not operate normally.

Remedy

- This suggests a CPU module hardware fault.
  Contact your local service center.

A01 MULTI CPU ERROR 1202

Details

- The operation circuit for sequence processing in the CPU module does not operate normally.

Remedy

- This suggests a CPU module hardware fault.
  Contact your local service center.

A01 MULTI CPU ERROR 1300

Details

- There is an output module with a blown fuse.

Remedy

- Check FUSE. LED of the output modules and replace the module whose LED is lit.
- The module with a blown fuse can also be identified using a programming tool. Check the special registers SD1300 to SD1331 to see if the bit corresponding to the module is "1".
- When a GOT is bus-connected to the base unit, check the connection status of the extension cable and the earth status of the GOT.

A01 MULTI CPU ERROR 1310

Details

- An interruption has occurred although there is no interrupt module.

Remedy

- Any of the mounted modules is experiencing a hardware fault. Therefore, check the mounted modules and change the faulty module.
  Contact your local service center.

A01 MULTI CPU ERROR 1311

Details

- An interrupt request from other than the interrupt module was detected.

Remedy

- Take action so that an interrupt will not be issued from other than the interrupt module.

Details

- An interrupt request from the module where interrupt pointer setting has not been made in the PLC parameter dialog box was detected.

Remedy

- Correct the interrupt pointer setting in the PLC system setting of the PLC parameter dialog box.
- Take measures so that an interrupt is not issued from the module where the interrupt pointer setting in the PLC system setting of the PLC parameter dialog box has not been made.
- Correct the interrupt setting of the network parameter.
- Correct the interrupt setting of the intelligent function module buffer memory.
- Correct the basic program of the QD51.

A01 MULTI CPU ERROR 1401

Details

- There was no response from the intelligent function module in the initial communication.
- The size of the buffer memory of the intelligent function module is invalid.
- There was no response from the intelligent function module.
  (When error is generated, the head I/O number of the special function module that corresponds to the common information is stored.)

Remedy

- The CPU module is experiencing a hardware fault.
  Contact your local service center.

A01 MULTI CPU ERROR 1402

Details

- The intelligent function module was accessed in the program, but there was no response.

Remedy

- The CPU module is experiencing a hardware fault.
  Contact your local service center.

A01 MULTI CPU ERROR 1403

Details

- There was no response from the intelligent function module when the END instruction is executed.
- An error is detected at the intelligent function module.
- The I/O module (intelligent function module) is nearly removed, completely removed, or mounted during running.

Remedy

- The CPU module, base unit and/or the intelligent function module that was accessed is experiencing a hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1411

Details

- When performing a parameter I/O allocation the intelligent function module could not be accessed during initial communications.
  (On error occurring, the head I/O number of the corresponding intelligent function module is stored in the common information.)

Remedy

- Reset and restart the CPU module.
  If the same error is displayed again, the intelligent function module, CPU module or base unit is faulty.
  Contact your local service center.

A01 MULTI CPU ERROR 1412

Details

- The FROM/TO instruction is not executable, due to a control bus error with the intelligent function module.
  (On error occurring, the program error location is stored in the individual information.)

Remedy

- Reset and restart the CPU module.
  If the same error is displayed again, the intelligent function module, CPU module or base unit is faulty.
  Contact your local service center.

A01 MULTI CPU ERROR 1413

Details

An error is detected on the system bus.

- Self-diagnosis error of the system bus.
- Self-diagnosis error of the CPU module

Remedy

- Reset and restart the CPU module.
  If the same error is displayed again, the intelligent function module, CPU module or base unit is faulty.
  Contact your local service center.

A01 MULTI CPU ERROR 1414

Details

- An error is detected on the system bus.
- Fault of a loaded module was detected.

Remedy

- Reset and restart the CPU module.
  If the same error is displayed again, the intelligent function module, CPU module or base unit is faulty.
  Contact your local service center.

A01 MULTI CPU ERROR 1415

Details

- Fault of the main or extension base unit was detected.

Remedy

- Reset and restart the CPU module.
  If the same error is displayed again, the intelligent function module, CPU module or base unit is faulty.
  Contact your local service center.

A01 MULTI CPU ERROR 1416

Details

- System bus fault was detected at PLC power-on or CPU module reset.
- In a multiple CPU system, a bus fault was detected at power-on or reset.

Remedy

- Reset and restart the CPU module.
  If the same error is displayed again, the intelligent function module, CPU module or base unit is faulty.
  Contact your local service center.

A01 MULTI CPU ERROR 1430

Details

- A faulty of host CPU module was detected during the multi-CPUs high-speed communication.

Remedy

- Reset and restart the CPU module.
  If the same error is displayed again, this suggests a CPU module hardware fault.
  Contact your local service center.

A01 MULTI CPU ERROR 1431

Details

- A communication error with another CPU module was detected during the multi-CPUs high-speed communication.

Remedy

- Take noise reduction measures.
- Check the configuration of the main base unit in the CPU module.
- Reset and restart the CPU module. If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1432

Details

- A communication timeout error with another CPU was detected during the multi-CPUs high-speed communication.

Remedy

- Reset and restart the CPU module. If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1433

Details

- A communication error with another CPU was detected during the multi-CPUs high-speed communication.

Remedy

- Take noise reduction measures.
- Check the configuration of the main base unit in the CPU module.
- Reset and restart the CPU module. If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1434

Details

- A communication error with another CPU was detected during the multi-CPUs high-speed communication.

Remedy

- Take noise reduction measures.
- Check the configuration of the main base unit in the CPU module.
- Reset and restart the CPU module. If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1435

Details

- A communication error with another CPU was detected during the multi-CPUs high-speed communication.

Remedy

- Take noise reduction measures.
- Check the configuration of the main base unit in the CPU module.
- Reset and restart the CPU module. If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1436

Details

A fault is detected on the main base module in the multi-CPUs high-speed communication. (An error was detected on the multi-CPUs high-speed communication bus.).

Remedy

- Reset and restart the CPU module. If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1437

Details

A fault is detected on the main base module in the multi-CPUs high-speed communication. (An error was detected on the multi-CPUs high-speed communication bus.)

Remedy

- Take noise reduction measures.
- Check the configuration of the main base unit in the CPU module.
- Reset and restart the CPU module. If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 1500

Details

- A momentary power supply interruption has occurred.
- The power supply went off.

Remedy

- Check the power supply.

A01 MULTI CPU ERROR 1600

Details

- The battery voltage in the CPU module has dropped below stipulated level.

Remedy

- Change the battery.

Details

- The lead connector of the CPU module battery is not connected.

Remedy

- If the battery is for program memory, standard RAM or for the back-up power function, install a lead connector.

A01 MULTI CPU ERROR 1601

Details

- Voltage of the battery on memory card 1 has dropped below stipulated level.

Remedy

- Change the battery.

A01 MULTI CPU ERROR 1610

Details

- The number of writing to the FLashROM (the standard RAM or the area reserved by the system) exceeded one hundred thousand times.
  (Number of writing>100,000 times)- Voltage of the battery on memory card 1 has dropped below stipulated level.

Remedy

- Replace the CPU modules.

A01 MULTI CPU ERROR 2000

Details

I/O module information power ON is changed.

- I/O module (or intelligent function module) not installed properly or installed on the base unit.

Remedy

Read the common information of the error using a programming tool, and check and/or change the module that corresponds to the numerical value (module number) there.

- Alternatively, monitor the special registers SD1400 to SD1431 using GX Developer, and change the fuse at the output module whose bit has a value of "1".
- When a GOT is bus-connected to the main base unit or extension base unit, check the connection status of the extension cable and the grounding status of the GOT.

A01 MULTI CPU ERROR 2001

Details

- During operation, a module was mounted on the slot where the empty setting of the CPU module was made.

Remedy

- During operation, do not mount a module on the slot where the empty setting of the CPU module was made.

A01 MULTI CPU ERROR 2011

Details

- The QA*B,QA1S*B was used as the base unit.

Remedy

- Do not use the QA*B,QA1S*B as the base unit.

A01 MULTI CPU ERROR 2100

Details

- In the I/O assignment setting of PLC parameter, intelligent function module was allocated to an I/O module or vice versa.
- In the I/O assignment setting of PLC parameter, a module other than CPU (or nothing) was allocated to the location of a CPU module or vice versa.
- In the I/O assignment setting of the PLC parameter, general switch setting was made to the module that has no general switch setting.
- In the I/O assignment setting of the PLC parameter dialog box, the number of points assigned to the intelligent function module is less than the number of points of the mounted module.

Remedy

- Make the PLC parameter I/O assignment setting again so it is consistent with the actual status of the intelligent function module and the CPU module.
- Delete the switch setting in the I/O assignment setting of the PLC parameter.

A01 MULTI CPU ERROR 2103

Details

- 2 or more interrupt module, QI60, where interrupt pointer setting has not been made are mounted.

Remedy

- Reduce the QI60 modules to one.
- Make interrupt pointer setting to the second QI60 module and later.

A01 MULTI CPU ERROR 2106

Details

- 5 or more MELSECNET/H modules have been installed.
- 5 or more Ethernet modules have been installed.
- The same network Nos. or station Nos. exist in the MELSECNET/10 network system.

Remedy

- Reduce the number of MELSECNET/H modules to 4 or less.
- Reduce the number of Ethernet modules to 4 or less.
- Check the network Nos. and station Nos.

A01 MULTI CPU ERROR 2107

Details

- The start X/Y set in the PLC parameter's I/O assignment settings is overlapped with the one for another module.

Remedy

- Make the PLC parameter's I/O assignment setting again so it is consistent with the actual status of the intelligent function module.

A01 MULTI CPU ERROR 2110

Details

- The location designated by the FROM/TO instruction set is not the intelligent function module.
- The module that does not include buffer memory has been specified by the FROM/TO instruction.
- The intelligent function module being accessed is faulty.
- Station not loaded was specified using the instruction whose target was the CPU shared memory.

Remedy

- Read the individual information of the error using a programming tool, check the FROM/TO instruction that corresponds to that numerical value (program error location), and correct when necessary.
- The intelligent function module that was accessed is experiencing a hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 2111

Details

- The location designated by a link direct device (J* \ *) is not a network module.
- The I/O module (intelligent function module) was nearly removed, completely removed, or mounted during running.

Remedy

- Read the individual information of the error using a programming tool, check the FROM/TO instruction that corresponds to that numerical value (program error location), and correct when necessary.
- The intelligent function module that was accessed is experiencing a hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 2112

Details

- The module other than intelligent function module is specified by the intelligent function module/special function module dedicated instruction. Or, it is not the corresponding intelligent function module.
- There is no network No. specified by the network dedicated instruction. Or the relay target network does not exist.

Remedy

- Read the individual information of the error using a programming tool and check the special function module dedicated instruction (network instruction) that corresponds to the value (program error part) to make modification.

A01 MULTI CPU ERROR 2116

Details

- An instruction that does not allow the module under the control of another CPU to be specified is being used for a similar task.

Remedy

- Read the individual information of the error using a programming tool and check the special function module dedicated instruction (network instruction) that corresponds to the value (program error part) to make modification.

A01 MULTI CPU ERROR 2124

Details

- A module is installed to the 65th or higher slot.
- A module is installed to the slot whose number is greater than the number of slots specified in the base setting.
- A module is installed to the location corresponding to the I/O points of the 4,096 or greater.
- A module is installed to the slot whose assigned I/O range includes the limit of.

Remedy

- Remove the module installed to the 65th or later slot.
- Remove the module installed at the slot whose number is greater than the number of slots specified in the base setting.
- Remove the module installed to the location of I/O points, 4,096 or greater.
- Replace the last module to a module which does not exceed the 4,096th point.

A01 MULTI CPU ERROR 2125

Details

- A module which the QCPU cannot recognize has been installed.
- There was no response form the intelligent function module/special function module.

Remedy

- Install a usable module.
- The intelligent function module is experiencing a hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 2140

Details

- Motion CPU (Q172LX, Q172EX (-S1, S2, S3) or Q173PX(-S1)) is mounted to the CPU slot or slot No. 0 to 2.(QnUD(H)CPU does not lead this error.)

Remedy

- Remove the motion CPU that is mounted to the CPU slot or slot No. 0 to 2.

A01 MULTI CPU ERROR 2150

Details

- In a multiple CPU system, the control CPU of the intelligent function module incompatible with the multiple CPU system is set to other than CPU No.1.

Remedy

- Change the intelligent function module for the one compatible with the multiple CPU system (function version B).
- Change the setting of the control CPU of the intelligent function module incompatible with the multiple CPU system to CPU No.1.

A01 MULTI CPU ERROR 2200

Details

- No parameter file is found all through the drives where the parameter should be validated.

Remedy

- Write the parameter file to PLC of the drive that validates the parameter.

A01 MULTI CPU ERROR 2210

Details

- The contents of the boot file are incorrect.

Remedy

- Check the boot setting.

A01 MULTI CPU ERROR 2211

Details

- File formatting failed at a boot.

Remedy

- Reboot.
- CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 2300

Details

- A memory card was removed without switching the memory card in/out switch OFF.
- The memory card in/out switch is turned ON although a memory card is not actually installed.

Remedy

- Remove memory card after placing the memory card in/out switch OFF.
- Turn on the card insert switch after inserting a memory card.

A01 MULTI CPU ERROR 2301

Details

- The memory card has not been formatted.
- Memory card format status is incorrect.

Remedy

- Format memory card.
- Reformat memory card.

Details

- A fault of the SRAM card was detected.
  (This error occurs when the automatic formatting is not set.)

Remedy

- Replace the battery of SRAM card and then format the SRAM card.

A01 MULTI CPU ERROR 2302

Details

- A memory card that cannot be used with the CPU module has been installed.

Remedy

- Format memory card.
- Reformat memory card.
- Check memory card.

A01 MULTI CPU ERROR 2400

Details

- The file designated at the PLC file settings in the parameters cannot be found.

Remedy

- Read the individual information of the error using a programming tool, check to be sure that the parameter drive name and file name correspond to the numerical values there (parameter number), and correct.
- Create a file created using parameters, and load it to the CPU module.

A01 MULTI CPU ERROR 2401

Details

- Program memory capacity was exceeded by performing boot operation.

Remedy

- Check and correct the parameters (boot setting).
- Delete unnecessary files in the program memory.
- Choose "Clear program memory" for boot in the parameter so that boot is started after the program memory is cleared.

Details

- The file designated by the parameter cannot be created (even during the boot operation).

Remedy

- Read the individual information of the error using a programming tool, check to be sure that the parameter drive name and file name correspond to the numerical values there (parameter number), and correct.
- Check the space remaining in the memory card.

A01 MULTI CPU ERROR 2410

Details

- The specified program does not exist in the program memory.
- The file specified by the sequence program does not exist.

Remedy

- Read the individual information of the error using a programming tool, check to be sure that the program corresponds to the numerical values there (program location), and correct.
- Create a file created using parameters, and load it to the CPU module.

A01 MULTI CPU ERROR 2411

Details

- The file is a comment file or the like, which cannot be designated by the sequence program.
- The specified program exists in the program memory, but has not been registered in the program setting of the Parameter dialog box.
  (This error may occur when the ECALL, EFCALL, PSTOP, PSCAN, POFF or PLOW instruction is executed.)

Remedy

- Read the individual information of the error using a programming tool, check to be sure that the program corresponds to the numerical values there (program location), and correct.

A01 MULTI CPU ERROR 2500

Details

- There is a program file that uses a device that is out of the range set in the PLC parameter device setting.
- After the PLC parameter device setting is changed, only the parameter is written into the PLC.

Remedy

- Read the common information of the error using a programming tool, check to be sure that the parameter device allocation setting and the program file device allocation correspond to the numerical values there (file name), and correct if necessary.
- If PLC parameter device setting is changed, batch-write the parameter and program file into the PLC.

Details

- After the PLC parameter index modification setting is changed, only the parameter is written into the PLC.

Remedy

- If PLC parameter index modification setting is changed, batch-write the parameter and program file into the PLC.

A01 MULTI CPU ERROR 2501

Details

- There are multiple program files although "none" has been set at the PLC parameter program settings.

Remedy

- Edit the PLC parameter program setting to "yes".
  Alternatively, delete unneeded programs.

A01 MULTI CPU ERROR 2502

Details

- The program file is incorrect.
  Alternatively, the file contents are not those of a sequence program.

Remedy

- Check whether the program version is ***.QPG, and check the file contents to be sure they are for a sequence program.

A01 MULTI CPU ERROR 2503

Details

- There are no program files at all.
  (The common information displays the drive name only.)

Remedy

- Check program configuration.
- Check parameters and program configuration.

A01 MULTI CPU ERROR 3000

Details

- In a multiple CPU system, the intelligent function module under control of another CPU is specified in the interrupt pointer setting of the PLC parameter.

Remedy

- Specify the head I/O number of the intelligent function module under control of the host CPU.
- Delete the interrupt pointer setting of the parameter.

Details

- The PLC parameter settings for timer time limit setting, the RUN-PAUSE contact, the common pointer No., number of vacant slots, system interrupt settings or service operation settings are outside the range that can be used by the CPU module.

Remedy

- Read the detailed information of the error using a programming tool, check the parameter items corresponding to those numerical values (parameter numbers), and correct when necessary.
- If the error is still generated following the correction of the parameter settings, the possible cause is the memory error of the CPU module's built-in RAM, program memory or the memory card. Contact your local service center.

Details

- [Memory card (ROM)] is designated as a drive for the file register and "Use the following file" or "Use the same file name as the program" is selected in the PLC file settings, while ATA card is actually set to the PC card slot.

Remedy

- Read the detailed information of the error using a programming tool, check the parameter items corresponding to those numerical values (parameter numbers), and correct when necessary.
- If the error is still generated following the correction of the parameter settings, the possible cause is the memory error of the CPU module's built-in RAM, program memory or the memory card. Contact your local service center.

A01 MULTI CPU ERROR 3001

Details

- The parameter settings are corrupted.

Remedy

- Read the detailed information of the error using a programming tool, check the parameter items corresponding to those numerical values (parameter numbers), and correct when necessary.
- If the error is still generated following the correction of the parameter settings, the possible cause is the memory error of the CPU module's built-in RAM, program memory or the memory card. Contact your local service center.

A01 MULTI CPU ERROR 3002

Details

- The designated memory has no file register file, although “Use the following file” and no capacity have been set for the file register in the PLC parameter, PLC file settings.

Remedy

- Read the detailed information of the error using a programming tool, check the parameter items corresponding to those numerical values (parameter numbers), and correct when necessary.
- If the error is still generated following the correction of the parameter settings, the possible cause is the memory error of the CPU module's built-in RAM, program memory or the memory card. Contact your local service center.

A01 MULTI CPU ERROR 3003

Details

- The number of devices set at the PLC parameter device settings exceeds the possible CPU module range.

Remedy

- Read the detailed information of the error using a programming tool, check the parameter items corresponding to those numerical values (parameter numbers), and correct when necessary.
- If the error is still generated following the correction of the parameter settings, the possible cause is the memory error of the CPU module's built-in RAM, program memory or the memory card. Contact your local service center.

Details

- The automatic refresh range of the multiple CPU system exceeded the file register capacity.

Remedy

- Change the file register file for the one refresh-enabled in the whole range.

A01 MULTI CPU ERROR 3004

Details

- The parameter file is incorrect.
  Alternatively, the contents of the file are not parameters.

Remedy

- Check whether the parameter file version is ***QPA, and check the file contents to be sure they are parameters.

A01 MULTI CPU ERROR 3012

Details

- Multiple CPU setting or control CPU setting differs from that of the reference CPU settings in a multiple CPU system.

Remedy

- Match the multiple CPU setting or control CPU setting in the PLC parameter with that of the reference CPU (CPU No.1) settings.

A01 MULTI CPU ERROR 3013

Details

Multiple CPU automatic refresh setting is any of the followings in a multiple CPU system.

- When a bit device is specified as a refresh device, a number other than a multiple of 16 is specified for the refresh-starting device.
- The device specified is other than the one that may be specified.
- The number of send points is an odd number.
- The total number of transmission points is greater than the maximum number of refresh points.
- The automatic refresh parameter was set for the CPU that does not support the automatic refresh.
- The device to transmit is not set.

Remedy

Check the following in the multiple CPU automatic refresh parameters and make correction.

- Specify the device that may be specified for the refresh device.
- Set the number of send points to an even number.
- The total number of transmission points is within the maximum number of refresh points.
- Set the parameter to the CPU that supports the automatic refresh.
- Specify the device to transmit.

A01 MULTI CPU ERROR 3015

Details

- In a multiple CPU system, the parameter settings are different between the modules.

Remedy

- Read the error details using a programming tool, check and correct the details of the parameter that corresponds to the read value (parameter No. or CPU No.), as well as the parameter of the target module.

A01 MULTI CPU ERROR 3100

Details

- The number of actually installed modules is different from that designated in the number of modules setting parameter of MELSECNET/H.
- The head I/O number of actually installed modules is different from that designated in the network parameter of MELSECNET/H.
- Some data in the parameters cannot be handled.
- The station type of MELSECNET/H has been changed while the power is ON. (Reset and restart is required to change the station type.)

Remedy

- Check the network parameters and actual mounting status, and if they differ, make them matched.
  If any network parameter has been corrected, write it to the CPU module.
- Check the extension base unit stage No. setting.
- Check the connection status of the extension base units and extension cables.
  When the GOT is bus-connected to the main base unit and extension base units, also check the connection status.
- If the error occurs after the above checks, the possible cause is a hardware fault. Contact your local service center.

Details

- In a multiple CPU system, the MELSECNET/H under control of another CPU is specified as the head I/O number in the network setting parameter of the MELSECNET/H.

Remedy

- Delete the MELSECNET/H network parameter of the MELSECNET/H under control of another CPU.
- Change the setting to the head I/O number of the MELSECNET/H under control of the host CPU.

Details

- The network parameters of the MELSECNET/H operating as a normal station was changed to the control station, or the network parameter of the MELSECNET/H operating as a control station was changed to a normal station.
  (The network parameter is updated on the module side by resetting.)

Remedy

- Reset the CPU module.

A01 MULTI CPU ERROR 3101

Details

- The link refresh range exceeded the file register capacity.

Remedy

- Set either the larger capacity for file register or the narrower range for link refresh.

Details

- The network No. specified by a network parameter is different from that of the actually mounted network.
- The head I/O No. specified by a network parameter is different from that of the actually mounted I/O unit.
- The network class specified by a network parameter is different from that of the actually mounted network.
- The network refresh parameter of the MELSECNET/H, MELSECNET/10 is out of the specified area.

Remedy

- Check the network parameters and mounting status, and if they differ, match the network parameters and mounting status. If any network parameter has been corrected, write it to the CPU module.
- Confirm the setting of the number of extension stages of the extension base units.
- Check the connection status of the extension base units and extension cables.
  When the GOT is bus-connected to the main base unit and extension base units, also check their connection status.
- If the error occurs after the above checks, the cause is a hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 3102

Details

- The result after checking network parameters in the network module shows an error.
- The parameters specific to MELSECNET/H and MELSECNET/10 are not normal.

Remedy

- Correct and write the network parameters.
- If the error occurs after correction, it suggests a hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 3103

Details

- Although the number of modules has been set to 1 or greater number in the Ethernet module count parameter setting, the number of actually mounted module is 0.
- The head I/O No. of the Ethernet network parameter differs from the I/O No. of the actually mounted module.

Remedy

- Correct and write the network parameters.
- If the error occurs after correction, it suggests a hardware fault. Contact your local service center.

Details

- In a multiple CPU system, the Q series Ethernet interface module under control of another station is specified to the head I/O number of the Ethernet network parameter.

Remedy

- Delete the Ethernet network parameter of MELSECNET/H module under control of another station.
- Change the setting to the head I/O number of the MELSECNET/H module under control of the host station.

A01 MULTI CPU ERROR 3104

Details

- The Ethernet and MELSECNET/H use the same network number.
- The network number, station number or group number set in the network parameter is out of range.
- The specified I/O No. is outside the range of the used CPU module.
- The Ethernet-specific parameter setting is not normal.

Remedy

- Correct and write the network parameters.
- If the error occurs after correction, it suggests a hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 3105

Details

- Though the number of CC-Link modules set in the network parameters is one or more, the number of actually mounted modules is zero.
- The head I/O number in the common parameters is different from that of the actually mounted module.
- The station type of the CC-Link module count setting parameters is different from that of the actually mounted station.

Remedy

- Correct and write the network parameters.
- If the error occurs after correction, it suggests a hardware fault. Contact your local service center.

Details

- In a multiple CPU system, the Q series CC-Link module under control of another station is specified as the head I/O number of the CC-Link network parameter.

Remedy

- Delete the CC-Link network parameter of the Q series CC-Link module under control of another station.
- Change the setting to the head I/O number of the Q series CC-Link module under control of the host station.

A01 MULTI CPU ERROR 3106

Details

- The CC-Link link refresh range exceeded the file register capacity.

Remedy

- Change the file register file for the one refresh-enabled in the whole range.

Details

- The network refresh parameter for CC-Link is out of range.

Remedy

- Check the parameter setting.

A01 MULTI CPU ERROR 3107

Details

- The CC-Link parameter setting is incorrect.
- The set mode is not allowed for the version of the mounted CC-Link module.

Remedy

- Check the parameter setting.

A01 MULTI CPU ERROR 3300

Details

- The head I/O number in the intelligent function module parameter set on GX Configurator differs from the actual I/O number.

Remedy

- Check the parameter setting.

A01 MULTI CPU ERROR 3301

Details

- The refresh setting of the intelligent function module exceeded the file register capacity.

Remedy

- Change the file register file for the one which allows refresh in the whole range.

Details

- The intelligent function module's refresh parameter setting is outside the available range.

Remedy

- Check the parameter setting.

A01 MULTI CPU ERROR 3302

Details

- The intelligent function module's refresh parameter is abnormal.

Remedy

- Check the parameter setting.

A01 MULTI CPU ERROR 3303

Details

- In a multiple CPU system, the automatic refresh setting or other parameter setting was made to the intelligent function module under control of another station.

Remedy

- Delete the automatic refresh setting or other parameter setting of the intelligent function module under control of another CPU.
- Change the setting to the automatic refresh setting or other parameter setting of the intelligent function module under control of the host CPU.

A01 MULTI CPU ERROR 3400

Details

- The head I/O number of the target module in the remote password file is set to other than 0H to 0FF0H.

Remedy

- Change the head I/O number of the target module to within the 0H to 0FF0H range.

A01 MULTI CPU ERROR 3401

Details

Position specified as the head I/O number of the remote password file is incorrect due to one of the following reasons:

- Module is not loaded.
- Other than a the intelligent function module (I/O, A, QnA module)
- Intelligent function module other than the Q series serial communication module, modem interface module or Ethernet module
- Q series serial communication module or Ethernet module of function version A
  The intelligent function module that allows the remote password setting is not mounted.

Remedy

- Mount the intelligent function module (QJ71C24(CMO) or QJ71E71, with version B or later), which allows the remote password setting, in the position specified in the head I/O No. of the remote password file.

Details

The Q series serial communication module, modem interface module or Ethernet module of function version B or later controlled by another CPU was specified in a multiple CPU system.

Remedy

- Change it for the Ethernet module of function version B or later connected by the host CPU.
- Delete the remote password setting.

A01 MULTI CPU ERROR 4000

Details

- The program contains an instruction code that cannot be decoded.
- An unusable instruction is included in the program.

Remedy

- Read the common information of the error using a peripheral device, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4002

Details

- The exclusive instruction designated by the program has an incorrect instruction name.
- The exclusive instruction specified in the program cannot be executed by the specified module.

Remedy

- Read the common information of the error using a peripheral device, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4003

Details

- The exclusive instruction designated by the program has an incorrect number of devices.

Remedy

- Read the common information of the error using a peripheral device, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4004

Details

- The exclusive instruction designated by the program a device which cannot be used.

Remedy

- Read the common information of the error using a peripheral device, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4010

Details

- There is no END (FEND) instruction in the program.

Remedy

- Read the common information of the error using a peripheral device, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4020

Details

- The total number of internal file pointers used by the program exceeds the number of internal file pointers set in the parameters.

Remedy

- Read the common information of the error using a peripheral device, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4021

Details

- The pointer Nos. overlap between common and local pointers, which are assigned to files.

Remedy

- Read the common information of the error using a peripheral device, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4030

Details

- The allocation pointer Nos. assigned by files overlap.

Remedy

- Read the common information of the error using a peripheral device, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4100

Details

- The instruction cannot process the contained data.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4101

Details

- The designated device number for data processed by the instruction exceeds the usable range.
- Alternatively, the stored data or constants for the devices designated by the instruction exceeds the usable range.
- In the settings of write to the host CPU shared memory, the write designation disabled area is specified as the write destination address.
- The stored data in the file register designated by the instruction exceeds the usable range. If not so, no file register is set. The ranges of stored data overlap among the devices designated by the instruction.
- The stored data in the file register designated by the instruction exceeds the usable range. If not so, no file register is set. The number of points of devices designated by the instruction exceeds the usable number of device points.
- The stored data in the file register designated by the instruction exceeds the usable range. If not so, no file register is set.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4102

Details

- The network No. or station No. specified for the network dedicated instruction is wrong.
- The link direct device (J(\W() setting is incorrect.
- The module No./network No/number of character strings specified for the exclusive instruction is beyond the allowed range.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

Details

- In a multiple CPU system, the link direct device (J(\G() was specified for the network module under control of another station.

Remedy

- Delete from the program the link direct device which specifies the network module under control of another CPU.
- Using the link direct device, specify the network module under control of the host CPU.

Details

- The character string designated with the exclusive instruction (enclosed in "") is not available.

Remedy

- Read the common information of the error using GX Developer, and check and correct the error step corresponding to that value (program error location).

A01 MULTI CPU ERROR 4111

Details

- An attempt was made to perform write/read to/from the CPU shared memory write/read disable area of the host station CPU module with the instruction.

Remedy

- Read the common information of the error using GX Developer, and check and correct the error step corresponding to that value (program error location).

A01 MULTI CPU ERROR 4112

Details

- The CPU module that cannot be specified with the multiple CPU dedicated instruction was specified.

Remedy

- Read the common information of the error using GX Developer, and check and correct the error step corresponding to that value (program error location).

A01 MULTI CPU ERROR 4140

Details

- The operation was executed with the input data that has any specific number (“-0”, unnormalized numbers, nonnumeric characters, ± ∞).

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location) and correct the problem.

A01 MULTI CPU ERROR 4141

Details

- An overflow occurred during the operation.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location) and correct the problem.

A01 MULTI CPU ERROR 4200

Details

- No NEXT instruction was executed following the execution of a FOR instruction. Alternatively, there are fewer NEXT instructions than FOR instructions.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location) and correct the problem.

A01 MULTI CPU ERROR 4201

Details

- A NEXT instruction was executed although no FOR instruction has been executed. Alternatively, there are more NEXT instructions than FOR instructions.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location) and correct the problem.

A01 MULTI CPU ERROR 4202

Details

- More than 16 nesting levels are programmed for FOR instructions.

Remedy

- Keep nesting levels at 16 or under.

A01 MULTI CPU ERROR 4203

Details

- A BREAK instruction was executed although no FOR instruction has been executed prior to that.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4210

Details

- The CALL instruction is executed, but there is no subroutine at the specified pointer.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4211

Details

- There was no RET instruction in the executed subroutine program.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4212

Details

- The RET instruction was before the FEND instruction in the main program.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4213

Details

- More than 16 nesting levels are programmed for CALL instructions.

Remedy

- Keep nesting levels at 16 or under.

A01 MULTI CPU ERROR 4220

Details

- Though an interrupt input occurred, the corresponding interrupt pointer does not exist.

Remedy

- Check whether the interrupt pointer No., specified in the parameter setting, exists in the program.
- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4221

Details

- An IRET instruction does not exist in the executed interrupt program.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4223

Details

- An IRET instruction exists before the FEND instruction of the main program.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4231

Details

- The number of IX and IXEND instructions is not equal.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4350

Details

An incorrect CPU module was designated by the exclusive instruction for multi-CPUs high-speed communication set in the program.

- The designated module has already been reserved.
- The designated module has not been mounted.
- 'Head I/O No. of target CPU module divided by 16' (n1) is not within 3E0H to 3E3H.
- The designated CPU module cannot execute the instruction.
- The instruction was executed in the single CPU system.
- The designated module is the host CPU module.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4351

Details

The designated CPU module cannot execute the exclusive instruction for multi-CPUs high-speed communication set in the program.

- The instruction name is not correct.
- The designated instruction is not supported by the CPU module.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4352

Details

- An incorrect number of devices were designated with the exclusive instruction for multi-CPUs high-speed communication set in the program.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4353

Details

- An unusable device was designated with the exclusive instruction for multi-CPUs high-speed communication set in the program.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4354

Details

- An unusable character string was designated with the exclusive instruction for multi-CPUs high-speed communication set in the program.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 4355

Details

- An invalid number of read/write data (number of request/response data) was designated with the exclusive instruction for multi-CPUs high-speed communication set in the program.

Remedy

- Read the common information of the error using a programming tool, check error step corresponding to its numerical value (program error location), and correct the problem.

A01 MULTI CPU ERROR 5000

Details

- The scan time of the initial execution type program exceeded the initial execution watch time specified in the PLC RAS setting of the PLC parameter dialog box.

Remedy

- Read the individual information of the error with a programming tool, check its value (time), and shorten the scan time.

A01 MULTI CPU ERROR 5001

Details

- The program scan time exceeded the WDT value specified in the PLC RAS setting of the PLC parameter dialog box.

Remedy

- Read the individual information of the error with a programming tool, check its value (time), and shorten the scan time.
- Resolve the endless loop caused by jump transition.

A01 MULTI CPU ERROR 5010

Details

- The program scan time exceeded the constant scan time specified in the PLC RAS setting of the PLC parameter dialog box.
- The low speed program execution time specified in the PLC RAS setting of the PLC parameter dialog box exceeded the excess time of the constant scan.

Remedy

- Review the constant scan setting time.
- Review the constant scan time and low speed program execution time in the PLC parameter so that the margin time of constant scan may be fully reserved.

A01 MULTI CPU ERROR 7000

Details

- In the operating mode of a multiple CPU system, a CPU error occurred at the CPU where "All station stop by stop error of CPU " was selected.
- In a multiple CPU system, a CPU module incompatible with the multiple CPU system was mounted.

Remedy

- Read the individual information of the error using the GX Developer, check the error of the PLC resulting in CPU module fault, and remove the error.

A01 MULTI CPU ERROR 7002

Details

- There is no response from the target CPU module in a multiple CPU system during initial communication.
- In a multiple CPU system, a CPU module incompatible with the multiple CPU system was mounted.

Remedy

- Reset restart the CPU module. If the same error is displayed again, this suggests the hardware fault of any of the CPU modules.
  Contact your local service center.

A01 MULTI CPU ERROR 7004

Details

- In a multiple CPU system, a data error occurred in communication between the CPU modules.

Remedy

- Check the system configuration to see if modules are mounted in excess of the number of I/O points.
- When there are no problems in the system configuration, this indicates the CPU module hardware s faulty. Contact your local service center.

A01 MULTI CPU ERROR 7010

Details

- In a multiple CPU system, a faulty CPU module was mounted.
- In a multiple CPU system, a CPU module incompatible with the multiple CPU system was mounted.
  (The CPU module compatible with the multiple CPU system was used to detect an error.)

Remedy

- Read the individual information of the error using GX Developer, and replace the faulty CPU module.

A01 MULTI CPU ERROR 7011

Details

Either of the following settings was made in a multiple CPU system.

- Multiple CPU automatic refresh setting was made for the inapplicable CPU module.
- 'I/O sharing when using multiple CPUs' setting was made for the inapplicable CPU module.

Remedy

- Replace the CPU module to the one applicable to the 'I/O sharing when using multiple CPUs'.

Details

The system configuration is not applicable to the multi-CPUs high-speed communication.

- QnUD(H)CPU is not used for CPU No.1.
- Main base module for multi-CPUs high-speed communication (Q3(DB) is not used.

Remedy

- Change the system configuration to be applicable to the multi-CPUs high-speed communication.

A01 MULTI CPU ERROR 7012

Details

The CPU module that cannot configure QnUD(H)CPU nor multi-CPU system is mounted on the CPU slot or slot No. 0 to 2.

- Qn(H)CPU or QnPHCPU is mounted.
- PC CPU or C language-based controller is mounted.

Remedy

- Remove the CPU module that does not support QnUD(H)CPU.

A01 MULTI CPU ERROR 7013

Details

- The motion CPU (Q172/3(H)CPU(N)), which cannot configure QnUD(H)CPU nor multi-CPU system, is mounted to the CPU slot or slot No. 0 to 2.

(Note) This error may lead the module failure.

Remedy

- Remove the CPU module that does not support QnUD(H)CPU.

A01 MULTI CPU ERROR 7020

Details

- In the operating mode of a multiple CPU system, an error occurred in the CPU where "system stop" was not selected.
  (The CPU module where no error occurred was used to detect an error.)

Remedy

- Read the individual information of the error using a programming tool, check the error of the CPU module resulting in CPU module fault, and remove the error.

A01 MULTI CPU ERROR 7030

Details

- An assignment error occurred in the CPU-mountable slot (CPU slot, I/O slot 0, 1) in excess of the number of CPU modules specified in the multiple CPU setting of the PLC parameter dialog box.

Remedy

- Set the same value to the number of CPU modules specified in the multiple CPU setting of the PLC parameter dialog box and the number of mounted CPU modules (including CPU (empty)).
- Make the type specified in the I/O assignment setting of the PLC parameter dialog box consistent with the CPU module configuration.

A01 MULTI CPU ERROR 7031

Details

- An assignment error occurred within the range of the number of CPUs specified in the multiple CPU setting of the PLC parameter dialog box.

Remedy

- Set the same value to the number of CPU modules specified in the multiple CPU setting of the PLC parameter dialog box and the number of mounted CPU modules (including CPU (empty)).
- Make the type specified in the I/O assignment setting of the PLC parameter dialog box consistent with the CPU module configuration.

A01 MULTI CPU ERROR 7035

Details

- The CPU module has been mounted on the inapplicable slot.

Remedy

- Mount the CPU module on the applicable slot (CPU slot or I/O slot 0, 1).
- Remove the CPU from the inapplicable slot.

A01 MULTI CPU ERROR 8031

Details

- An error of a stored file (valid parameter file) was detected.

Remedy

- Write the file indicated by the individual information SD17 to SD22 into the individual information SD16(L), and turn the CPU power OFF and ON or execute 'reset' and 'reset canceling'.
- If the same error is displayed again, this suggests a CPU module hardware fault. Contact your local service center.

A01 MULTI CPU ERROR 9000

Details

- Annunciator (F) was set ON

Remedy

- Read the individual information of the error using a programming tool, and check the program corresponding to the numerical value (annunciator number).

A01 MULTI CPU ERROR 10001-10999

Details

- Motion controller in the multi-CPU system has an error.
  (QnUD(H)CPU does not lead this error.)

Remedy

- Read the error details using a programming tool for the motion controller, and remove the error factor.

A01 MULTI CPU ERROR 11000-11999

Details

- CNC CPU in the multi-CPU system has an error.
  (QnUD(H)CPU does not lead this error.)

Remedy

- See the error details on the NC display and remove the error factor.

A01 MULTI CPU ERROR 12000-12999

Details

- RC CPU in the multi-CPU system has an error.
  (QnUD(H)CPU does not lead this error.)

Remedy

(Reserved for RC.)

Appendix 7.11 Network Errors (L)

L10 DN INIT. ERR. 1 36 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FFH.

- The value of the local node No. (MAC ID) is out of range.
- The value of the mode switch is out of range.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set the local node number between 0 and 63.
- Set the mode switch to other than D - F.

L10 DN INIT. ERR. 1 39 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FFH.

- Two or more modes that have the same node No. (MAC ID) exist in the network.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set non-duplicate node Nos.

L10 DN INIT. ERR. 1 E0 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FFH.

- Network power is not being supplied.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Supply the network power (24VDC).

L10 DN INIT. ERR. 1 E1 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FFH.

- Other modules are not found in the network.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Connect other modules in the network.

L10 DN INIT. ERR. 1 F0 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FFH.

- Node No. setting switch or mode switch setting was changed during operation.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Return the node No. setting switch or mode switch setting to be the original setting.

L11 DN INIT. ERR. 2 02 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The number of input points and output points of the slave node set by parameters are both "0".

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set the number of input points and output points according to the slave node specifications.

L11 DN INIT. ERR. 2 03 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The lower byte of the slave node No. in the buffer memory is out of range.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set it between 0 and 63.

L11 DN INIT. ERR. 2 04 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The higher byte of the slave node No. in the buffer memory is out of range.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set it between 01H and 04H, or to 80H.

L11 DN INIT. ERR. 2 05 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The connection type in the buffer memory is out of range.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set it to one of the following: 0001H, 0002H, 0004H, or 0008H.

L11 DN INIT. ERR. 2 06 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- A slave node having the same node No. as the node No. of the local node is set in the buffer memory.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set non-duplicate node Nos. in all nodes.

L11 DN INIT. ERR. 2 07 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- No slave node has been set.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set at least one slave node.

L11 DN INIT. ERR. 2 08 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The total input data length of all slave nodes is too long.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Reduce the total data length of all slave nodes to 512 bytes or less.

L11 DN INIT. ERR. 2 09 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The total output data length of all slave nodes is too long.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Reduce the total data length of all slave nodes to 512 bytes or less.

L11 DN INIT. ERR. 2 0A (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The watchdog timeout action value in a parameter is invalid.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set it to one of the following: 0000H, 0001H, 0002H, or 0003H.

L11 DN INIT. ERR. 2 0B (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The expected packet rate value in the buffer memory is smaller than the production inhibit time value.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Change the value so that the expected packet rate is greater than or equal to the production inhibit time value.

L11 DN INIT. ERR. 2 0C (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- Flash ROM checksum error (parameter area for the master function)

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Save the parameters again.
- Do not turn OFF the power or reset while saving the parameters.

L11 DN INIT. ERR. 2 0D (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- Flash ROM checksum error (parameter area for the slave function)

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Save the parameters again.
- Do not turn OFF the power or reset while saving the parameters.

L11 DN INIT. ERR. 2 0E (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- Flash ROM checksum error (auto communication start setting area)

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Save the parameters again.
- Do not turn OFF the power or reset while saving the parameters.

L11 DN INIT. ERR. 2 0F (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- Flash ROM all clear error

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Save the parameters again.
- Do not turn OFF the power or reset while clearing all parameters.

L11 DN INIT. ERR. 2 10 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The number of input points per slave node has exceeded 256 bytes.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Correct the number of input points per slave node to 256 bytes or less.

L11 DN INIT. ERR. 2 11 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The number of output points per slave node has exceeded 256 bytes.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Correct the number of output points per slave node to 256 bytes or less.

L11 DN INIT. ERR. 2 15 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The production inhibit time value was set to 0 ms (setting value "1") in cyclic.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set the production inhibit time value to a value other than 0 ms.

L11 DN INIT. ERR. 2 16 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- Slave nodes that were set by parameters are all reserved nodes.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set the parameters according to the slave nodes connected to the network.

L11 DN INIT. ERR. 2 80 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The number of reception bytes of the slave function is out of range.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set it within the range between 0 and 128 bytes.

L11 DN INIT. ERR. 2 81 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The number of transmission bytes of the slave function is out of range.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set it within the range between 0 and 128 bytes.

L11 DN INIT. ERR. 2 82 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The numbers of transmission bytes and reception bytes of the slave function are both set to "0".

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set either the number of transmission bytes or the number of reception bytes to a value other than "0".

L11 DN INIT. ERR. 2 A0 (Error-detected module I/O No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is FEH.

- The numbers of I/O points of both the master and slave functions were set to "0" when both the master and slave functions were used.

"Error-detected module I/O No." shows the error-detected module I/O No. with the last digit removed.

Remedy

- Set the number of I/O points of the slave node in a master function parameter.
- Set the number of transmission/reception bytes in a slave function parameter. (Be sure to set either the master function or slave function.)

L12 DN LINK ERROR 01 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- A network problem was detected after communication was started.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check that the cable is connected correctly.

L12 DN LINK ERROR 1E (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- Slave node did not respond.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check the entire network and slave node statuses such as whether or not the MAC ID and baud rate are set correctly, a slave node is down, or a terminal resistor is disconnected.

L12 DN LINK ERROR 20 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- Slave node responded with a non-prescribed error.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Read the communication error information, and take an appropriate action according to the error information.

L12 DN LINK ERROR 23 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- Slave node responded with an error when establishing a connection.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Read the communication error information, and take an appropriate action according to the error information.

L12 DN LINK ERROR 24 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- The input data size of a parameter is different from the size of the actual slave node.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check the slave node manual and set the correct input data size.

L12 DN LINK ERROR 25 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- The output data size of a parameter is different from the size of the actual slave node.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check the slave node manual and set the correct output data size.

L12 DN LINK ERROR 26 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- Response data of a function that is not supported by the QJ71DN91 was received.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check the slave node manual, and change the setting so that any data of functions not supported by the QJ71DN91 will not be sent by the slave node.
- Check the entire network and slave node statuses such as whether or not a terminal resistor is disconnected.

L12 DN LINK ERROR 27 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- The connection is set to the mode that was already specified.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check the entire network and slave node statuses such as whether or not a terminal resistor is disconnected.

L12 DN LINK ERROR 28 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- Unexpected invalid data was received when establishing a connection.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check the entire network and slave node statuses such as whether or not a terminal resistor is disconnected.

L12 DN LINK ERROR 29 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- Connection has already been established with that slave node.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Wait a while, and reset the slave node if the connection cannot be established.

L12 DN LINK ERROR 2A (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- The data length of a polling response is different from the data length read from the slave node when establishing a connection.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check the entire network and slave node statuses such as whether or not a terminal resistor is disconnected.

L12 DN LINK ERROR 2B (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- The first division data was received twice in the division reception of a polling response.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check the entire network and slave node statuses such as whether or not a terminal resistor is disconnected.

L12 DN LINK ERROR 2C (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- The received division data No. is different from the expected No. in the division reception of a polling response.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check the entire network and slave node statuses such as whether or not a terminal resistor is disconnected.

L12 DN LINK ERROR 2D (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- Intermediate data or last data was received before receiving the first division data in the division reception of a polling response.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check the entire network and slave node statuses such as whether or not a terminal resistor is disconnected.

L12 DN LINK ERROR 3B (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- Two or more identical node Nos. (MAC IDs) were detected in parameters.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Two or more slave nodes having the same node No. are set in the parameters. Set the correct node Nos.
- A slave node having the same node No. as that of the local node exists in the parameters.

L12 DN LINK ERROR 47 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- Incorrect connection type was specified.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check that the connection type value is correct.
- Read the communication error information, and take an appropriate action according to the error information.

L12 DN LINK ERROR 80 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- Timeout occurred during the polling connection of the slave function.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Check the entire network and slave node statuses such as whether or not a terminal resistor is disconnected.
- Check the master node status.

L12 DN LINK ERROR 81 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- A connection other than explicit messages and polling was allocated.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Do not allocate I/O connections other than polling.

L12 DN LINK ERROR 82 (Error-detected slave node No.)

Details

A deviceNet initialization error has occurred. The error-detected node No. is other than FFH and FEH.

- The number of reception bytes of polling is greater than the maximum number of reception points.

"Error-detected slave node No." shows the error-detected slave node No. in decimal.

Remedy

- Match the I/O point setting of the master node with that of the QJ71DN91.

L13 DN MESSAGE ERR. 161

Details

A message communication execution error has occurred.

- The specified slave node No. is other than 0 to 63.

Remedy

- Set it between 0 and 63.

Appendix 7.12 Program Errors (P)

(Note) Program error messages are displayed in abbreviation on the screen.

P10 EXCS. AXIS. No.

Details

The number of axis addresses commanded in a block is exceeds the specifications.

Remedy

- Divide the alarm block command into two.
- Check the specifications.

P11 AXIS ADR. ERROR

Details

The axis address commanded by the program does not match any of the ones set by the parameter.

Remedy

- Correct the axis names in the program.

P20 DIVISION ERROR

Details

The issued axis command cannot be divided by the command unit.

Remedy

- Correct the program.

P30 PARITY H

Details

The number of holes per character on the paper tape is even for EIA code and odd for ISO code.

Remedy

- Check the paper tape.
- Check the tape puncher and tape reader.

P31 PARITY V

Details

The number of characters per block on the paper tape is odd.

Remedy

- Make the number of characters per block on the paper tape even.
- Set the parameter parity V selection OFF.

P32 ADDRESS. ERROR

Details

An address not listed in the specifications has been used.

Remedy

- Correct the program address.
- Correct the parameter settings.
- Check the specifications.

P33 FORMAT ERROR

Details

The command format in the program is not correct.

Remedy

- Correct the program.

P34 G-CODE ERROR

Details

The commanded G code is not in the specifications.
An illegal G code was commanded during the coordinate rotation command (G68).

Remedy

- Correct the G code address in the program.

Details

G51.2 or G50.2 was commanded when "#1501 polyax (Rotational tool axis number)" was set to "0".
G51.2 or G50.2 was commanded when the tool axis was set to the linear axis ("#1017 rot (Rotational axis)" is set to"0").

Remedy

- Correct the parameter settings.

P35 CMD-VALUE OVER

Details

The setting range for the addresses has been exceeded.
The program coordinates overflowed because commands to the linear type rotary axis accumulated in one direction.

Remedy

- Correct the program.

P36 PROGRAM END ERR

Details

"EOR" has been read during memory mode.

Remedy

- Enter the M02 and M30 command at the end of the program.
- Enter the M99 command at the end of the subprogram.

P37 PROG. No. ZERO

Details

"0" has been specified for program or sequence No.

Remedy

- Designate program Nos. within a range from 1 to 99999999.
- Designate sequence Nos. within a range from 1 to 99999.
- Add M02 or M03 to the end of the program running in FTP operation.

P39 NO SPEC ERR

Details

- A non-specified G code was commanded.
- The selected operation mode is out of specifications.

Remedy

- Check the specifications.

P45 G-CODE COMB.

Details

The combination of G codes in a block is inappropriate.
A part of unmodal G codes and modal G codes cannot be commanded in a same block.

Remedy

Correct the combination of G codes.
Separate the incompatible G codes into different blocks.

P60 OVER CMP. LENG.

Details

The commanded movement distance is excessive (over 2^{31}).

Remedy

- Correct the command range for the axis address.

P62 F-CMD. NOTHING

Details

- No feed rate command has been issued.
- There is no F command in the cylindrical interpolation or polar coordinate interpolation immediately after the G95 mode is commanded.

Remedy

- The default movement modal command at power ON is G01. This causes the machine to move without a G01 command if a movement command is issued in the program, and an alarm results. Use an F command to specify the feed rate.
- Specify F with a thread lead command.

P65 No G05P3 SPEC

Details

Remedy

- Check whether the specifications are provided for the high-speed mode III.

P70 ARC ERROR

Details

- There is an error in the arc start and end points as well as in the arc center.
- The difference of the involute curve through the start point and the end point is large.
- When arc was commanded, one of the two axes configuring the arc plane was a scaling valid axis.

Remedy

- Correct the numerical values of the addresses that specify the start and end points, arc center as well as the radius in the program.
- Correct the "+" and "-" directions of the address numerical values.
- Check for the scaling valid axis.

P71 ARC CENTER

Details

- An arc center cannot be obtained in R-specified circular interpolation.
- A curvature center of the involute curve cannot be obtained.

Remedy

- Correct the numerical values of the addresses in the program.
- Correct the start and end points if they are inside of the base circle for involute interpolation. When carrying out tool radius compensation, make sure that the start and end points after compensation will not be inside of the base circle for involute interpolation.
- Correct the start and end points if they are at an even distance from the center of the base circle for involute interpolation.

P72 NO HELICAL SPEC

Details

A helical command has been issued though it is out of specifications.

Remedy

- Check whether the specifications are provided for the helical cutting.
- An Axis 3 command has been issued by the circular interpolation command. If there is no helical specification, move the linear axis to the next block.

P90 NO THREAD SPEC

Details

A thread cutting command was issued though it is out of specifications.

Remedy

- Check the specifications.

P93 SCREW PITCH ERR

Details

An illegal thread lead (thread pitch) was specified at the thread cutting command.

Remedy

- Correct the thread lead for the thread cutting command.

P111 PLANE CHG (CR)

Details

Plane selection commands (G17, G18, G19) were issued during a coordinate rotation (G68) was being commanded.

Remedy

- Always command G69 (coordinate rotation cancel) after the G68 command, and then issue a plane selection command.

P112 PLANE CHG (CC)

Details

- Plane selection commands (G17, G18, G19) were issued while tool radius compensation (G41, G42) and nose R compensation (G41, G42, G46) commands were being issued.
- Plane selection commands were issued after completing nose R compensation commands when there were no further axis movement commands after G40, and compensation has not been cancelled.

Remedy

- Issue plane selection commands after completing (axis movement commands issued after G40 cancel command) tool radius compensation and nose R compensation commands.

P113 ILLEGAL PLANE

Details

The circular command axis does not correspond to the selected plane.

Remedy

- Select a correct plane before issuing a circular command.

P122 NO AUTO C-OVR

Details

An auto corner override command (G62) was issued though it is out of specifications.

Remedy

- Check the specifications.
- Delete the G62 command from the program.

P130 2nd AUX. ADDR

Details

The 2nd miscellaneous function address, commanded in the program, differs from the address set in the parameters.

Remedy

- Correct the 2nd miscellaneous function address in the program.

P131 NO G96 SPEC

Details

A constant surface speed control command (G96) was issued though it is out of specifications.

Remedy

- Check the specifications.
- Issue a rotation speed command (G97) instead of the constant surface speed control command (G96).

P132 SPINDLE S = 0

Details

No spindle rotation speed command has been issued.

Remedy

- Correct the program.

P133 G96 P-No. ERR

Details

The illegal No. was specified for the constant surface speed control axis.

Remedy

- Correct the parameter settings and program that specify the constant surface speed control axis.

P134 G96 Clamp Err.

Details

The constant surface speed control command (G96) was issued without commanding the spindle speed clamp (G92/G50).

Remedy

Press the reset key and carry out the remedy below.

- Check the program.
- Issue the G92/G50 command before the G96 command.
- Command the constant surface speed cancel (G97) to switch to the rotation speed command.

P150 NO C-CMP SPEC

Details

- Tool radius compensation commands (G41 and G42) were issued though they are out of specifications.
- Nose R compensation commands (G41, G42, and G46) were issued though they are out of specifications.

Remedy

- Check the specifications.

P151 G2, 3 CMP. ERR

Details

A compensation command (G40, G41, G42, G43, G44, or G46) has been issued in the arc modal (G02 or G03).

Remedy

- Issue the linear command (G01) or rapid traverse command (G00) in the compensation command block or cancel block.
  (Set the modal to linear interpolation.)

P152 I.S.P NOTHING

Details

In interference block processing during execution of a tool radius compensation (G41 or G42) or nose R compensation (G41, G42, or G46) command, the intersection point after one block is skipped cannot be determined.

Remedy

- Correct the program.

P153 I.F ERROR

Details

An interference error has occurred while the tool radius compensation command (G41 or G42) or nose R compensation command (G41, G42 or G46) was being executed.

Remedy

- Correct the program.

P155 F-CYC ERR (CC)

Details

A fixed cycle command has been issued in the radius compensation mode.

Remedy

- Issue a radius compensation cancel command (G40) to cancel the radius compensation mode that has been applied since the fixed cycle command was issued.

P156 BOUND DIRECT

Details

A shift vector with undefined compensation direction was found at the start of G46 nose R compensation.

Remedy

- Change the vector to that which has the defined compensation direction.
- Change the tool to that which has a different tip point No.

P157 SIDE REVERSED

Details

During G46 nose R compensation, the compensation direction is reversed.

Remedy

- Change the G command to that which allows the reversed compensation direction (G00, G28, G30, G33, or G53).
- Change the tool to that which has a different tip point No.
- Enable "#8106 G46 NO REV-ERR".

P158 ILLEGAL TIP P.

Details

An illegal tip point No. (other than 1 to 8) was found during G46 nose R compensation.

Remedy

- Correct the tip point No.

P170 NO CORR. NO.

Details

No compensation No. (DOO, TOO or HOO) command was given when the radius compensation (G41, G42, G43 or G46) command was issued. Otherwise, the compensation No. is larger than the number of sets in the specifications.

Remedy

- Add the compensation No. command to the compensation command block.
- Check the number of sets for the tool compensation Nos. and correct the compensation No. command to be within the number of sets.

P171 NO G10 SPEC

Details

Compensation data input by program (G10) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P172 G10 L-No. ERR

Details

An address of G10 command is not correct.

Remedy

- Correct the address L No. of the G10 command.

P173 G10 P-No. ERR

Details

The compensation No. at the G10 command is not within the permitted number of sets in the specifications.

Remedy

- Check the number of sets for the tool compensation Nos. and correct the address P designation to be within the number of sets.

P174 NO G11 SPEC

Details

Compensation data input by program cancel (G11) was commanded though there is no specification of compensation data input by program.

Remedy

- Check the specifications.

P177 LIFE COUNT ACT

Details

Registration of tool life management data with G10 was attempted when the "usage data count valid" signal was ON.

Remedy

- The tool life management data cannot be registered during the usage data count. Turn the "usage data count valid" signal OFF.

P178 LIFE DATA OVER

Details

The number of registration groups, total number of registered tools or the number of registrations per group exceeded the range in the specifications.

Remedy

- Correct the number of registrations.

P179 GROUP NO. ILL.

Details

- A duplicate group No. was found at the registration of the tool life management data with G10.
- A group No. that was not registered was designated during the T****99 command.
- An M code command, which must be issued as a single command, coexists in the same block as that of another M code command.
- The M code commands set in the same group exist in the same block.

Remedy

- Register the tool life data once for one group: commanding with a duplicate group No. is not allowed.
- Correct to the group No.

P180 NO BORING CYC.

Details

A fixed cycle command (G72 - G89) was issued though it is out of specifications.

Remedy

- Check the specifications.
- Correct the program.

P181 NO S-CMD (TAP)

Details

Spindle rotation speed (S) has not been commanded in synchronous tapping.

Remedy

- Command the spindle rotation speed (S) in synchronous tapping.
- When "#8125 Check Scode in G84" is set to "1", enter the S command in the same block where the synchronous tapping command is issued.

P182 SYN TAP ERROR

Details

- Connection to the main spindle unit was not established.
- The synchronous tapping was attempted with the spindle not serially connected under the multiple-spindle control I.

Remedy

- Check connection to the main spindle.
- Check that the main spindle encoder exists.
- Set 1 to the parameter #3024 (sout).

P183 PTC/THD No.

Details

The pitch or number of threads has not been commanded in the tap cycle of a fixed cycle for drilling command.

Remedy

- Specify the pitch data and the number of threads by F or E command.

P184 NO PTC/THD CMD

Details

- The pitch or the number of threads per inch is illegal in the tap cycle of the fixed cycle for drilling command.
- The pitch is too small for the spindle rotation speed.
- The thread number is too large for the spindle rotation speed.

Remedy

- Correct the pitch or the number of threads per inch.

P185 NO SYN TAP SPC

Details

Synchronous tapping cycle (G84/G74) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P187 Tap SP clamp 0

Details

The external spindle speed clamp signal was turned ON without setting the tapping spindle's external spindle speed when commanding the synchronous tapping.

Remedy

- Set the external spindle speed clamp speed parameter.
- Turn the external spindle speed clamp signal OFF.

P190 NO CUTTING CYC

Details

A lathe cutting cycle command was issued though it is out of specifications.

Remedy

- Check the specification.
- Delete the lathe cutting cycle command.

P191 TAPER LENG ERR

Details

In the lathe cutting cycle, the specified length of taper section is illegal.

Remedy

- Set the smaller radius value than the axis travel amount in the lathe cycle command.

P192 CHAMFERING ERR

Details

Chamfering in the thread cutting cycle is illegal.

Remedy

- Set a chamfering amount not exceeding the cycle.

P200 NO MRC CYC SPC

Details

The compound type fixed cycle for turning machining I (G70 to G73) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P201 PROG. ERR (MRC)

Details

- The subprogram, called with a compound type fixed cycle for turning machining I command, has at least one of the following commands: reference position return command (G27, G28, G29, G30); thread cutting (G33, G34); fixed cycle skip-function (G31, G31.n).
- An arc command was found in the first movement block of the finished shape program in compound type fixed cycle for turning machining I.

Remedy

- Delete G27, G28, G29, G30, G31, G33, G34, and fixed cycle G codes from the subprogram called with the compound type fixed cycle for turning machining I commands (G70 to G73).
- Delete G02 and G03 from the first movement block of the finished shape program in compound type fixed cycle for turning machining I.

P202 BLOCK OVR (MRC)

Details

The number of blocks in the shape program of the compound type fixed cycle for turning machining I is over 50 or 200 (the maximum number differs according to the model).

Remedy

- Set a 50/200 or less value for the number of blocks in the shape program called by the compound type fixed cycle for turning machining I commands (G70 to G73). (The maximum number differs according to the model).

P203 CONF. ERR (MRC)

Details

A proper shape will not obtained by executing the shape program for the compound type fixed cycle for turning machining I (G70 to G73).

Remedy

- Correct the shape program for the compound type fixed cycle for turning machining I (G70 to G73).

P204 VALUE ERR (MRC)

Details

A command value of the compound type fixed cycle for turning machining (G70 to G76) is illegal.

Remedy

- Correct the command value of the compound type fixed cycle for turning machining (G70 to G76).

P210 NO PAT CYC SPC

Details

A compound type fixed cycle for turning machining II (G74 to G76) command was commanded though it is out of specifications.

Remedy

- Check the specifications.

P220 NO SPECIAL CYC

Details

There are no special fixed cycle specifications.

Remedy

- Check the specifications.

P221 NO HOLE (S-CYC)

Details

"0" has been specified for the number of holes in special fixed cycle mode.

Remedy

- Correct the program.

P222 G36 ANGLE ERR

Details

A G36 command specifies "0" for angle intervals.

Remedy

- Correct the program.

P223 G12 G13 R ERR

Details

The radius value specified with a G12 or G13 command is below the compensation amount.

Remedy

- Correct the program.

P224 NO G12, G13 SPC

Details

There are no circular cutting specifications.

Remedy

- Check the specifications.

P230 NESTING OVER

Details

Over 8 times of subprogram calls have been done in succession from a subprogram.

- A M198 command was found in the program in the data server.
- The program in the IC card has been called more than once (the program in the IC card can be called only once during nested).

Remedy

- Correct the program so that the number of subprogram calls does not exceed 8 times.

P231 NO N-NUMBER

Details

The sequence No., commanded at the return from the subprogram or by GOTO in the subprogram call, was not set.

Remedy

- Specify the sequence Nos. in the call block of the subprogram.

P232 NO PROGRAM No.

Details

- The machining program has not been found when the machining program is called.
- The file name of the program registered in IC card is not corresponding to O No.

Remedy

- Enter the machining program.
- Check the subprogram storage destination parameters.
- Ensure that the external device (including IC card) that contains the file is mounted.

P241 NO VARI NUMBER

Details

The variable No. commanded is out of the range specified in the specifications.

Remedy

- Check the specifications.
- Correct the program variable No.

P242 EQL. SYM. MSSG.

Details

The "=" sign has not been commanded when a variable is defined.

Remedy

- Designate the "=" sign in the variable definition of the program.

P243 VARIABLE ERR.

Details

An invalid variable has been specified in the left or right side of an operation expression.

Remedy

- Correct the program.

P260 NO COOD-RT SPC

Details

A coordinate rotation command was issued though it is out of specifications.

Remedy

- Check the specifications.

P270 NO MACRO SPEC

Details

A macro specification was commanded though it is out of specifications.

Remedy

- Check the specifications.

P271 NO MACRO INT.

Details

A macro interruption command has been issued though it is out of specifications.

Remedy

- Check the specifications.

P272 MACRO ILL.

Details

An executable statement and a macro statement exist together in the same block.

Remedy

- Place the executable statement and macro statement in separate blocks in the program.

P273 MACRO OVERCALL

Details

The number of macro call nests exceeded the limit imposed by the specifications.

Remedy

- Correct the program so that the macro calls do not exceed the limit imposed by the specifications.

P275 MACRO ARG. EX.

Details

The number of argument sets in the macro call argument type II has exceeded the limit.

Remedy

- Correct the program.

P276 CALL CANCEL

Details

A G67 command was issued though it was not during the G66 command modal.

Remedy

- Correct the program.
- Issue G66 command before G67 command, which is a call cancel command.

P277 MACRO ALM MESG

Details

An alarm command has been issued in #3000.

Remedy

- Refer to the operator messages on the diagnosis screen.
- Refer to the instruction manual issued by the machine tool builder.

P280 EXC. [,]

Details

Over five times have the parentheses "[" or "]" been used in a single block.

Remedy

- Correct the program so that the number of "[" or "]" is five or less.

P281 [,] ILLEGAL

Details

A single block does not have the same number of commanded parentheses "[" as that of "]".

Remedy

- Correct the program so that "[" and "]" parentheses are paired up properly.

P282 CALC. IMPOSS.

Details

The arithmetic formula is incorrect.

Remedy

- Correct the formula in the program.

P283 DIVIDE BY ZERO

Details

The denominator of the division is zero.

Remedy

- Correct the program so that the denominator for division in the formula is not zero.

P290 IF SNT. ERROR

Details

There is an error in the "IF[<conditional>]GOTO(" statement.

Remedy

- Correct the program.

P291 WHILE SNT. ERR

Details

There is an error in the "WHILE[<conditional>]DO(-END(" statement.

Remedy

- Correct the program.

P292 SETVN SNT. ERR

Details

There is an error in the "SETVN(" statement when the variable name setting was made.

Remedy

- Correct the program.
- The number of characters in the variable name of the SETVN statement must be 7 or less.

P293 DO-END EXCESS

Details

The number of DO-END nesting levels in the "WHILE[<conditional>]DO(-END(" statement has exceeded 27.

Remedy

- Correct the program so that the nesting levels of the DO-END statement does not exceed 27.

P294 DO-END MMC.

Details

The DOs and ENDs are not paired off properly.

Remedy

- Correct the program so that the DOs and ENDs are paired off properly.

P295 WHILE/GOTO TPE

Details

There is a WHILE or GOTO statement on the tape during FTP operation.

Remedy

- Apply memory mode operation instead of FTP operation that does not allow the execution of the program with a WHILE or GOTO statement.

P296 NO ADR (MACRO)

Details

A required address has not been specified in the user macro.

Remedy

- Correct the program.

P297 ADR-A ERR.

Details

The user macro does not use address A as a variable.

Remedy

- Correct the program.

P298 PTR OP (MACRO)

Details

User macro G200, G201, or G202 was specified during tape or MDI mode.

Remedy

- Correct the program.

P300 VAR. NAME ERROR

Details

The variable names have not been commanded properly.

Remedy

- Correct the variable names in the program.

P301 VAR. NAME DUPLI

Details

A duplicate variable name was found.

Remedy

- Correct the program so that no duplicate name exists.

P360 NO PROG.MIRR.

Details

A mirror image (G50.1 or G51.1) command has been issued though the programmable mirror image specifications are not provided.

Remedy

- Check the specifications.

P380 NO CORNER R/C

Details

The corner R/C was issued though it is out of specifications.

Remedy

- Check the specifications.
- Delete the corner chamfering/corner rounding command in the program.

P381 NO ARC R/C SPC

Details

Corner chamfering II or corner rounding II was commanded in the arc interpolation block though it is out of specifications.

Remedy

- Check the specifications.

P382 CORNER NO MOVE

Details

The block next to corner chamfering/ corner rounding is not a travel command.

Remedy

- Replace the block succeeding the corner chamfering/ corner rounding command by G01 command.

P383 CORNER SHORT

Details

The travel distance in the corner chamfering/corner rounding command was shorter than the value in the corner chamfering/corner rounding command.

Remedy

- Set the smaller value for the corner chamfering/corner rounding than the travel distance.

P384 CORNER SHORT

Details

The travel distance in the following block in the corner chamfering/corner rounding command was shorter than the value in the corner chamfering/corner rounding command.

Remedy

- Set the smaller value for the corner chamfering/corner rounding than the travel distance in the following block.

P385 G0 G33 IN CONR

Details

A block with corner chamfering/corner rounding was given during G00 or G33 modal.

Remedy

- Correct the program.

P390 NO GEOMETRIC

Details

A geometric command was issued though it is out of specifications.

Remedy

- Check the specifications.

P391 NO GEOMETRIC 2

Details

There are no geometric IB specifications.

Remedy

- Check the specifications.

P392 LES AGL (GEOMT)

Details

The angular difference between the geometric line and line is 1° or less.

Remedy

- Correct the geometric angle.

P393 INC ERR (GEOMT)

Details

The second geometric block has a command with an incremental value.

Remedy

- Issue a command with an absolute value in the second geometric block.

P394 NO G01 (GEOMT)

Details

The second geometric block contains no linear command.

Remedy

- Issue the G01 command.

P395 NO ADRS (GEOMT)

Details

The geometric format is invalid.

Remedy

- Correct the program.

P396 PL CHG. (GEOMT)

Details

A plane switching command was issued during geometric command processing.

Remedy

- Complete the plane switching command before geometric command processing.

P397 ARC ERR (GEOMT)

Details

In geometric IB, the circular arc end point does not contact or cross the next block start point.

Remedy

- Correct the geometric circular arc command and the preceding and following commands.

P398 NO GEOMETRIC1B

Details

A geometric command was issued though the geometric IB specifications are not provided.

Remedy

- Check the specifications.

P420 NO PARAM IN

Details

Parameter input by program (G10) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P421 PRAM. IN ERROR

Details

- The specified parameter No. or set data is illegal.
- An illegal G command address was input in parameter input mode.
- A parameter input command was issued during fixed cycle modal or nose R compensation.
- G10L50, G10L70, G11 were not commanded in independent blocks.

Remedy

- Correct the program.

P430 AXIS NOT RET.

Details

- A command was issued to move an axis, which has not returned to the reference position, away from that reference position.
- A command was issued to an axis removal axis.

Remedy

- Execute reference position return manually.
- Disable the axis removal on the axis for which the command was issued.

P431 NO 2ndREF. SPC

Details

A command for second, third or fourth reference position return was issued though there are no such command specifications.

Remedy

- Check the specifications.

P434 COLLATION ERR

Details

One of the axes did not return to the reference position when the reference position check command (G27) was executed.

Remedy

- Correct the program.

P435 G27/M ERROR

Details

An M command was issued simultaneously in the G27 command block.

Remedy

- Place the M code command, which cannot be issued in a G27 command block, in separate block from G27 command block.

P436 G29/M ERROR

Details

An M command was issued simultaneously in the G29 command block.

Remedy

- Place the M code command, which cannot be issued in a G29 command block, in separate block from G29 command block.

P438 NOT USE (G52)

Details

A local coordinate system command was issued during execution of the G54.1 command.

Remedy

- Correct the program.

P450 NO CHUCK BARR.

Details

The chuck barrier on command (G22) was specified although the chuck barrier is out of specifications.

Remedy

- Check the specifications.

P460 TAPE I/O ERROR

Details

An error has occurred in the tape reader. Otherwise an error has occurred in the printer during macro printing.

Remedy

- Check the power and cable of the connected devices.
- Correct the I/O device parameters.

P461 FILE I/O ERROR

Details

- A file of the machining program cannot be read.

Remedy

- In memory mode, the programs stored in memory may have been destroyed. Output all of the programs and tool data and then format the system.

P501 Cross (G110) impossible

Details

Mixed control (cross axis control) command (G110) was issued in the following modes.

- During nose R compensation mode
- During pole coordinate interpolation mode
- During cylindrical interpolation mode
- During balance cut mode
- During fixed cycle machining mode
- During facing turret mirror image
- During constant surface speed control mode
- During hobbing mode
- During axis name switch

Remedy

- Correct the program.

P503 Illegal G110 axis

Details

- The commanded axis does not exist.
- The mixed control (cross axis control) (G110) was commanded to the axis for which the mixed control (cross axis control) is disabled.
- The number of axes included in the mixed control (cross axis control) (G110) command is exceeding the maximum number of axes per part system.

Remedy

- Correct the program.

P600 NO AUTO TLM.

Details

An automatic tool length measurement command (G37) was issued though it is out of specifications.

Remedy

- Check the specifications.

P601 NO SKIP SPEC.

Details

A skip command (G31) was issued though it is out of specifications.

Remedy

- Check the specifications.

P602 NO MULTI SKIP

Details

A multiple skip command (G31.1, G31.2 or G31.3) was issued though it is out of specifications.

Remedy

- Check the specifications.

P603 SKIP SPEED 0

Details

The skip speed is "0".

Remedy

- Specify the skip speed.

P604 TLM ILL. AXIS command

Details

No axis was specified in the automatic tool length measurement block. Otherwise, two or more axes were specified.

Remedy

- Specify only one axis.

P605 T-CMD IN BLOCK

Details

The T code is in the same block as the automatic tool length measurement block.

Remedy

- Specify the T code before the automatic tool length measurement block.

P606 NO T-CMD BEFOR

Details

The T code was not yet specified in automatic tool length measurement.

Remedy

- Specify the T code before the automatic tool length measurement block.

P607 TLM ILL. SIGNL

Details

The measurement position arrival signal turned ON before the area specified by the D command or "#8006 ZONE d". Otherwise, the signal remained OFF to the end.

Remedy

- Correct the program.

P608 SKIP ERROR (CC)

Details

A skip command was issued during radius compensation processing.

Remedy

- Issue a radius compensation cancel (G40) command or remove the skip command.

P609 NO PLC SKIP

Details

PLC skip has been commanded (L to G31) while PLC skip is out of specifications.

Remedy

- Check the specifications.

P610 ILLEGAL PARA.

Details

- The parameter setting is not correct.
- G114.1 was commanded while the spindle synchronization was selected with the PLC interface signal.
- G113 was commanded while the spindle-spindle polygon machining option was OFF and the spindle synchronization was selected with the PLC interface signal.
- G110 was commanded while the mixed control (cross axis control) was selected with the PLC interface signal.
- G125 was commanded while the control axis synchronization across part systems was selected with the PLC interface signal.

Remedy

- Check whether the values are set in descending order (from the largest value) for "#1549 Iv0vR1" to "#1553 Iv0vR5".
- Check whether the values are set in descending order for "#1554 Iv0rd2" to "#1557 Iv0rd5".
- Correct the settings of "#1514 expLinax (Exponential function interpolation linear axis)" and "#1515 expRotax (Exponential function interpolation rotary axis)".
- Correct the program.
- Correct the parameter settings.

P990 PREPRO S/W ERR

Details

Combining commands that required pre-reading (nose R offset, corner chamfering/corner rounding, geometric I, geometric IB, and compound type fixed cycle for turning machining) resulted in eight or more pre-read blocks.

Remedy

- Delete some or all of the combinations of commands that require pre-reading.

Appendix 8. User Parameters

The parameters with "(PR)" requires the CNC to be turned OFF after the settings. Turn the power OFF and ON to enable the parameter settings.

【#8001】 WRK COUNT M

Set the M code for counting the number of the workpiece repeated machining.
The number of the M-codes set by this parameter is counted.
The No. will not be counted when set to "0".

---Setting range---
0 to 99

【#8002】 WRK COUNT

Set the initial value of the number of workpiece machining. The number of current workpiece machining is displayed.

---Setting range---
0 to 999999

【#8003】 WRK COUNT LIMIT

Set the maximum number of workpiece machining.
A signal will be output to PLC when the number of machining times is counted to this limit.

---Setting range---
0 to 999999

【#8004】 SPEED

Set the feedrate during automatic tool length measurement.

---Setting range---
1 to 1000000 (mm/min)

【#8005】 ZONE r

Set the distance between the measurement point and deceleration start point.

---Setting range---
0 to 99999.999 (mm)

【#8006】 ZONE d

Set the tolerable range of the measurement point.
An alarm will occur when the sensor signal turns ON before the range, set by this parameter, has not been reached from the measurement point, or when the signal does not turn ON after the range is passed.

---Setting range---
0 to 99999.999 (mm)

【#8007】 OVERRIDE

Set the override value for automatic corner override.

---Setting range---
0 to 100 (%)

【#8008】 MAX ANGLE

Set the maximum corner opening angle where deceleration should start automatically.
When the angle is larger than this value, deceleration will not start.

---Setting range---
0 to 180 (°)

【#8009】 DSC. ZONE

Set the position where deceleration starts at the corner.
Designate at which length point before the corner deceleration should start.

---Setting range---
0 to 99999.999 (mm)

【#8010】 ABS. MAX. (for L system only)

Set the maximum value when inputting the tool wear compensation amount.
A value exceeding this setting value cannot be set.

---Setting range---
0 to 99.999 (mm)

【#8011】 INC. MAX. (for L system only)

Set the maximum value for when inputting the tool wear compensation amount in the incremental mode.
A value exceeding this setting value cannot be set.

---Setting range---
0 to 99.999 (mm)

【#8012】 G73 n (for M system only)

Set the return amount for G73 (step cycle).

---Setting range---
0 to 99999.999 (mm)

【#8013】 G83 n

Set the return amount for G83 (deep hole drilling cycle).

---Setting range---
0 to 99999.999 (mm)

【#8014】 CDZ-VALE (for L system only)

Set the screw cut up amount for G76 and G78 (thread cutting cycle).

---Setting range---
0 to 127 (0.1 lead)

【#8015】 CDZ-ANGLE (for L system only)

Set the screw cut up angle for G76 and G78 (thread cutting cycle).

---Setting range---
0 to 89 (°)

【#8016】 G71 MINIMUM (for L system only)

Set the minimum value of the last cutting amount by the rough cutting cycle (G71, G72).
The cutting amount of the last cutting will be the remainder. When the remainder is smaller than this parameter setting, the last cycle will not be executed.

---Setting range---
0 to 99.999 (mm)

【#8017】 G71 DELTA-D (for L system only)

Set the change amount of the rough cutting cycle.
The rough cutting cycle (G71, G72) cutting amount repeats d+ ⊿d, d, d- ⊿d using the value (d) commanded with D as a reference. Set the change amount ⊿d.

---Setting range---
0 to 99.999 (mm)

【#8018】 G84/G74 n (for M system only)

Not used. Set to "0".

【#8019】 R COMP

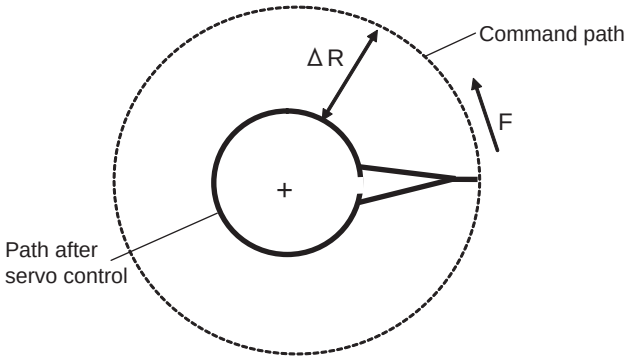
Set a compensation coefficient for reducing a control error in the reduction of a corner roundness and arc radius.
Indicates a maximum control error (mm) in parentheses.
The larger the set value is, the smaller the theoretical error will be. However, since the speed at the corner goes down, the cycle time will be extended.
Coefficient = 100 - set value

(Note) This function will be enabled when "#8021 COMP_CHANGE" is set to "0".

---Setting range---
0 to 99 (%)

Theoretical radius decrease error amount

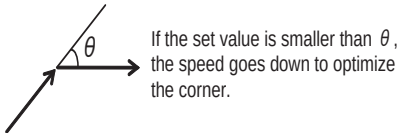
Displays the theoretical radius decrease error amount, ΔR(mm), from the automatic calculation by NC.



Theoretical radius decrease amount in arc

【#8020】 DCC. angle

Set the minimum value of an angle (external angle) that should be assumed to be a corner.
When an inter-block angle (external angle) in high-accuracy mode is larger than the set value, it will be determined as a corner and the speed will go down to sharpen the edge.



(Note) If "0" is set, it will be handled as "5" degrees.
The standard setting value is "0".

---Setting range---
0 to 30 (°)
0: 5 degree (Equals to setting "5")

【#8101】 MACRO SINGLE

Select how to control the blocks where the user macro command continues.
0: Do not stop while macro blocks continue.
1: Stop every block during signal block operation.

【#8102】 COLL. ALM OFF

Select the interference (bite) control to the workpiece from the tool diameter during tool radius compensation and nose R compensation.
0: An alarm will be output and operation stops when an interference is judged.
1: Changes the path to avoid interference.

【#8103】 COLL. CHK OFF

Select the interference (bite) control to the workpiece from the tool diameter during tool radius compensation and nose R compensation.
0: Performs interference check.
1: Does not perform interference check.

【#8105】 EDIT LOCK B

Select the edit lock for program Nos. 8000 to 9999 in the memory.
0: Enable the editing.
1: Prohibit the editing of above programs.

【#8106】 G46 NO REV-ERR (for L system only)

Select the control for the compensation direction reversal in G46 (nose R compensation).
0: An alarm will be output and operation will stop when the compensation direction is reversed (G41 -> G42' G42 -> G41).
1: An alarm won't occur when the compensation direction is reversed, and the current compensation direction will be maintained.

【#8107】 R COMPENSATION

Select whether to move to the inside because of a delay in servo response to a command during arc cutting mode.
0: Move to the inside, making the arc smaller than the command value.
1: Compensate the movement to the inside.

【#8108】 R COMP Select

Select the arc radius error compensation target.
0: Perform compensation over all axes.
1: Perform compensation axis by axis.

(Note) This parameter is effective only when "#8107 R COMPENSATION" is "1".

【#8201】 AX. RELEASE

Select the function to remove the control axis from the control target.
0: Control as normal.
1: Remove from control target.

【#8202】 OT-CHECK OFF

Select whether to enable the stored stroke limit II function set in #8204 and #8205.
0: Enable
1: Disable

【#8203】 OT-CHECK-CANCEL

When the simple absolute position method ("#2049 type" is "9") is selected, the stored stroke limits I, II (or IIB) and IB can be disabled until the first reference position return is executed after the power is turned ON.
0: Enable (according to #8202)
1: Temporarily cancel

(Note) "#8203 OT-CHECK-CANCEL" affects all the stored stroke limits.

【#8204】 OT-CHECK-N

Set the coordinates of the (-) direction in the movable range of the stored stroke limit II or the lower limit coordinates of the prohibited range of stored stroke limit IIB.
If the sign and value are the same as #8205, the stored stroke limit II (or IIB) will be invalid.
If the stored stroke limit IIB function is selected, the prohibited range will be between two points even when #8204 and #8205 are set in reverse. When II is selected, the entire range will be prohibited if #8204 and #8205 are set in reverse.

---Setting range---
-99999.999 to 99999.999 (mm)

【#8205】 OT-CHECK-P

Set the coordinates of the (+) direction in the movable range of the stored stroke limit II or the upper limit coordinates of the prohibited range of stored stroke limit IIB.

---Setting range---
-99999.999 to 99999.999 (mm)

【#8206】 TOOL CHG. P

Set the coordinates of the tool change position for G30. n (tool change position return).
Set with coordinates in the basic machine coordinate system.

---Setting range---
-99999.999 to 99999.999 (mm)

【#8207】 G76/87 IGNR (for M system only)

Select whether to enable the shift operation at G76 (fine boring) and G87 (back boring).
0: Enable
1: Disable

【#8208】 G76/87 (-) (for M system only)

Select the shift direction at G76 and G87.
0: Shift to (+) direction
1: Shift to (-) direction

【#8209】 G60 SHIFT (for M system only)

Set the last positioning direction and distance for a G60 (unidirectional positioning) command.

---Setting range---
-99999.999 to 99999.999 (mm)

【#8210】 OT INSIDE

Select whether the stored stoke limit function set by #8204 and #8205 prevents the machine from moving to the inside or outside of the specified range.
0: Inhibits outside area (Select stored stroke limit II.)
1: Inhibits inside area (Select stored stroke limit II B.)

【#8213(PR)】 Rotation axis type

Select the rotation type (short-cut enabled/disabled) or linear type (workpiece coordinate linear type/all coordinate linear type).
This parameter is enabled only when "#1017 rot" is set to "1".
0: Disable short-cut
1: Enable short-cut
2: Workpiece coordinate linear type

(Note 1) When "2" is set, PLC axes will move as same as when "0" is set.
(Note2) The movement method varies as follows according to the rotary axis type you designate.

<Workpiece coordinate value>
0,1 : Display range 0° to 359.999°
2 : Display range 0° to ±99999.999°
<Machine coordinate value/relative position>
0,1,2 : Display range 0° to 359.999°
<ABS command>
0 : The incremental amount from the end point to the current position is divided by 360, and the axis moves by the remainder amount according to the sign.
1 : Moves with a short-cut to the end point.
2 : In the same manner as the normal linear axis, moves according to the sign by the amount obtained by subtracting the current position from the end point.
<INC command>
0,1,2 : Moves in the direction of the commanded sign by the commanded incremental amount starting at the current position.
<Reference position return>
0,1,2 : The movement to the middle point follows the ABS command or the INC command.
Returns with movement within 360 degrees from the middle point to reference position.

【#8300】 P0 (for L system only)

Set the reference X-coordinates of the chuck and the tail stock barrier.
Set the center coordinate (radius value) of workpiece by the basic machine coordinate system.

---Setting range---
-99999.999 to 99999.999 (mm)

【#8301】 P1 (for L system only)

Set the area of the chuck and tail stock barrier.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
-99999.999 to 99999.999 (mm)

【#8302】 P2 (for L system only)

Set the area of the chuck and tail stock barrier.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
-99999.999 to 99999.999 (mm)

【#8303】 P3 (for L system only)

Set the area of the chuck and tail stock barrier.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
-99999.999 to 99999.999 (mm)

【#8304】 P4 (for L system only)

Set the area of the chuck and tail stock barrier.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
-99999.999 to 99999.999 (mm)

【#8305】 P5 (for L system only)

Set the area of the chuck and tail stock barrier.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
-99999.999 to 99999.999 (mm)

【#8306】 P6 (for L system only)

Set the area of the chuck and tail stock barrier.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
-99999.999 to 99999.999 (mm)

Revision History

Date of revision	Manual No.	Revision details
Dec. 2006	IB(NA)1500267-A	First edition created.
Jan. 2007	IB(NA)1500267-B	Second edition created. • Section 2.6.4 was deleted in Chapter I. • "6.16 Deceleration Check" and "7. Stored Stroke Limit" was moved from Chapter II to the Setup Manual. • "3. CNC Data Input/Output" was added to Chapter III. • Mistakes were corrected.
Jan. 2008	IB(NA)1500267-C	Third edition created. • Chapter I "2. Screen Operation of GOT": The title was changed to "2. CNC Monitor Screen". The explanations on the GOT operation were included in the section 1.2 under "1. Setting and Display Unit". • Chapter I "2. CNC Monitor Screen": The explanations of the screens were revised. • Chapter I "2.4.1 Workpiece Coordinate" was included in "2.3 Tool Offset". • Chapter I: "2.7.6 NC Data Sampling" was added. • Chapter III: "1.4 Consumables" was added. • Chapter III "3. CNC Data Input/Output": The explanations were revised. • Mistakes were corrected.
Aug. 2009	IB(NA)1500267-E	Fourth edition created. Corrections are made corresponding to C70 S/W version B2. · Chapter I "Operation HIstory" was revised. · Chapter IV "User Parameter List" is added (Moved from "I SCREEN OPERATIONS" - Chapter III "3. CNC Data Input/Output" is revised. · Mistakes were corrected.
Jul. 2010	IB(NA)1500267-F	・ Reviewed "Precautions for Safety". ・ Corrected the mistakes.
Feb. 2011	IB(NA)1500267-G	Corrections are made corresponding to C70 S/W version C5. - Chapter I "2.5 Program" was revised. - Chapter I "2.5.2 Edit Type" was added - Chapter I "2.5.4.3 Word Editing" was added - Chapter I "2.7.6 PLC Axis Monitor" was added - Chapter I "2.7.9 NC Data Sampling" was revised. - Chapter III was revised - Mistakes were corrected.

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)

Central Region Service Center
500 CORPORATE WOODS PARKWAY, VERNON HILLS, ILLINOIS 60061, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Michigan Service Satellite
ALLEGAN, MICHIGAN 49010, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-876-6535

Ohio Service Satellite
LIMA, OHIO 45804, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Minnesota Service Satellite
RICHFIELD, MINNESOTA 55423, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

West Region Service Center
5665 PLAZA DRIVE, CYPRESS, CALIFORNIA 90630, U.S.A.
TEL: +1-714-220-4796 / FAX: +1-714-229-3818

East Region Service Center
200 COTTONTAIL LANE SOMERSET, NEW JERSEY 08873, U.S.A.
TEL: +1-732-560-4500 / FAX: +1-732-560-4531

Pennsylvania Service Satellite
ERIE, PENNSYLVANIA 16510, U.S.A.
TEL: +1-814-897-7820 / FAX: +1-814-987-7820

South Region Service Center
2810 PREMIERE PARKWAY SUITE 400, DULUTH, GEORGIA 30097, U.S.A.
TEL: +1-678-258-4500 / FAX: +1-678-258-4519

Texas Service Satellites
GRAPEVINE, TEXAS 76051, U.S.A.
TEL: +1-817-251-7468 / FAX: +1-817-416-5000
FRIENDSWOOD, TEXAS 77546, U.S.A.
TEL: +1-832-573-0787 / FAX: +1-678-573-8290

Florida Service Satellite
SATELLITE BEACH, FLORIDA 32937, U.S.A.
TEL: +1-321-610-4436 / FAX: +1-321-610-4437

Canada Region Service Center
4299 14TH AVENUE MARKHAM, ONTARIO L3R OJ2, CANADA
TEL: +1-905-475-7728 / FAX: +1-905-475-7935

Mexico Region Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-9171-7662 / FAX: +52-55-9171-7649

Monterrey Service Satellite
MONTERREY, N.L., 64720, MEXICO
TEL: +52-81-8365-4171 / FAX: +52-81-8365-4171

Brazil Region Service Center
ACESSO JOSE SARTORELLI, KM 2.1 CEP 18550-000, BOITUVA-SP, BRAZIL
TEL: +55-15-3363-9900 / FAX: +55-15-3363-9911

Brazil Service Satellites
PORTO ALEGRE AND CAXIAS DO SUL BRAZIL
TEL: +55-15-3363-9927
SANTA CATARINA AND PARANA STATES
TEL: +55-15-3363-9927

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V. (EUROPE FA CENTER)
GOTHAER STRASSE 10, 40880 RATINGEN, GERMANY
TEL: +49-2102-486-0 / FAX: +49-2102-486-5910

Germany Service Center
KURZE STRASSE. 40, 70794 FILDERSTADT-BONLANDEN, GERMANY
TEL: + 49-711-3270-010 / FAX: +49-711-3270-0141

France Service Center
25, BOULEVARD DES BOUVETS, 92741 NANTERRE CEDEX FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

France (Lyon) Service Satellite
120, ALLEE JACQUES MONOD 69800 SAINT PRIEST FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

Italy Service Center
VIALE COLLEONI 7-PALAZZO SIRIO CENTRO DIREZIONALE COLLEONI,
20041 AGRATE BRIANZA MILANO ITALY
TEL: +39-039-60531-342 / FAX: +39-039-6053-206

Italy (Padova) Service Satellite
VIA SAVELLI 24 - 35129 PADOVA ITALY
TEL: +39-039-60531-342 / FAX: +39-039-6053-206

U.K. Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-27-6100 / FAX: +44-1707-27-8992

Spain Service Center
CTRA. DE RUBI, 76-80-APDO. 420
08190 SAINT CUGAT DEL VALLES, BARCELONA SPAIN
TEL: +34-935-65-2236 / FAX: +34-935-89-1579

Poland Service Center
UL.KRAKOWSKA 50, 32-083 BALICE, POLAND
TEL: +48-12-630-4700 / FAX: +48-12-630-4727

Poland (Wroclaw) Service Center
UL KOBIERZYCKA 23, 52-315 WROCLAW, POLAND
TEL: +48-71-333-77-53 / FAX: +48-71-333-77-53

Turkey Service Center
BAYRAKTAR BULVARI, NUTUK SOKAK NO.5, YUKARI DUDULLU
ISTANBUL, TURKEY
TEL: +90-216-526-3990 / FAX: +90-216-526-3995

Czech Republic Service Center
TECHNOLOGICKA 374/6,708 00 OSTRAVA-PUSTKOVEC, CZECH REPUBLIC
TEL: +420-59-5691-185 / FAX: +420-59-5691-199

Russia Service Center
213, B.NOVODMITROVSKAYA STR., 14/2, 127015 MOSCOW, RUSSIA
TEL: +7-495-748-0191 / FAX: +7-495-748-0192

Sweden Service Center
STRANDKULLEN, 718 91 FROVI, SWEDEN
TEL: +46-581-700-20 / FAX: +46-581-700-75

Bulgaria Service Center
4 A. LYAPCHEV BOUL., 1756 - SOFIA, BULGARIA
TEL: +359-2-8176000 / FAX: +359-2-9744061

Ukraine (Kharkov) Service Center
APTEKARSKIY PEREULOK 9-A, OFFICE 3, 61001 KHARKOV, UKRAINE
TEL: +38-57-732-7744 / FAX: +38-57-731-8721

Ukraine (Kiev) Service Center
4-B, M. RASKOVOYI STR., 02660 KIEV, UKRAINE
TEL: +38-044-494-3355 / FAX: +38-044-494-3366

Belarus Service Center
703, OKTYABRSKAYA STR., 16/5, 220030 MINSK, BELARUS
TEL: +375-17-210-4626 / FAX: +375-17-227-5830

South Africa Service Center
P.O. BOX 9234, EDLEEN, KEMPTON PARK GAUTENG, 1625 SOUTH AFRICA
TEL: +27-11-394-8512 / FAX: +27-11-394-8513

ASEAN

MITSUBISHI ELECTRIC ASIA PTE. LTD. (ASEAN FA CENTER)

Singapore Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 / FAX: +65-6476-7439

Indonesia Service Center
THE PLAZZA OFFICE TOWER, 28TH FLOOR JL.M.H. THAMRIN KAV.28-30, JAKARTA, INDONESIA
TEL: +62-21-2992-2333 / FAX: +62-21-2992-2555

Malaysia (KL) Service Center
60, JALAN USJ 10 /1B 47620 UEP SUBANG JAYA SELANGOR DARUL EHSAN, MALAYSIA
TEL: +60-3-5631-7605 / FAX: +60-3-5631-7636

Malaysia (Johor Baru) Service Center
NO. 16, JALAN SHAH BANDAR 1, TAMAN UNGKU TUN AMINAH, 81300 SKUDAI, JOHOR MALAYSIA
TEL: +60-7-557-8218 / FAX: +60-7-557-3404

Vietnam Service Center-1
ROOM 1004, 1005, FLOOR 10, 255 TRAN HUNG DAO CO GIANG WARD, DIST. 1, HCMC, VIETNAM
TEL: +84-8-3838-6931 / FAX: +84-8-3838-6932

Vietnam Service Center-2
LOT G10 - AREA 4 - HIEP BINH CHANH WARD - THU DUC DISTRICT - HCMC, VIETNAM
TEL: +84-8-2240-3587 / FAX: +84-8-3726-7968

Vietnam (Hanoi) Service Center
5FL, 59 - XA DAN STR., DONG DA DIST., HN, VIETNAM
TEL: +84-4-3573-7646 / FAX: +84-4-3573-7650

Philippines Service Center
UNIT NO.411, ALABAMG CORPORATE CENTER KM 25. WEST SERVICE ROAD
SOUTH SUPERHIGHWAY, ALABAMG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416 / FAX: +63-2-807-2417

MITSUBISHI ELECTRIC AUTOMATION (THAILAND) CO., LTD. (THAILAND FA CENTER)

BANG-CHAN INDUSTRIAL ESTATE NO.111 SOI SERITHAI 54
T.KANNAYAO, A.KANNAYAO, BANGKOK 10230, THAILAND
TEL: +66-2906-8255 / FAX: +66-2906-3239

Thailand Service Center
898/19,20,21,22 S.V. CITY BUILDING OFFICE TOWER 1, FLOOR 7
RAMA III RD., BANGPONGPANG, YANNAWA, BANGKOK 10120, THAILAND
TEL: +66-2-682-6522 / FAX: +66-2-682-9750

INDIA

MITSUBISHI ELECTRIC ASIA PVT. LTD.

FIRST & SECOND FLOOR, AVR BASE, MUNICIPAL NO.BC-308,
HENNURE BANASWADI ROAD, HRBR RING ROAD, BANGALORE-560 043, INDIA
TEL: +91-80-4020-1600 / FAX: +91-80-4020-1699

India (Pune) Service Center
EL-3, J BLOCK, M.I.D.C., BHOSARI PUNE 411026, INDIA
TEL: +91-20-2710-2000 / FAX: +91-20-2710-2185

India (Bangalore) Service Center
S 615, 6TH FLOOR, MANIPAL CENTER, BANGALORE 560001, INDIA
TEL: +91-80-509-2119 / FAX: +91-80-532-0480

India (Gurgaon) Service Center
UNIT NUMBER 107, 1ST FLOOR, SEVA CORPORATE PARK, MG ROAD, NEAR IFFCO CHOWK,
GURGAON, HARYANA INDIA
TEL: +91-124-4982901 / FAX: +91-124-4982900

OCEANIA

MITSUBISHI ELECTRIC AUSTRALIA LTD.

Oceania Service Center
348 VICTORIA ROAD, RYDALMERE, N.S.W. 2116 AUSTRALIA
TEL: +61-2-9684-7269 / FAX: +61-2-9684-7245

CHINA

MITSUBISHI ELECTRIC AUTOMATION (CHINA) LTD. (CHINA FA CENTER)

China (Shanghai) Service Center
4/F., ZHI FU PLAZA, NO. 80 XIN CHANG ROAD, HUANG PU DISTRICT,
SHANGHAI 200003, CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2308-2830
China (Ningbo) Service Dealer
China (Wuxi) Service Dealer
China (Jinan) Service Dealer
China (Wuhan) Service Satellite

China (Beijing) Service Center
9/F, OFFICE TOWER 1, HENDERSON CENTER, 18 JIANGUOMENNEI DAJIE,
DONGCHENG DISTRICT, BEIJING 100005, CHINA
TEL: +86-10-6518-8830 / FAX: +86-10-6518-3907
China (Beijing) Service Dealer

China (Tianjin) Service Center
B-2 801/802, YOUYI BUILDING, NO.50 YOUYI ROAD, HEXI DISTRICT,
TIANJIN 300061, CHINA
TEL: +86-22-2813-1015 / FAX: +86-22-2813-1017
China (Shenyang) Service Satellite
China (Changchun) Service Satellite

China (Chengdu) Service Center
ROOM 407-408, OFFICE TOWER AT SHANGRI-LA CENTER, NO. 9 BINJIANG DONG ROAD,
JINJIANG DISTRICT, CHENGDU, SICHUAN 610021, CHINA
TEL: +86-28-8446-8030 / FAX: +86-28-8446-8630

China (Shenzhen) Service Center
ROOM 2512-2516, 25/F., GREAT CHINA INTERNATIONAL EXCHANGE SQUARE, JINTIAN RD.S.,
FUTIAN DISTRICT, SHENZHEN 518034, CHINA
TEL: +86-755-2399-8272 / FAX: +86-755-8218-4776
China (Xiamen) Service Dealer
China (Dongguan) Service Dealer

KOREA

MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD. (KOREA FA CENTER)

Korea Service Center
1480-6, GAYANG-DONG, GANGSEO-GU SEOUL 157-200, KOREA
TEL: +82-2-3660-9602 / FAX: +82-2-3664-8668

Korea Taegu Service Satellite
603 CRYSTAL BUILDING 1666, SANBYEOK-DONG, BUK-KU, DAEGU, 702-010, KOREA
TEL: +82-53-604-6047 / FAX: +82-53-604-6049

TAIWAN

MITSUBISHI ELECTRIC TAIWAN CO., LTD. (TAIWAN FA CENTER)

Taiwan (Taichung) Service Center
NO.8-1, GONG YEH 16TH RD., TAICHUNG INDUSTRIAL PARK TAICHUNG CITY, TAIWAN R.O.C.
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan (Taipei) Service Center
3RD. FLOOR, NO.122 WUKUNG 2ND RD., WU-KU HSIANG, TAIPEI HSIEN, TAIWAN R.O.C.
TEL: +886-2-2299-2205 / FAX: +886-2-2298-1909

Taiwan (Tainan) Service Center
2F(C),1-1, CHUNGHWA-RD., YONGKANG CITY, TAINAN HSIEN, TAIWAN R.O.C.
TEL: +886-6-313-9600 / FAX: +886-6-313-7713

Notice

Every effort has been made to keep up with software and hardware revisions in the contents described in this manual. However, please understand that in some unavoidable cases simultaneous revision is not possible.
Please contact your Mitsubishi Electric dealer with any questions or comments regarding the use of this product.

Duplication Prohibited

MITSUBISHI CNC

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : TOKYO BUILDING,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	C70
MODEL CODE	100—017
Manual No.	IB-1500267

IB(NA)1500267(ENG)-G(1102)MEE
Made in JAPAN