Changes for the Better

MITSUBISHI ELECTRIC

CNC
MELDAS C6/C64/C64T

INSTRUCTION MANUAL

BNP-B2259C(ENG)

Introduction

This manual is referred to when using the MELDAS C64 Series.
This manual explains how to operate the screens of the C64 Series. 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.

⚠ DANGER

Not applicable in this manual.

⚠ WARNING

Do not put any startup switch for C6/C64 on GOT's touch key. If a communication fault (including cable disconnection) occurs between GOT and CNC C6/C64, 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.

⚠ 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.

⚠ CAUTION

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.

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.

⚠ CAUTION

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.

9. Items related to servo/spindle parameters

- To change the control mode to the High-gain amp (MDS-B-V14/V24) mode after replacement of the Standard amp (MDS-B-V1/V2), it is need to change the parameters and to adjust the servo parameters to fit to the High-gain amp.
- Mode change between the Standard amp mode and the High-gain amp mode is actually performed when the power (200V) is turned ON. Thus, when changing some parameters unique to each amp, an alarm “7F” occurs and requests to turn the power ON again.
  Note that the alarm “7F” may occur when the amp is mounted on the machine for the first time.
  When the alarm “7F” occurs, turn the power ON again.
  The alarm “7F” may not occur at second turning ON or later unless the above-mentioned parameters are changed.
- With MDS-C1 series, only the serial encoder is applied as the motor end detector. Thus, OHE/OHA type detector cannot be used as the motor end detector.
- Do not make remarkable adjustments or changes of the parameters as the operation may became unstable.

CONTENTS

I. SCREEN OPERATION

II. MACHINE OPERATIONS

III. MAINTENANCE

IV. APPENDICES

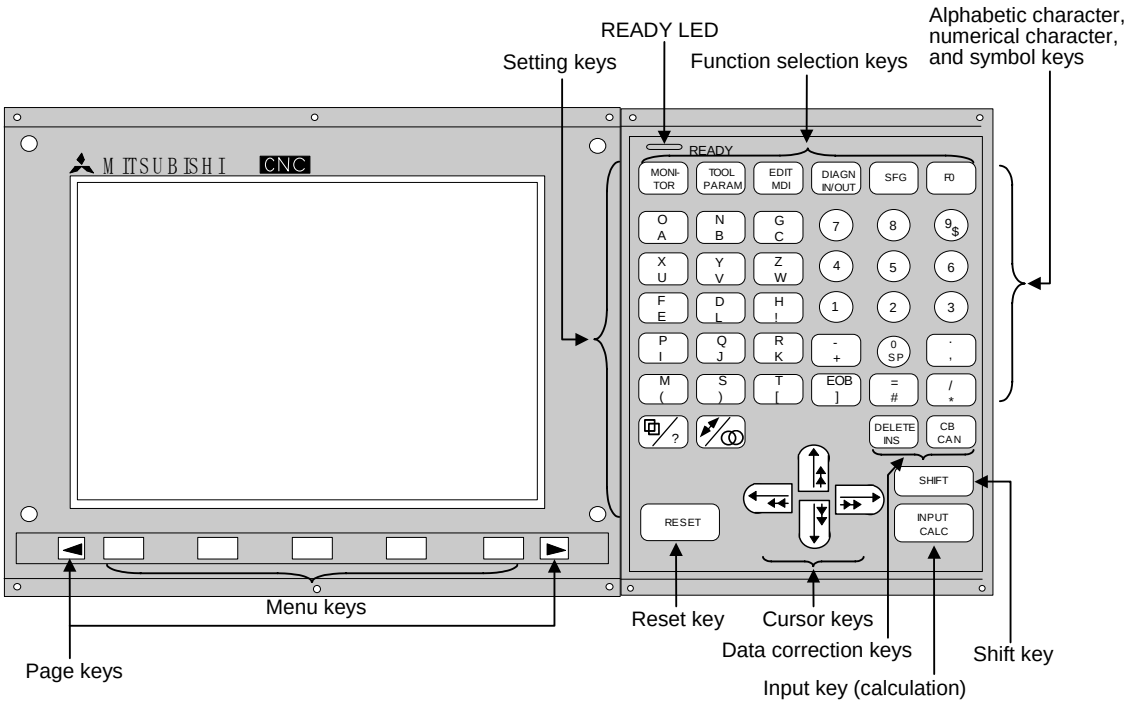
CHAPTER 1　SCREEN OPERATION

1. Setting and Display Unit

1.1 NC-dedicated Display Unit

NC-dedicated display unit consists of a display, keys, and menu keys, as illustrated below.

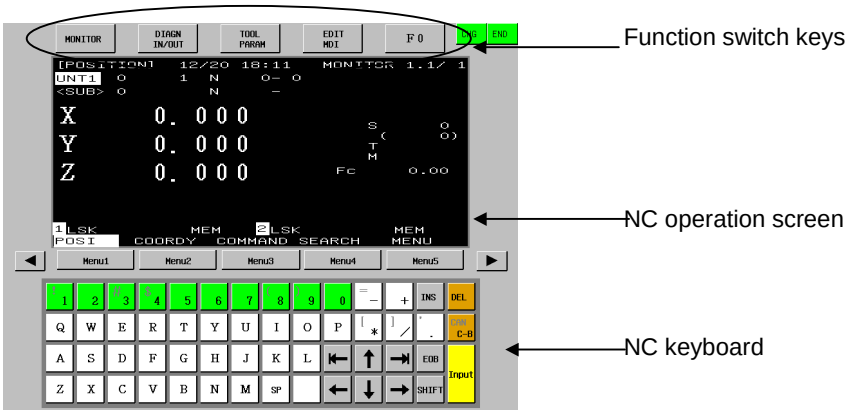
(1) Appearance of the NC-dedicated display unit (sample)



(Note) Refer to the section “2.1 Setting and Display Unit Operation” for details of NC-dedicated display unit and onwards.

1.2 Graphic Operation Terminal (A985GOT)

An NC operation screen and an NC keyboard shown below are displayed on A985GOT.

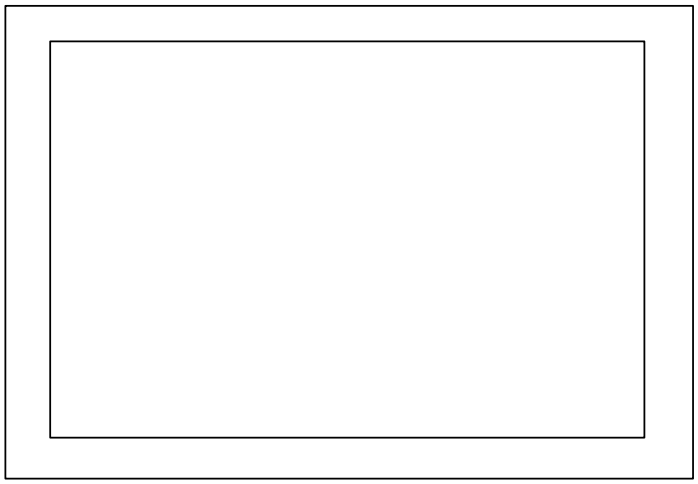


This screen is dedicated for NC operation and its size is fixed.
Refer to “2. Screen Operation of A985GOT” for details.

1.3 Panel Computer

Panel computer includes the personal computer with the touch-panel display as shown below. Refer to “3. Screen Operation When Using a Panel Computer” for details.

(1) Appearance of the panel computer (sample)

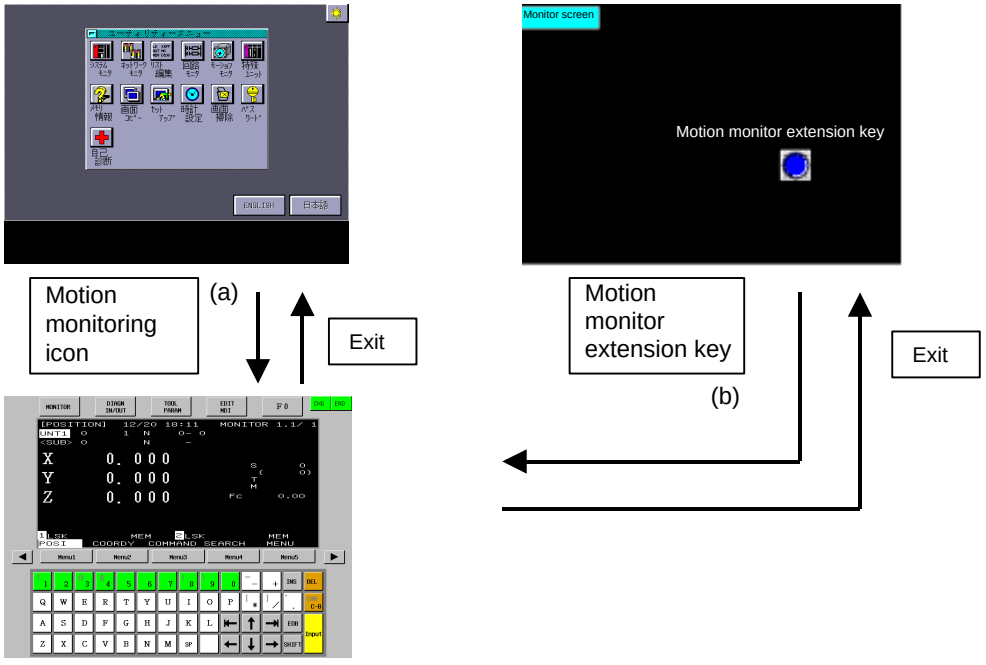


2. Screen Operation of A985GOT

Using A985GOT enables the NC screen as follows.

C6/C64(NC)	GOT	Size / Layout	Details	Usage
CNC system software Escape sequence code Key code	 NC screen NC keyboard	Fixed	Method 1: CNC system software controls GOT screen directly. • Display speed is not affected by the cycle time of CNC built-in PLC. • Display size is fixed on GOT.	NC screen display on GOT
Wide-use registers	O:123 N:456 X100.00 Y150.00 NC counter,modal, execution program, etc.	Variable	Method 2: GOT displays the data output periodically from CNC system software to general purpose registers. • Display speed is not affected by the cycle time of CNC built-in PLC. • The type and quantity of the data output periodically from CNC are limited.	Access from GOT to NC memory
APLC Ladder Wide-use registers	Lamp, calc-result, etc. Switch, button	Variable	Method 3: CNC built-in PLC (ladder) or APLC, that is C-laguage program released to CNC users, prepares data in general-purpose registers. GOT reads or writes the data. • There is not much restriction, however, display speed will be affected by the cycle time of CNC built-in PLC or APLC.	APLC release

This manual describes the screen indicated by method 1 (area)



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

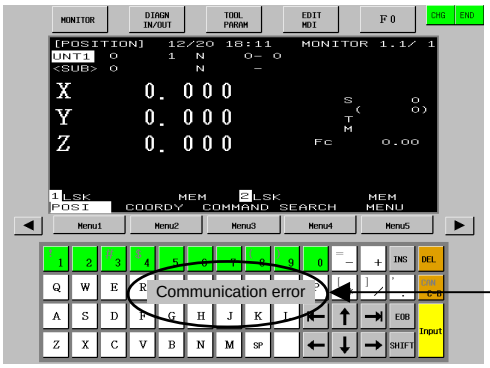
NC screen appears by either method.
(a) Select “Motion monitor” key on Utility screen.
(b) Select “Motion monitor extension” key on Monitor screen.

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.)

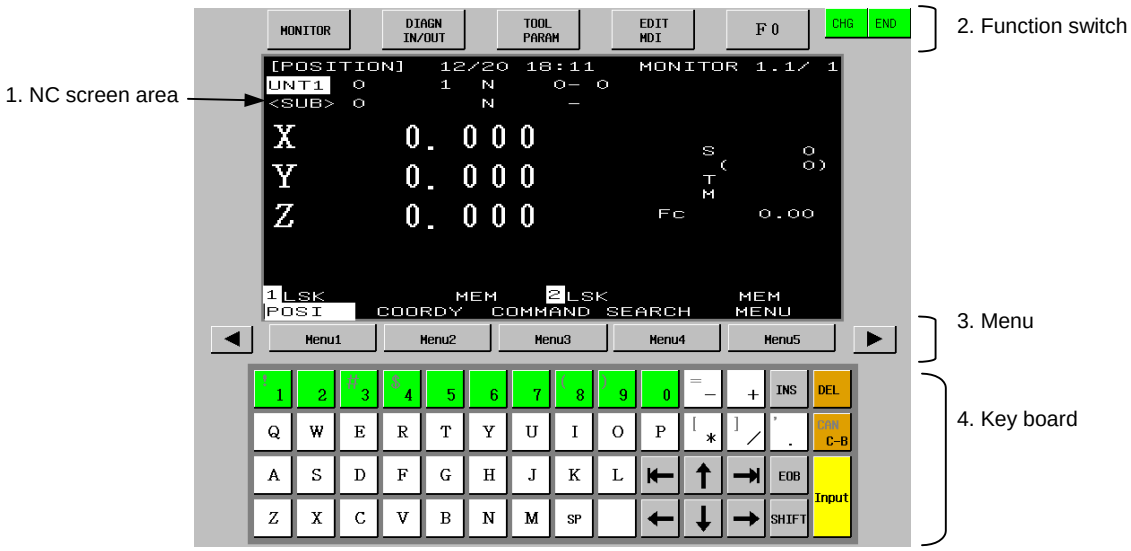
Priority	Error message	Remedy
High	E71 communication driver is not installed. (NC monitor)	Install E71 communication driver.
Middle	Monitor data not found. (NC monitor)	Download the NC screen data (special module screen).
Low	The IP address of NC to monitor is not set up. (NC monitor)	Set the IP address 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.



The display contents and composition of the screen by A985GOT are described hereafter.

In this screen, the various information which are needed to setup and maintain the machine and NC system are displayed and set.
When the NC-dedicated display unit is used, refer to the section “2.1 Setting and Display Unit Operation” and onwards.



(1) Display items

Display items	Details
1. NC screen area	The functions such as position monitoring, alarm diagnosis, tool offset/parameters, and program edit. They have equivalent function as the screens of NC-dedicated display unit. Note that graphic trace function and PLC ladder edit function are not provided.
2. Function switch	The following keys used to select the display function. These keys are equivalent to Function Select Key of NC-dedicated display unit.

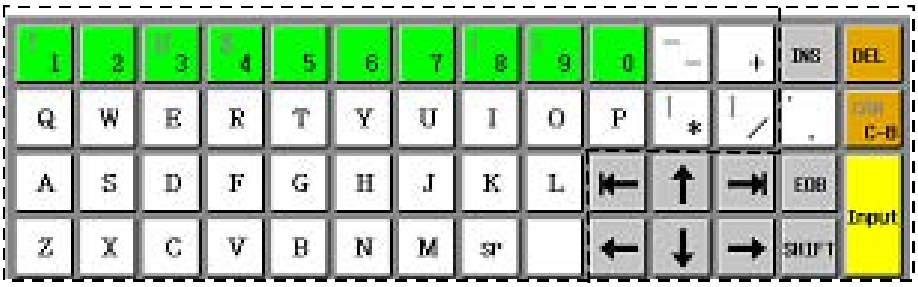
Key	Description	During communication error
CHG	For switching the NCs being monitored when more than one NC is set to be monitored.	Valid
END	For finishing the NC monitoring function.	
MONITOR	Selects position monitoring function. Refer to “2.2 Monitor”.	Invalid
DIAGN IN/OUT	Selects alarm diagnosis function 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 Tool Offset” and “2.4 Parameters”.	
EDIT MDI	Selects program edit function. Refer to “2.5 Program”.	
F0	Selects APLC release screen.	

Display items		Details
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.

(3) Precautions at connecting with both the NC-dedicated display unit and the A985GOT

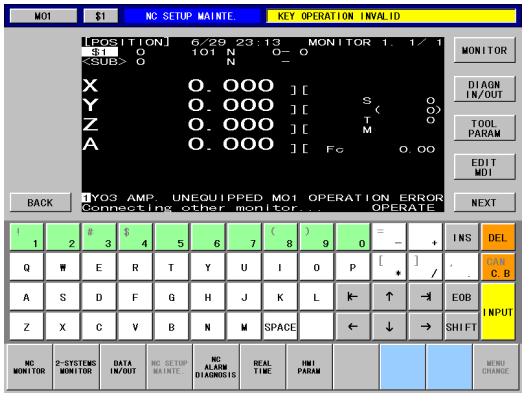
A message appears on the GOT's title bar and menu area 2 when an NC- dedicated display unit is connected. The menu keys and the keys on the panel computer's keyboard such as data setting keys and special keys became invalid. The keys pressed on an NC-dedicated display unit are valid. To validate the key operations on the A985GOT, press the OPERATE menu.

Screen select keys on the bottom of the screen such as NC MONITOR and 2-SYSTEMS MONITOR are available.

1)

Connect with NC-dedicated display unit.

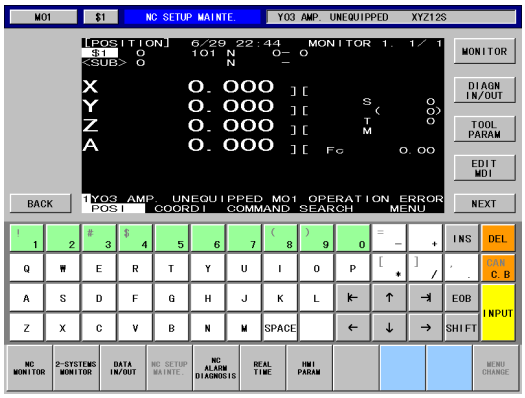
A message will appear on the keyboard and menu.
Key input other than OPERATE menu key are invalid.
Part systems cannot be changed while the key operation is invalid.



2)

Press the OPERATE menu.

Key operations on this display unit will be enabled.



If the OPERATE menu is pressed on another display unit, that display unit will be validated.

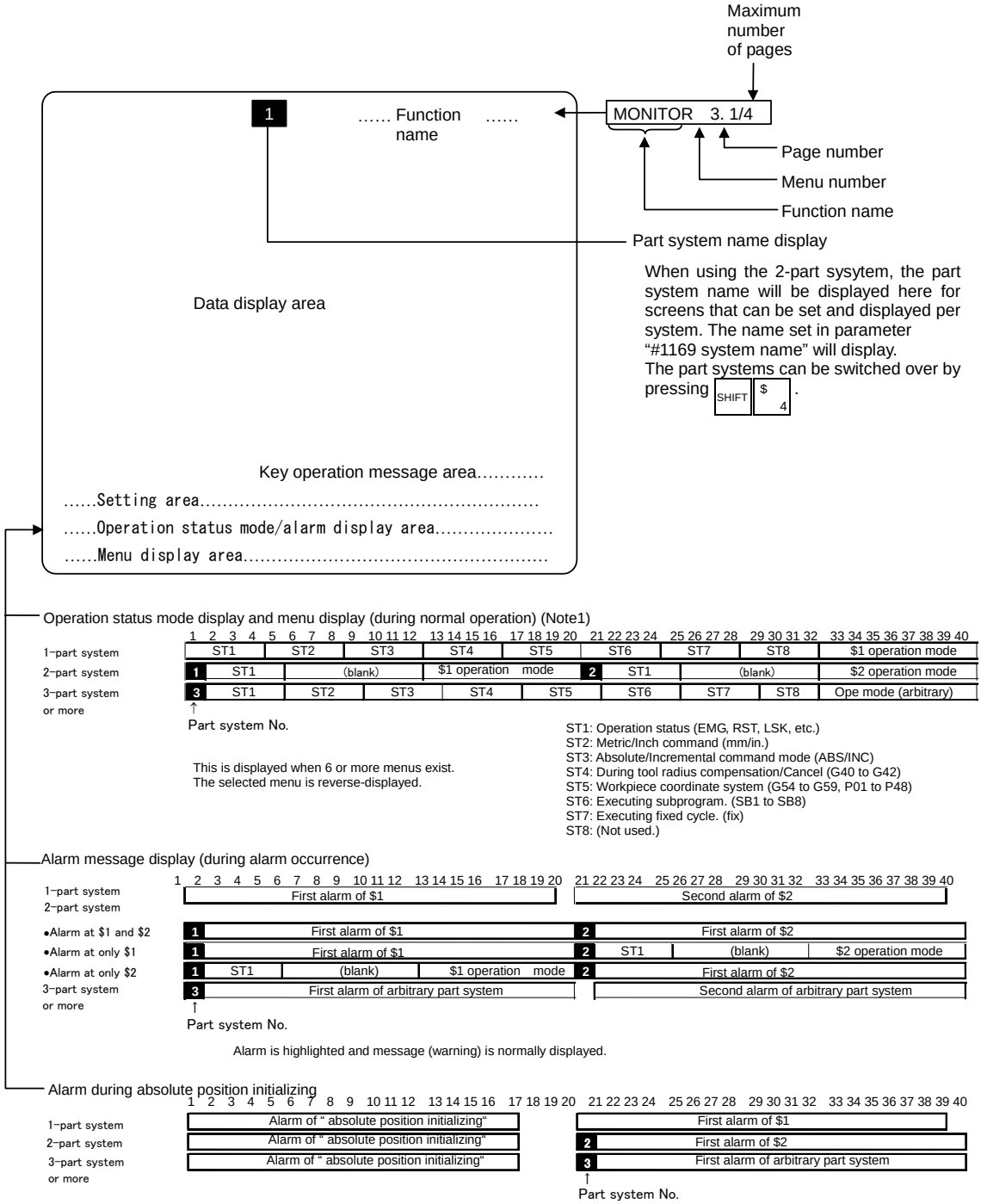
2.1 Setting and Display Unit Operation

The keys for the NC screen of A985GOT are mainly used in the explanation hereafter.
When the NC-dedicated display unit is used, refer to the following table together.

	NC-dedicated Display Unit	A985GOT	Panel Computer
Function select keys	MONITOR	MONITOR	MONITOR
	DIAGN INOUT	DIAGN IN/OUT	DIAGN IN/OUT
	TOOL PARAM	TOOL PARAM	TOOL PARAM
	EDIT MDI	EDIT MDI	EDIT MDI
	F0	F0	None
	SFG	None	None
Page select keys	BACK	BACK	BACK
	NEXT	NEXT	NEXT
Cursor keys		↑ ↓ → ←	↑ ↓ → ←
Tab keys	Shift + Cursor, etc.	→| |←	→| |←
Other control keys	DELETE INS	DEL	DEL
	SHIFT + DELETE INS	INS	INS
	SHIFT	SHIFT	SHIFT
	C.B CAN	CAN C.B	CAN C.B
	SHIFT + C.B CAN SHIFT after C.B CAN	SHIFT + CAN C.B	SHIFT + CAN C.B
	INPUT INPUT CALC	INPUT	INPUT
Alphabet keys	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		
Numerical keys	! 1 2 # 3 $ 4 5 6 7 (8) 9 0		
Symbol keys	= - + * / , . [ ] EOB		
	SHIFT + 0 SP SP	SP	SPACE
	$	SHIFT + $ 4	SHIFT + $ 4

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



(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.

(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 G59	Selection of the workpiece coordinate is indicated.
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.

(1) Alarms/Messages

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.

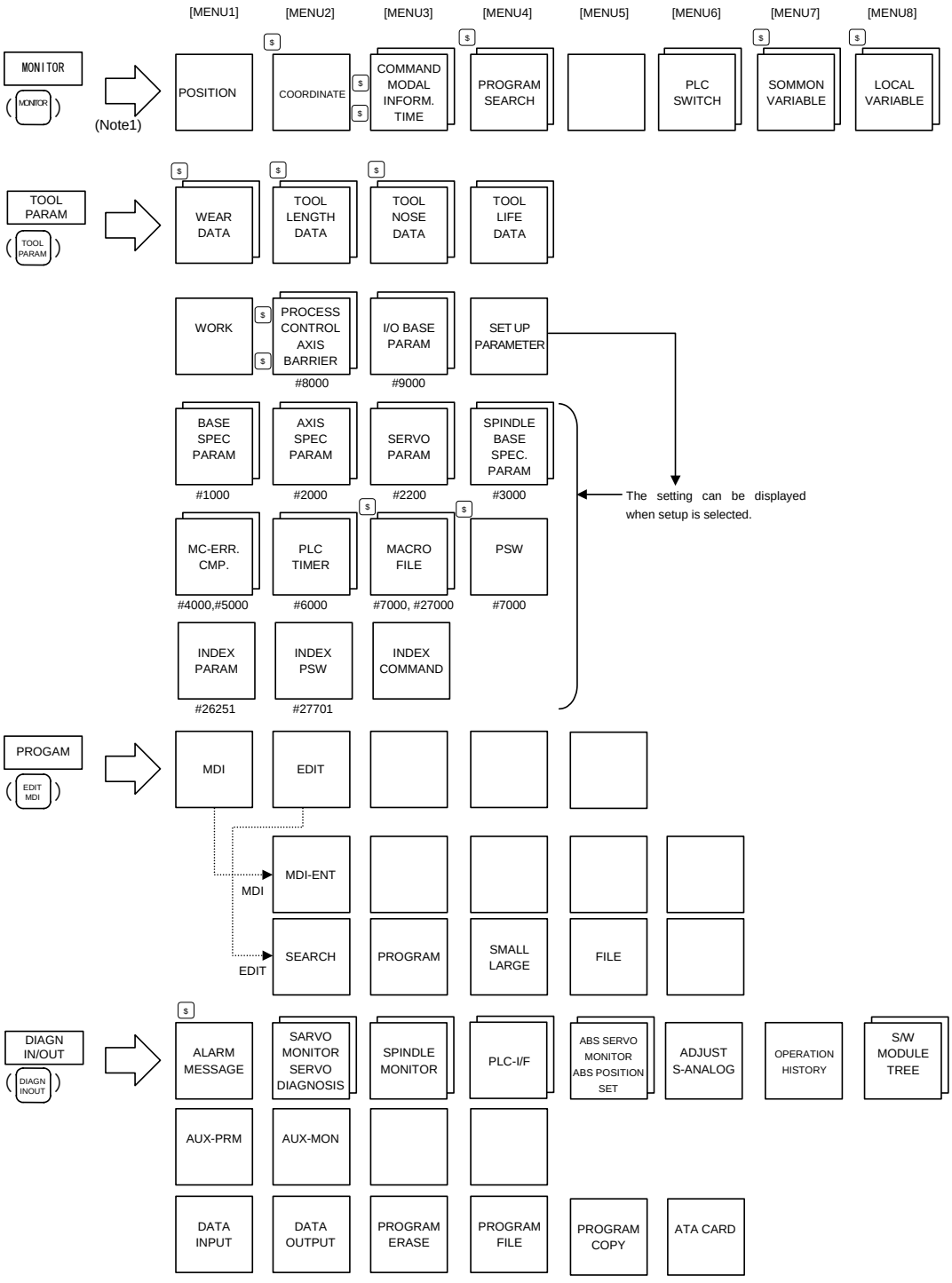
Message	Details of message	Remedy
Communication error	An error has occurred in the communication with the NC.	• Check the connection between the NC and GOT. (Cable connection, noise, etc.) • Check the NC and GOT network settings.
KEY OPERATION INVALID	The NC-dedicated display unit or display unit other 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.

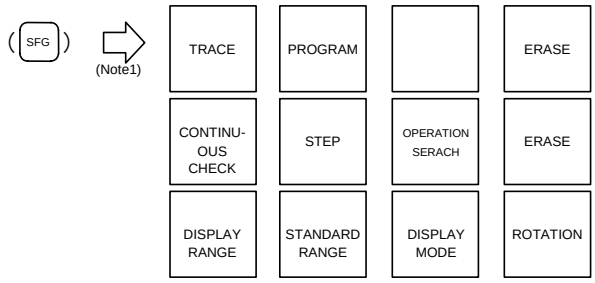
Refer to “Chapter IV Appendices” for details of the other alarms or messages.

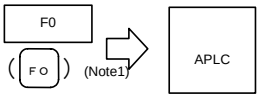
2.1.2 Screen Transition Diagram (L system)

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.



(Note1) The keys in the parenthesis are the function keys used by NC-dedicated display unit.

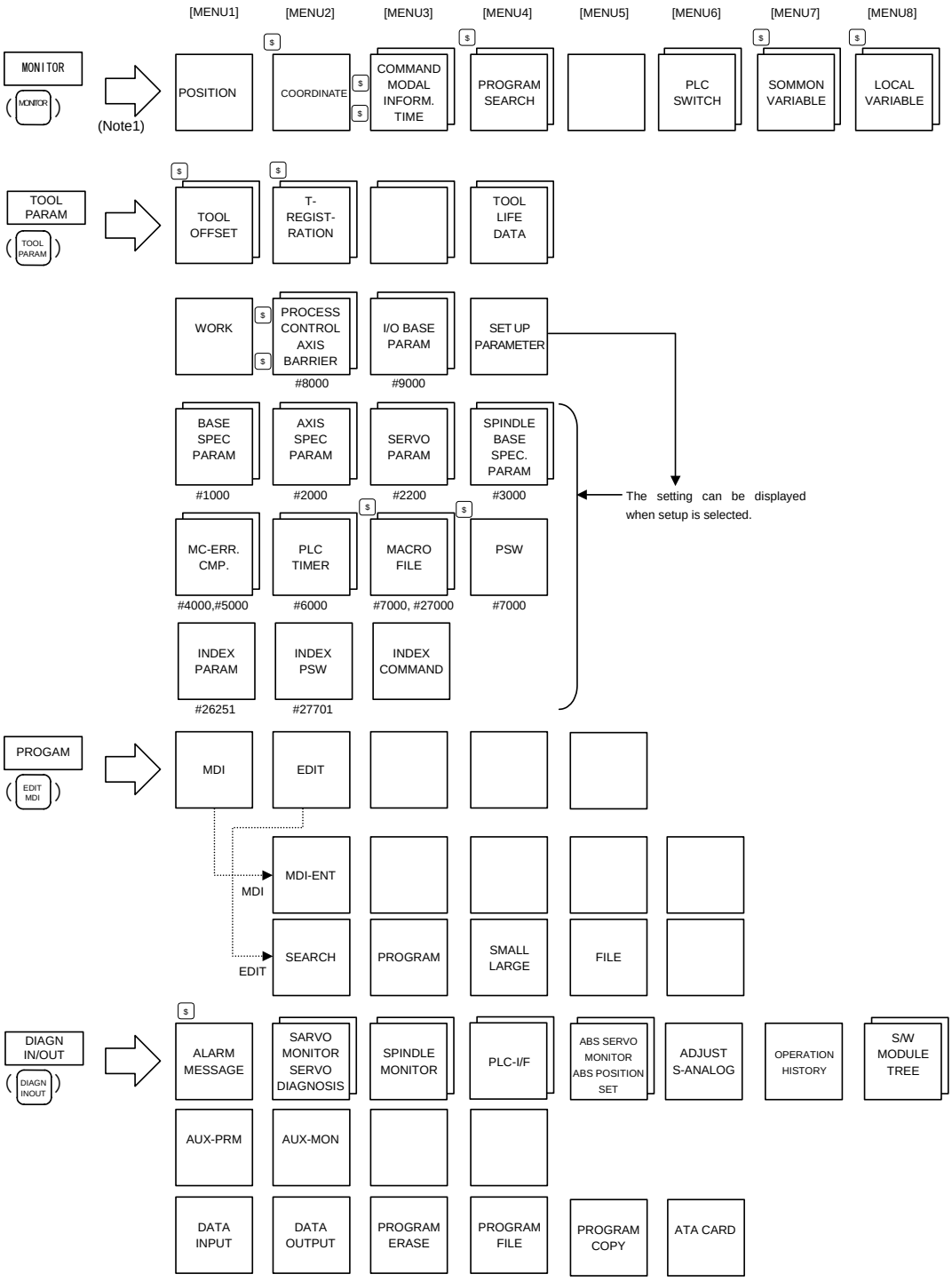




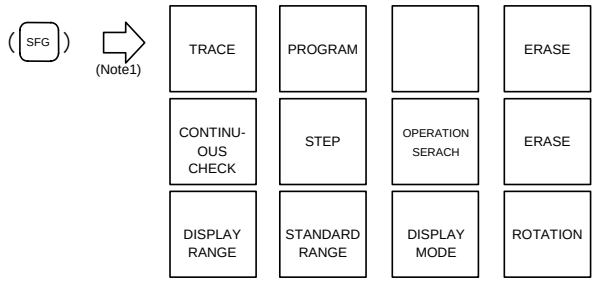
(Note1) SFG key and F0 are available only when the NC-dedicated display unit (CT100).

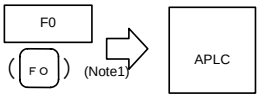
2.1.3 Screen Transition Diagram (M system)

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.



(Note1) The keys in the parenthesis are the function keys used by NC-dedicated display unit.

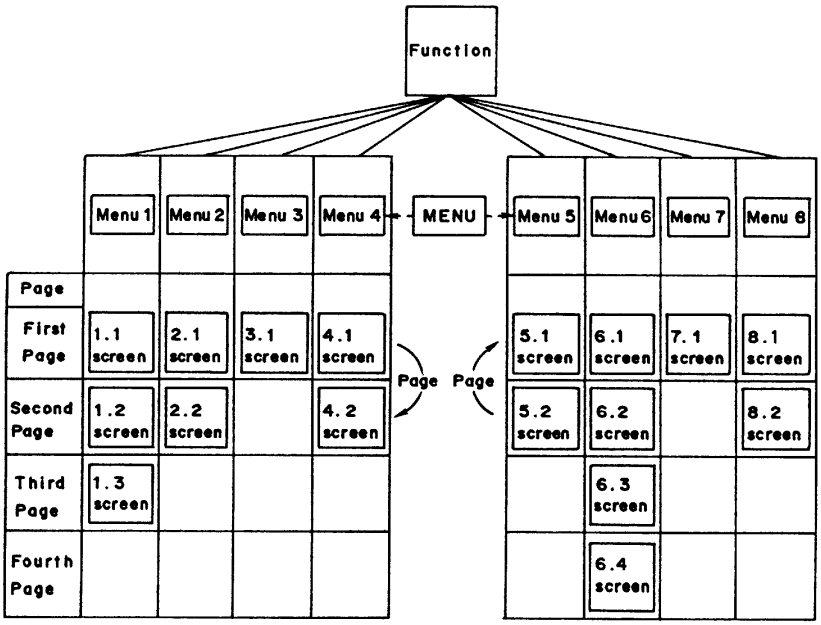




(Note1) SFG key and F0 are available only when the NC-dedicated display unit (CT100).

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.

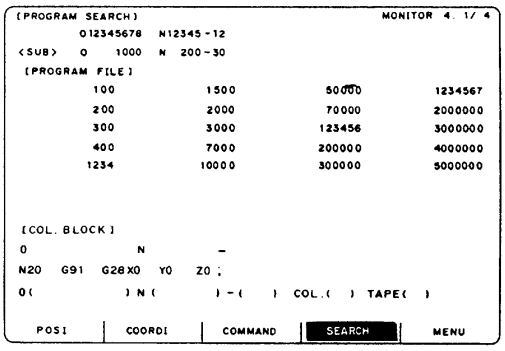


(1) Select a function screen

MONITOR | DIAGN IN/OUT | TOOL PARAM | EDIT MDI

Press the function selection key corresponding to the function screen to be displayed.

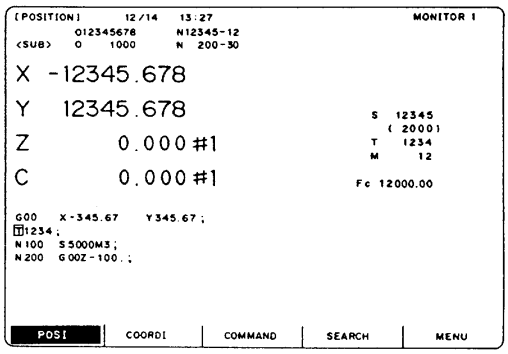
(Example) Press the MONITOR key.



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.

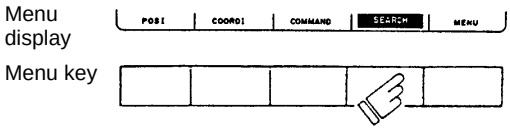
(Example) Press the MONITOR key again.

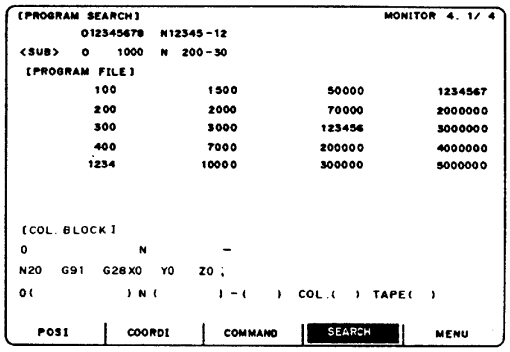


(2) Select a menu screen in the function

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.

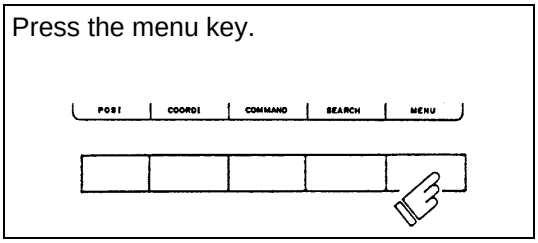


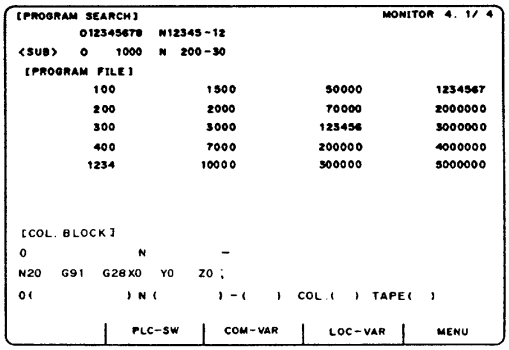


1) The selected menu screen is displayed in the data display area.
2) The selected menu is highlighted in the menu display area.

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.

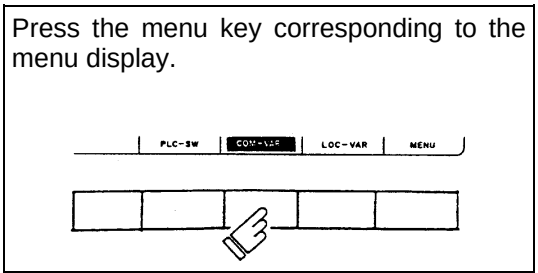
(1) Press the menu key.

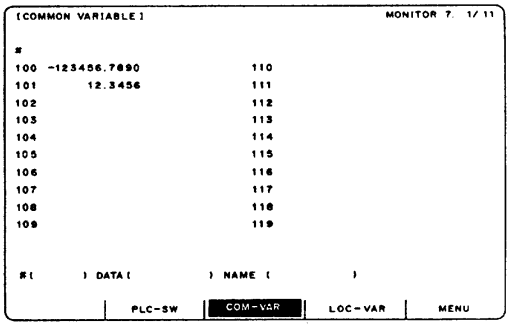




1) Only the menu display area is changed and the remaining menu group is displayed.

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



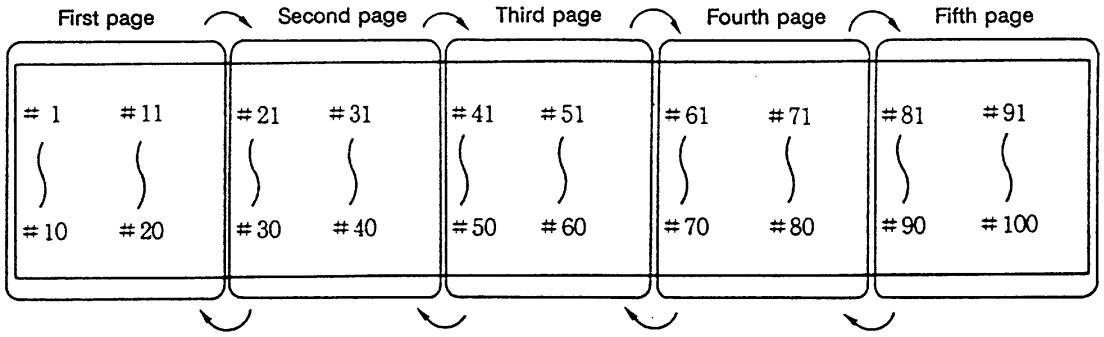


(3) Select a page in the menu screen

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.

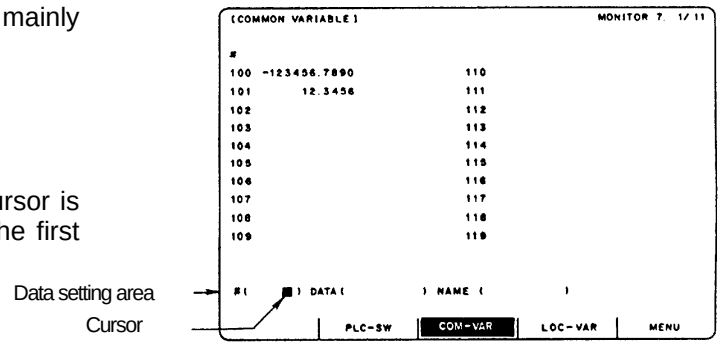
2.1.5 Data Setting Method

(1) Outline of data setting

The data setting method consists mainly of the following steps:

(1) Enter the data number.
(2) Move the cursor.
(3) Press data keys.
(4) Press the INPUT key.

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.

To move the cursor to the next parentheses, press the [→] key.

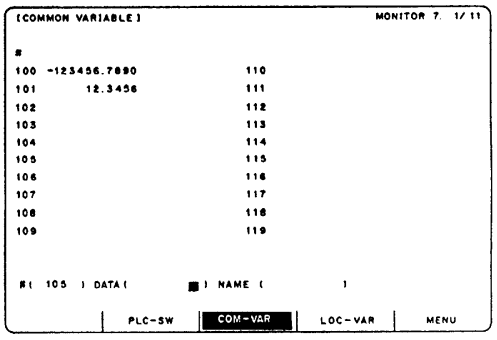
(3) Press data keys.

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.



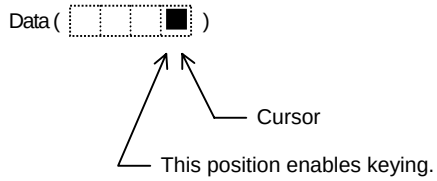
1) Data setting processing is performed according to the setting area contents, and the result is displayed in the data display area.
2) The data number in the setting area is incremented by one, and the cursor is displayed in the right end within the second parentheses.
   After the last data number is input, it is not displayed. At this time, the cursor is displayed in the right end of the first parentheses.

3) To consecutively set data, repeat (3) and (4).
4) To change the data number, press the [INPUT] key. The number is incremented by one. When the [↓] key is pressed, the number is incremented by one. When the [↑] key is pressed, the number is decremented by one. The data number can also be directly changed by moving the cursor to the data number setting area.

(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

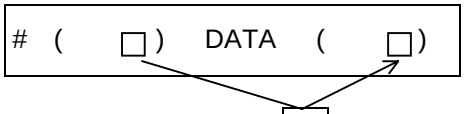
1) Data write into the display screen (by keying) is made at the position indicated by the cursor. When the cursor is not displayed, keying is not effective.



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.



When the [→] key is pressed, the cursor is moved to the right end within the next parentheses.

When the [|←] keys are pressed, the cursor is moved to the preceding parentheses.

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 [CAN C.B.] key.

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

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

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

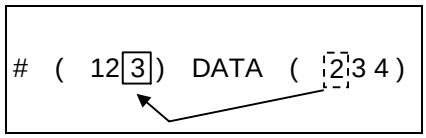
If you press [0] [0] [0] ,	☞	# (10) DATA (13.00[0])

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.



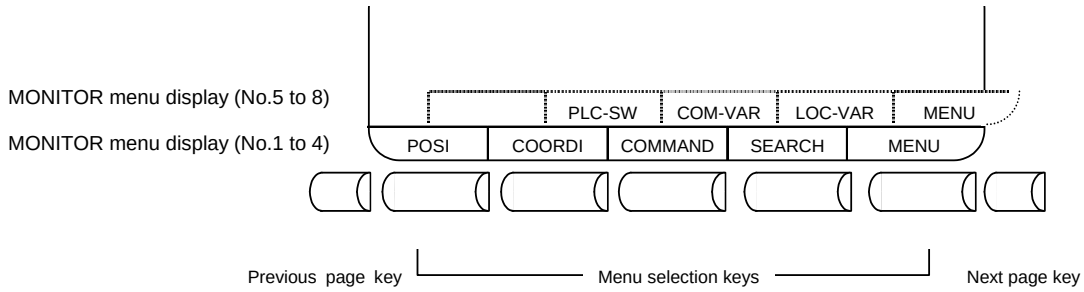
If you press the [|←] key, the cursor is moved to the right end within the preceding parentheses.

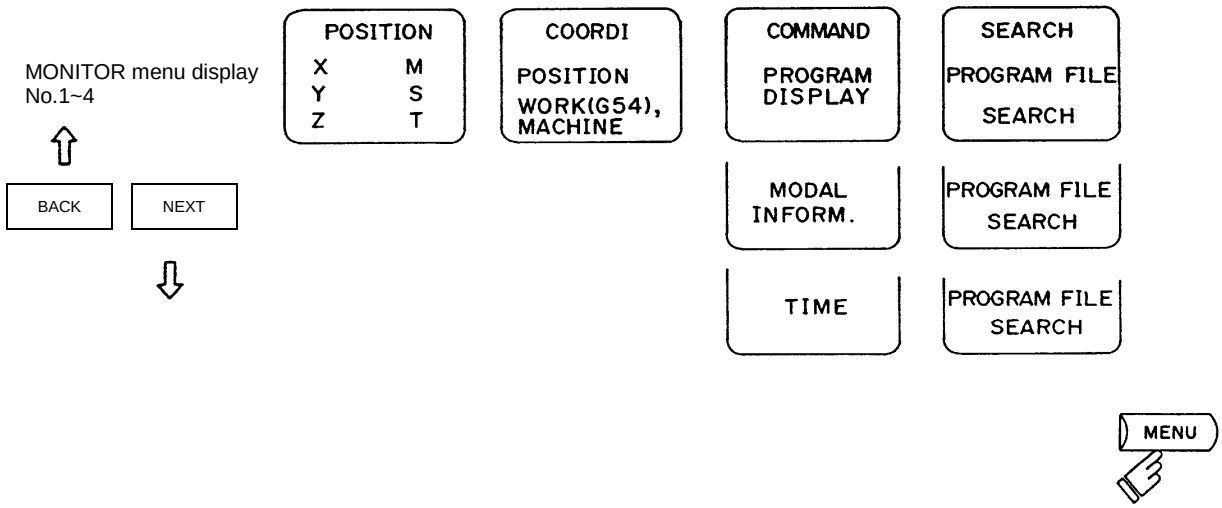
(3) Miscellaneous information

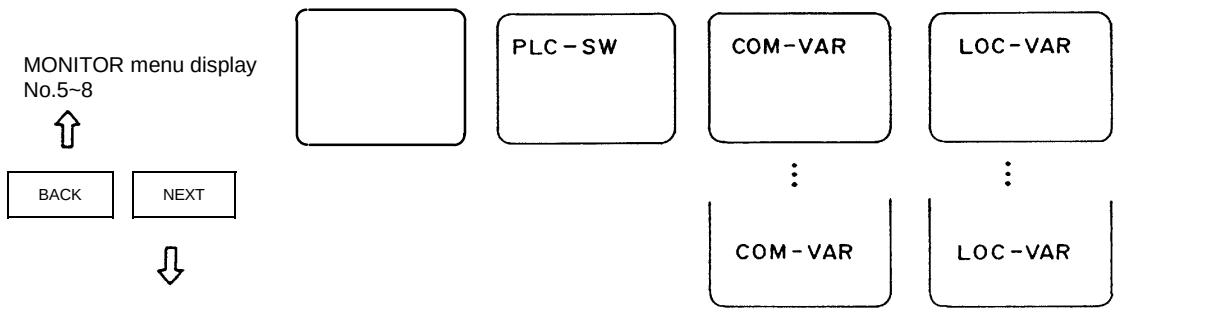
1) Data can also be set by other special methods. See the appropriate items. (For example, manual numeric command setting is performed by the reverse display setting method.)
2) If an invalid key is pressed when data is set within parentheses, a "setting error" will occur at input time and the data will 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:

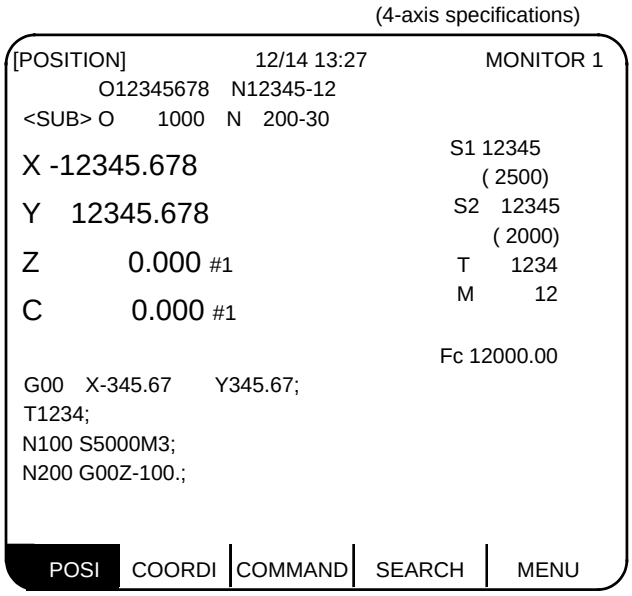






2.2.1 Position

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



The following can be performed on the POSITION screen:
(1) Full screen erase
(2) Origin set. The current value (POSITION) data of each axis can be set to 0.
(3) Manual numeric command. Miscellaneous function output of M, S, T, etc., can be set through the screen.

Display item	Explanation
12/14 13:27	The date and the time are displayed.
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.
[POSITION] X-12345.678 Y 12345.678 Z 0.000 #1 C 0.000 #1	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 (2500) S2 12345 (2000)	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	The tool command value is displayed. The last four digits of the miscellaneous function command value are displayed.
Fc 12000.00	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.

Display item	Explanation
G00 X-345.67 Y345.67; T1234; N100 S5000M3; N200 G00Z-100.;	Four blocks of the current program being executed are displayed. The top block is currently executed. The subsequent three lines are the subsequent block program.

2.2.1.1 Total Clear of CRT Screen

If you do not use the unit for extended periods, clear the entire CRT screen to prevent deterioration of the CRT by the following procedures.

(1) Select the menu [POSI] on the MONITOR screen and press the [SHIFT] [CAN C.B.] keys to clear whole screen.

(2) If you want to display screen after clearing of whole screen, press a function select key such as [MONITOR] to display the screen you desire.

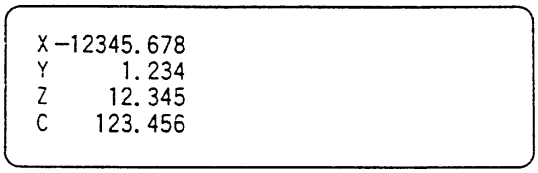
2.2.1.2 Position Display Counter Zero and Origin Zero

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 ;.

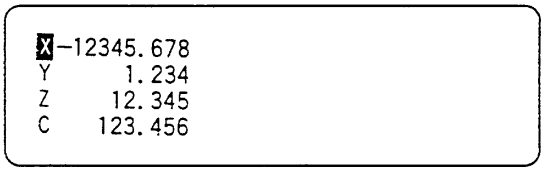
(Note) Origin zero is valid only when "#1123 origin" is set to 0.

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



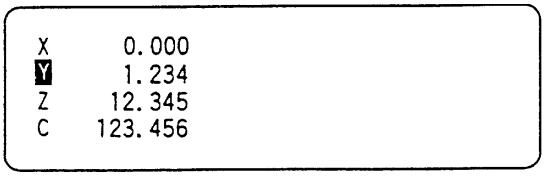
Press the address key [X] .

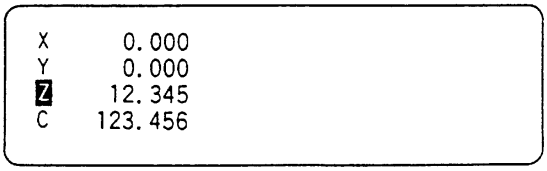
1) The address indication corresponding to the key is highlighted.



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





2.2.1.3 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.

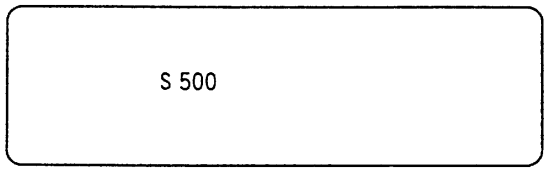
(2) Operating procedures of manual numerical value commands

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.

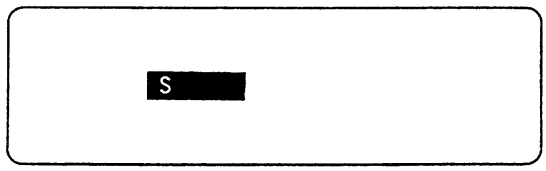
(Note 1) Press [S] [2] for the 2nd spindle. Manual numerical value command cannot be performed for the 2nd miscellaneous functions.

(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.



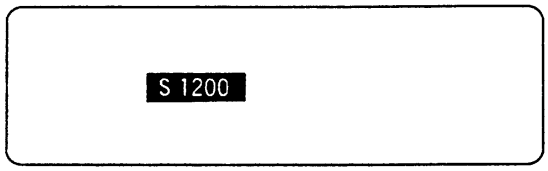
Press the address key [S].



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

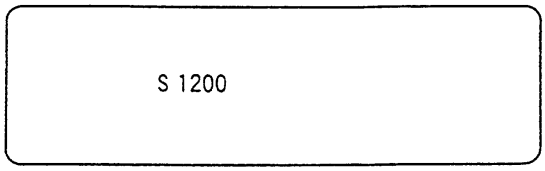
Set the numerical value by number keys.

[!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.

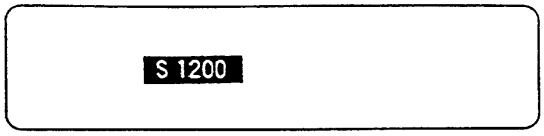
(3) Action to be taken when an erroneous numeric is set and the correct one is desired to be set

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.

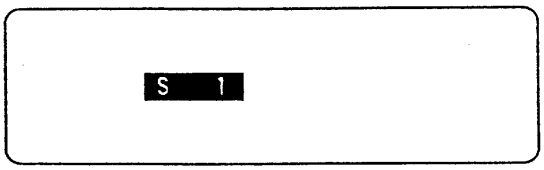
Method (2) Retry the entry, beginning with pressing the address key corresponding to the command.

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

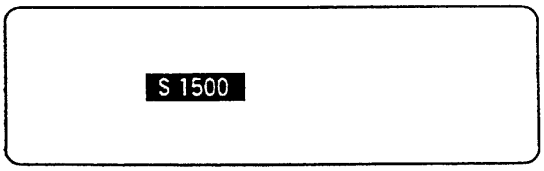


Method (1)

Press DEL DEL DEL to delete the erroneously set numeric. ☞

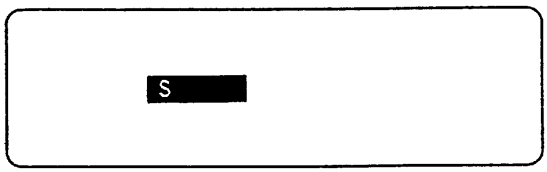


Enter 5 0 0 . ☞

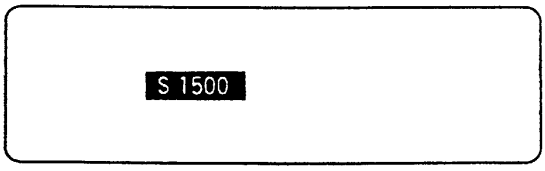


Method (2)

Press address key S to return to the initial status. ☞



Enter ! 1 5 0 0 . ☞



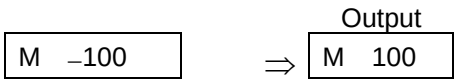
(4) Setting/output range of manual numeric command

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

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

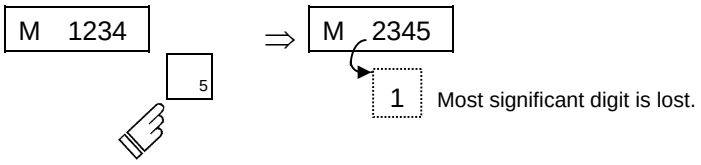
(Note 1) If the type is BCD output and a negative number is set, the positive value converted from it will be output.

(Example) Manual numeric command



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

(Example)

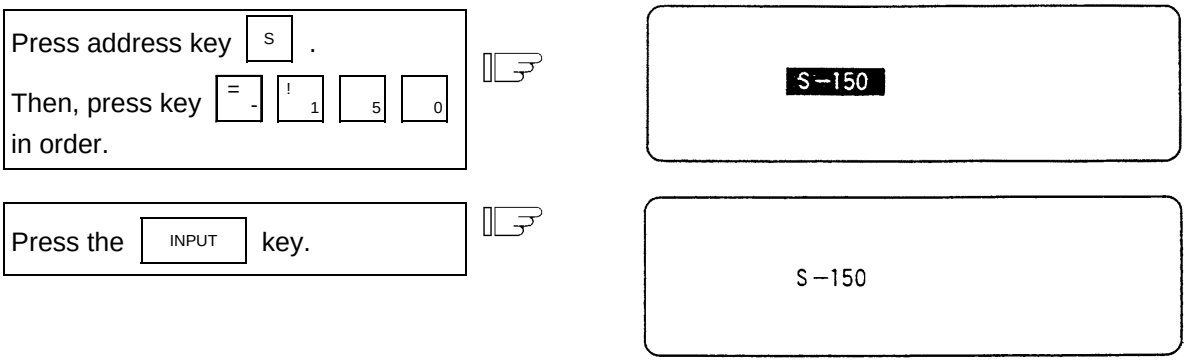


(5) Other notes on operation

(1) When a minus command is set:

Before setting the numeric, press the [= -] (minus) key.

(Example) If S-150 is specified:



1) A minus value will be output, but a positive value will display.

(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] [CAN C.B.] 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 M99 is 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.

2.2.1.4 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

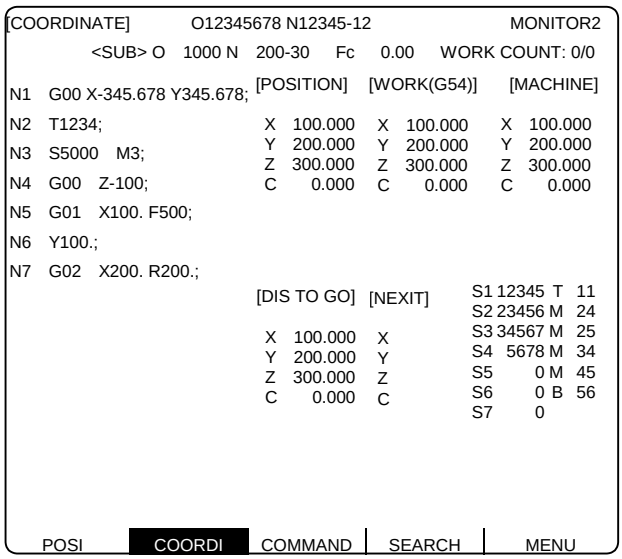
1) A command block containing an EOB only or a command program consisting of only a comment statement is not regarded as an execution block but is processed together with the following command program.

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 key 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> O 1000 N 200-30	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.
N1 G00 X-345.678 Y345.678; N2 T1234; N3 S5000 M3; N4 G00 Z-100; N5 G01 X100.F500; N6 Y100.; N7 G02 X200.R200.;	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 -345.678 Y 345.678 Z 0.000 #1 C 0.000	G54 to G59 workpiece coordinate system modal numbers and the coordinates in the workpiece coordinate system are displayed.

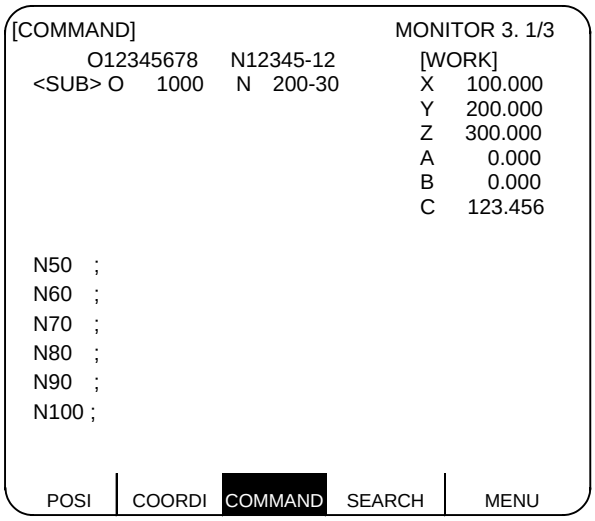
Display item	Explanation
[MACHINE] X -345.678 Y 345.678 Z 0.000 C 0.000	The coordinates of each axis in the basic machine coordinate system in which the unique position determined depending on the machine is used as the zero point are displayed.
[WORK (G54)] X -345.678 Y 345.678 Z 0.000 C 0.000	G54 to G59 workpiece coordinate system modal numbers and the coordinates in the workpiece coordinate system are displayed.
[DIS TO GO] X 0.000 Y 0.000 Z 0.000 C 0.000	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 C	The coordinates of the end point in the block executed next are displayed.
S1 12345 : : S7 0 T 11 M 24 : B 56	The program command S modal value (maximum 7 sets) currently being executed are displayed. The program command T modal value currently being executed is displayed. The program command M modal values (maximum 4 sets) currently being executed are displayed. The program command 2nd miscellaneous function modal value currently being executed is displayed.

2.2.3 Command

When the menu key 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

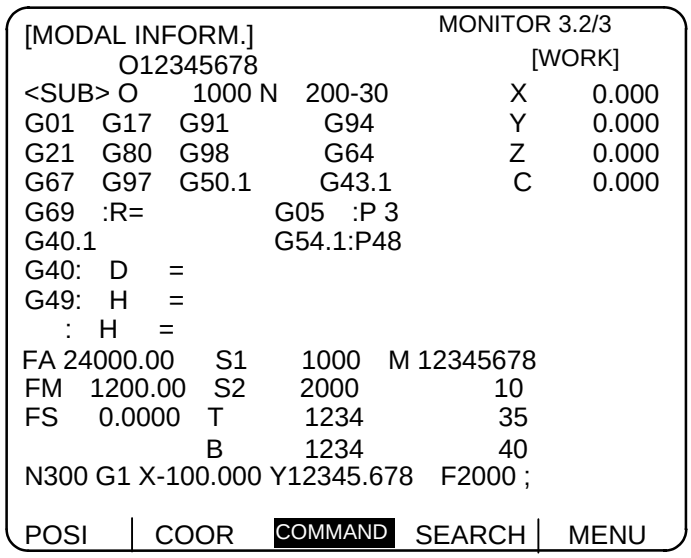
This screen displays the active machining program's execution blocks for monitoring.



Display item	Explanation
O12345678 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.
N50; ⋮ N90; N100;	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 100.000 Y 200.000 Z 300.000 A 0.000 B 0.000 C 123.456	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.

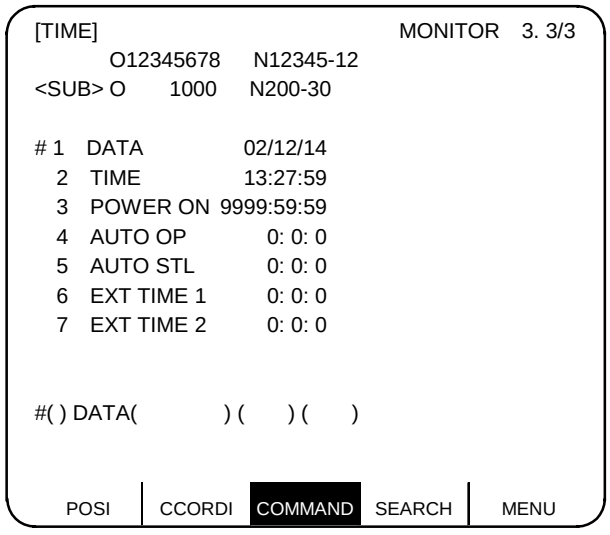


Display item	Explanation
O12345678 N12345-12	The currently executing program number, sequence number, and block number are displayed.
<SUB> O1000 N200-30	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.
[WORK] X 100.000 Y 200.000 Z 300.000 C 0.000	The coordinates in the workpiece coordinate system being currently executed are displayed. (Same as POSITION screen)
[MODAL INFORM.] G01.....G94 ((G66.1.....G50.1	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 =	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.

Display item	Explanation
FA 24000.00	The modal value of the current program command F being executed is displayed. (Unit: mm/min)
FM 1200.00	The manual feed rate is displayed. (Unit: mm/min)
FS 0.0000	The manual feed rate is displayed. (Unit: mm/rev)
S1: 1000 S2: 2000	The modal value of the current program command S being executed is displayed. (Up to two spindles)
T 1234	The modal value of the current program command T being executed is displayed.
M 12345678 10 35 40	A maximum of four modal values of the current program command M being executed are displayed.
B 1234	The second miscellaneous function modal value of the current program command being executed is displayed.
N300 G1X-100.000~	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.



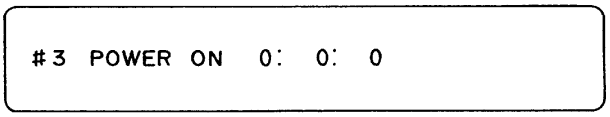
Display item	Explanation
O12345678 N12345-12	The currently executing program number, sequence number, and block number are displayed.
<SUB> O1000 N200-30	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.
#1 DATE 02/12/14 2 TIME 13:27:59 3 POWER ON 9999:59:59 4 AUTO OP 0: 0: 0 5 AUTO STL 0: 0: 0 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.



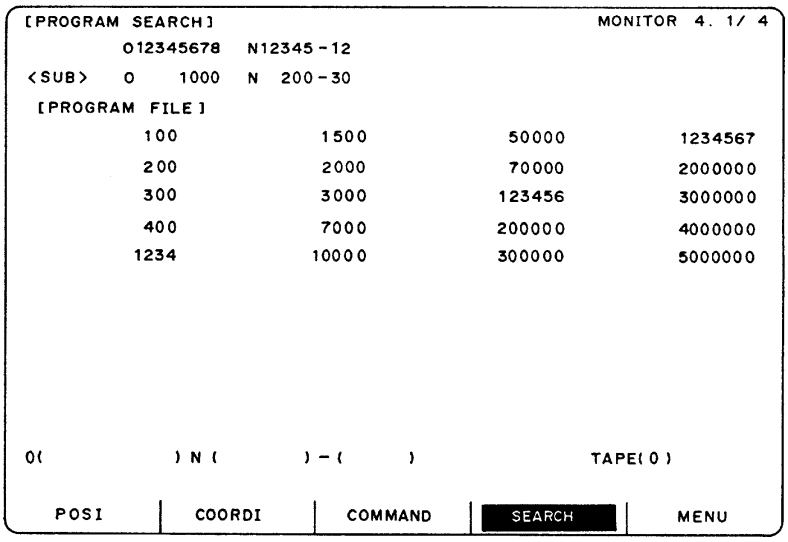
DATE : Set date (set "YEAR" in the Christian Era).
TIME : Set time in 24-hour mode.
POWER ON : Total integrating time of the time from control unit power ON to OFF.
AUTO OP : Total integrating time of the machining time from AUTO STL button pressed in the memory mode to M02/M30 or reset button pressed.
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 key [SEARCH] 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.



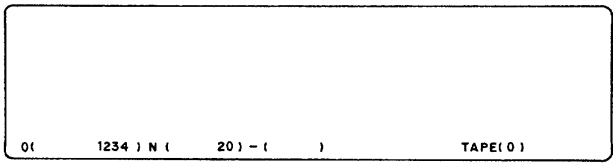
Display item	Explanation
O12345678 N12345-12	The currently executing program number, sequence number, and block number are displayed.
<SUB> O 1000 N 200-30	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.
[PROGRAM FILE] 100 1500 50000 1234567 200 2000 70000 2000000 300 3000 123456 3000000 400 7000 200000 4000000 1234 10000 300000 5000000	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.

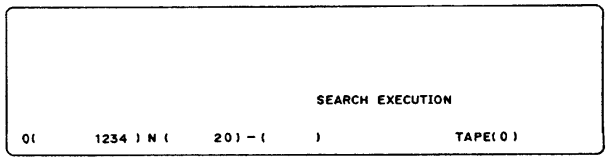
2.2.4.1 Memory Search

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.

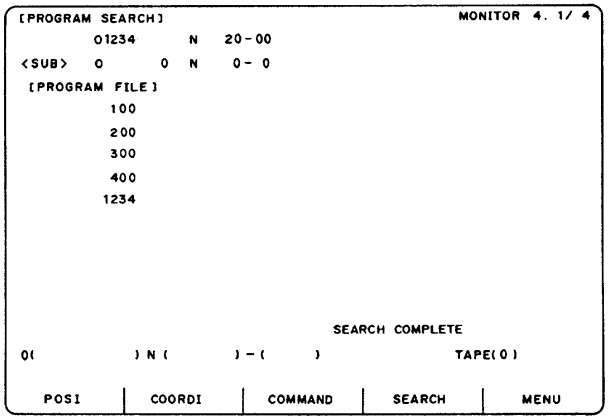
Set the program number to be called. Set the sequence number and block number as required.
(Example)
To call O1234 N20 block,
O (1 2 3 4) N (2 0) - ( )
COL. ( ) TAPE (0)



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.

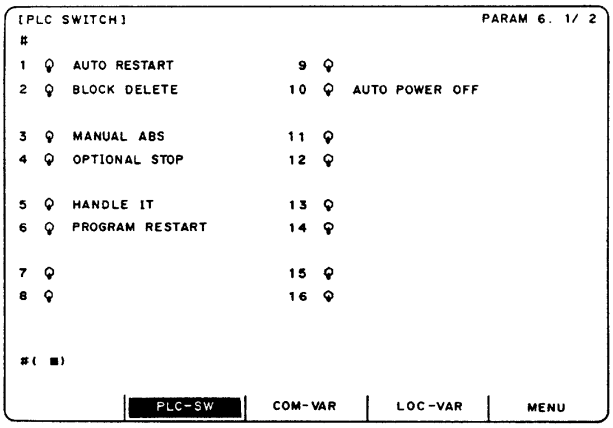
(Note 2) One block which the control unit executes in one automatic start cycle can be searched in this case. The block with ; (EOB) or sequence number only is not regarded as a one cycle execution block. Axis movement command or control command such as M, S, or T is contained in it.

2.2.5 PLC Switch

When the menu key [PLC-SW] is pressed, the PLC SWITCH screen is displayed.

The control signals for operation are assigned by using user PLC. The PLC-SWITCH screen enables you to set each control signal to on or off. (A maximum of 32 signals)

This screen is created with the user PLCs, so each screen will differ. Refer to the instruction manual issued by the machine maker.



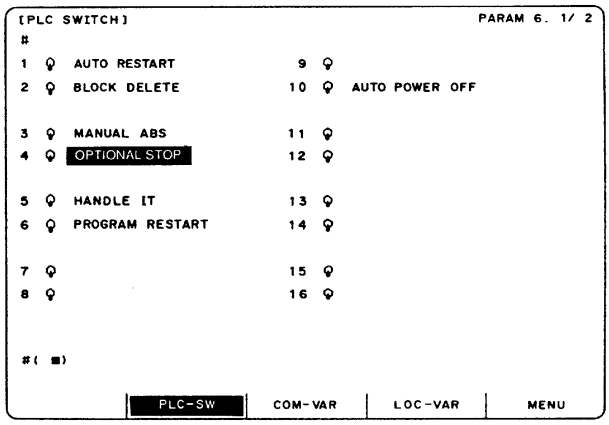
2.2.5.1 PLC Switch ON and OFF Operation

Set the number of the switch to be set to on in # ( ) and press the [INPUT] key. The mark of the switch is set to the up position.
In this state, the switch function becomes effective and is controlled.

To set OPTIONAL STOP to ON,

Set 4 in # ( ).

Press the [INPUT] key.



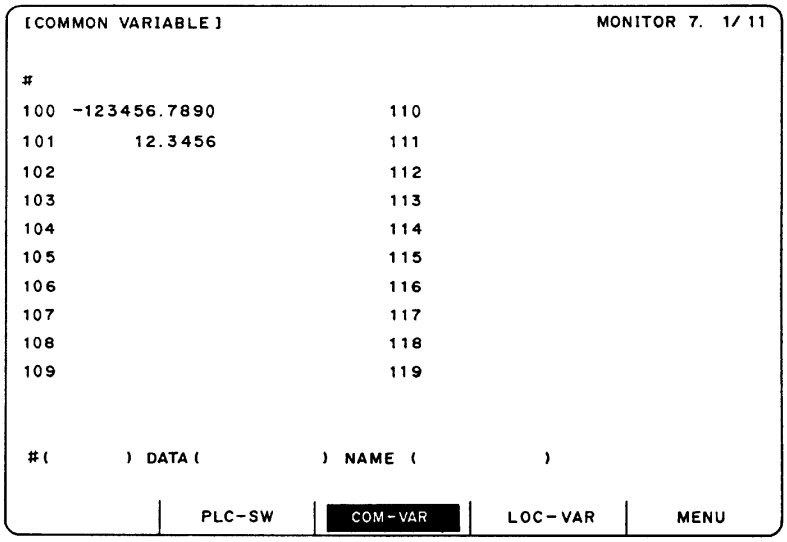
The switch mark of OPTIONAL STOP is set to the up position, indicating the switch on state.

To set the up-position switch (ON state) to OFF (down-position switch), set the number of the ON-state switch in # ( ) and press the [INPUT] key.

The PLC switch names (message display) and the function to reverse selected message display are prepared by using user PLC. These vary depending on the machine manufacturer.

2.2.6 Common Variable

When the menu key)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).



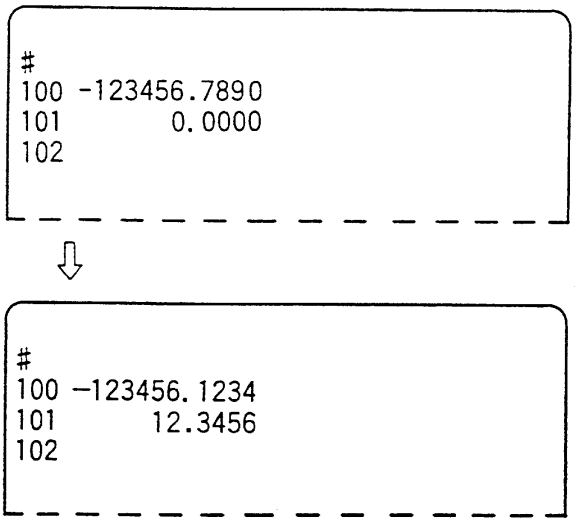
Display item	Explanation
# 100 -123456.7890 101 12.3456 102	The variable numbers and contents are displayed. If variable data is "null", the data display field will be blank. If the number of columns of data is too large (the data contains more than six characters in the integer part or more than four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

2.2.6.1 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.

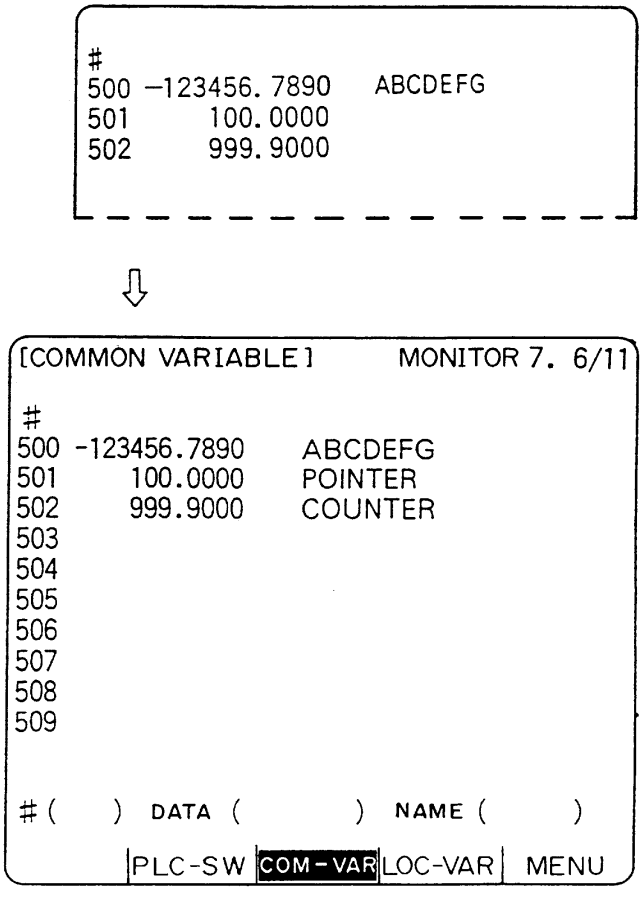
```
(
#101=12.3456
(
```



(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];
(
```



2.2.6.2 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] key is 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] key is first pressed, the screen is changed to the page corresponding to the setup variable number. If again the [INPUT] key is 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.

2.2.6.3 Common Variable Data Deleting

To delete all data set as common variables at a time, press the [SHIFT] [CAN C.B.] 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.

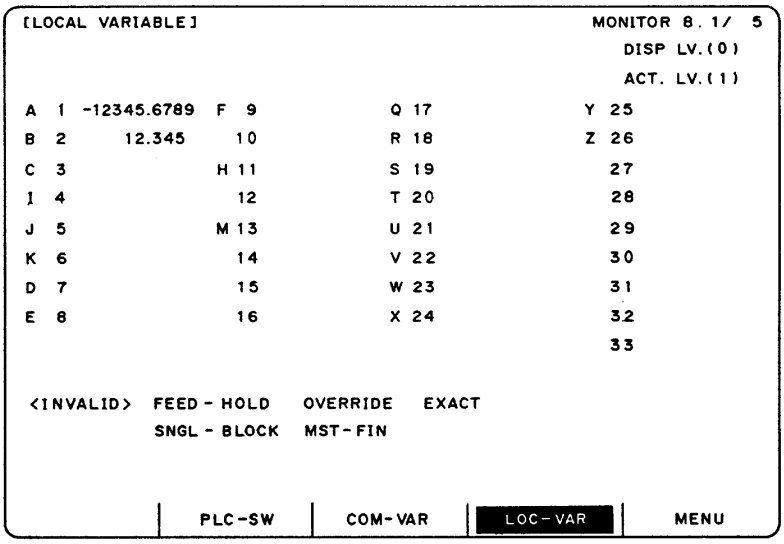
When [SHIFT] [CAN C.B.] keys are pressed, only the display is deleted. When the [INPUT] key is then pressed, the variable data is deleted. This delete operation causes the common variable data to be "null".

(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.

2.2.7 Local Variable

When the menu key [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

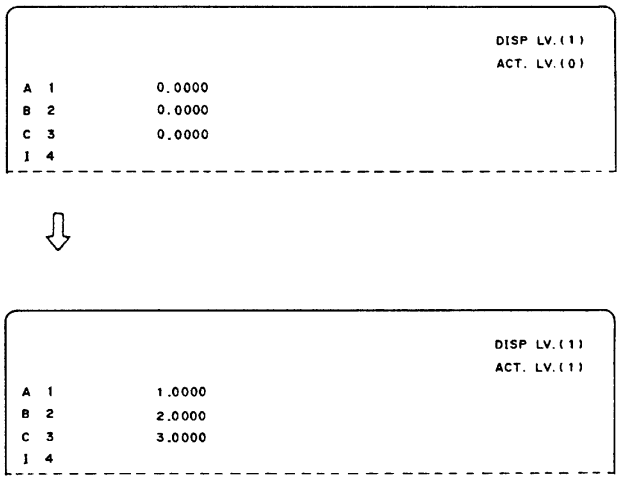
Display item	Explanation
<INVALID> FEED-HOLD OVERRIDE EXACT SNGL-BLOCK MST-FIN	This indicates the modal state of the operation control status by the #3003, #3004 command. FEED-HOLD : Is displayed when command is programmed with #3004 bit 0 set to 1, indicating that feed hold is invalid. OVERRIDE : Is 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.

2.2.7.1 Local Variable Data Display

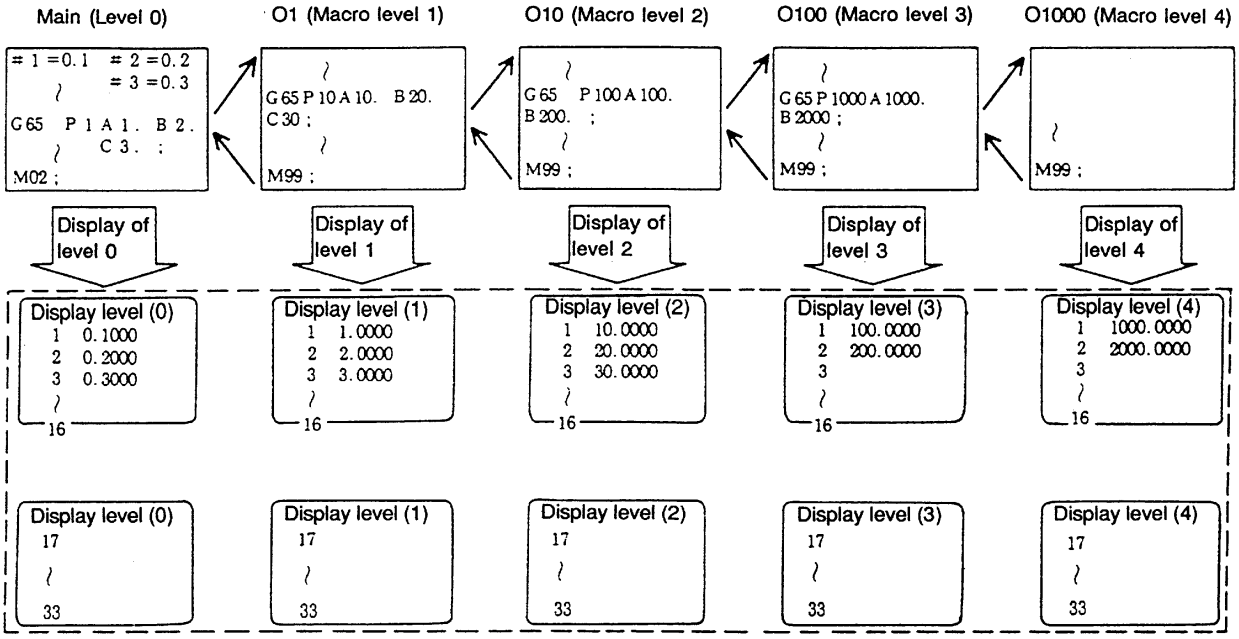
(1) When local variable #1~#32 command exists in user macro or argument specification is made in user macro subprogram call, if the block is executed, the execution result is displayed.

(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:

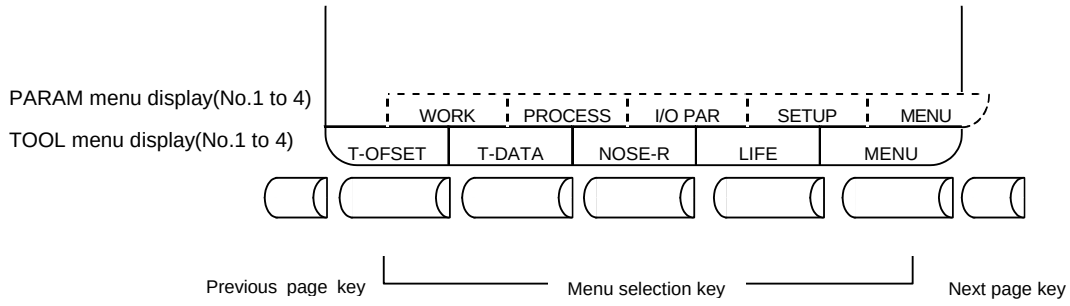


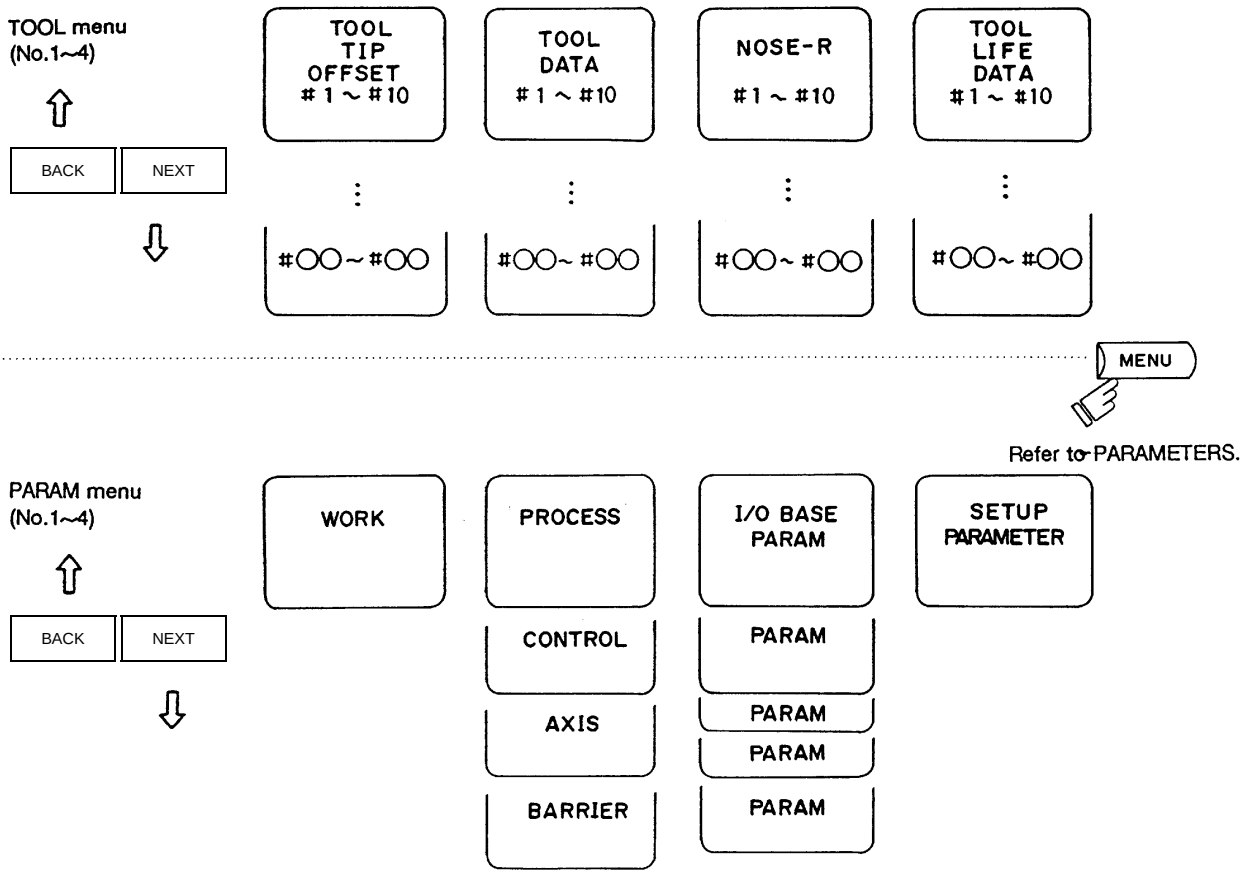
(3) A local variable display page is selected by using the page keys [BACK], [NEXT]. Display can be changed as desired independently of the executing level.

(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.



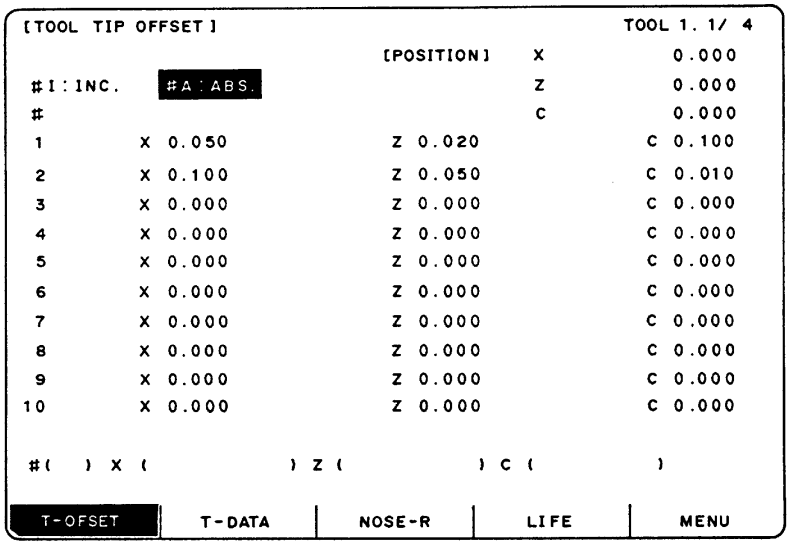


⚠ CAUTION

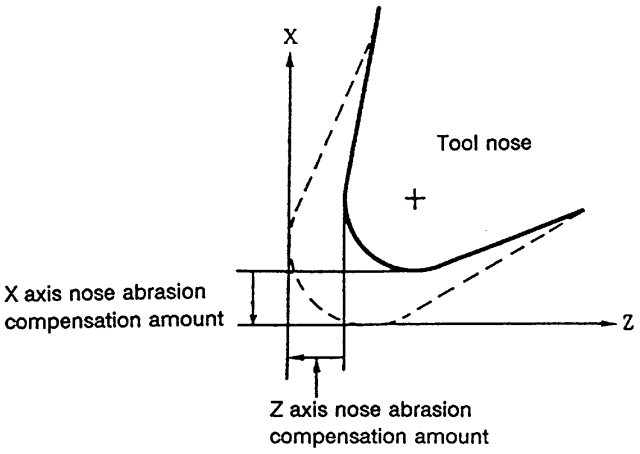
⚠ If a tool offset amount is changed during automatic operation (including during single block stop), the new offset is validated from the command of the next block or blocks onwards.

2.3.1 Wear Data

The TOOL TIP OFFSET screen will appear when the menu key [T-OFSET] 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)
TOOL TIP OFFSET	X, Z, C	Tool wear	±99.999 (mm)
TOOL DATA	X, Z, C	Tool length offset	±999.999 (mm)
NOSE-R	R	Tool radius (nose R)	0~99.999 (mm)
	r	Tool radius (nose R) wear	0~99.999 (mm)
	P	Tool nose point	0~8

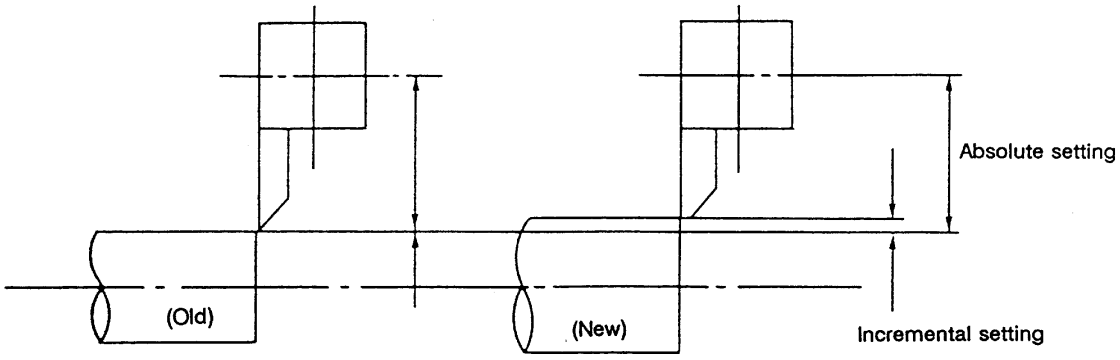
(Note) When parameter "#1019 dia" (diameter command) is set to 0, set the radius. When it is set to 1, set the diameter.

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 CAN C.B 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 ( )

☞

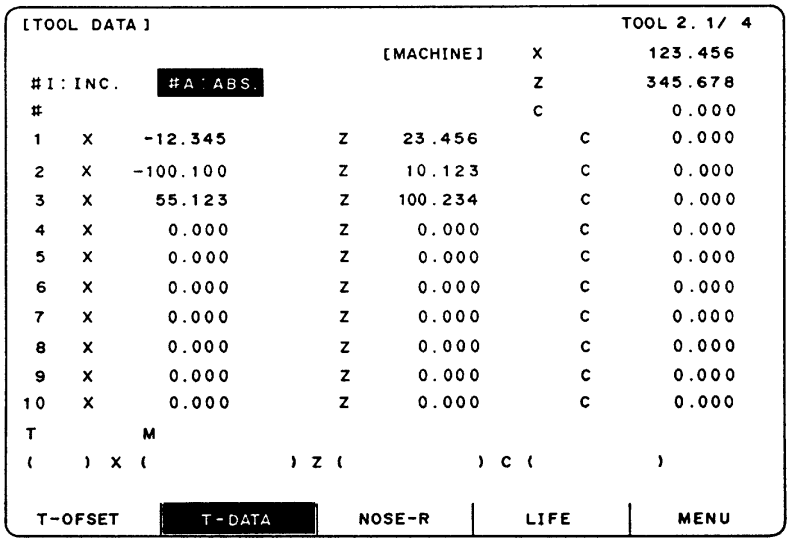
[TOOL DATA]
#I:INC. #A:ABS.

"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

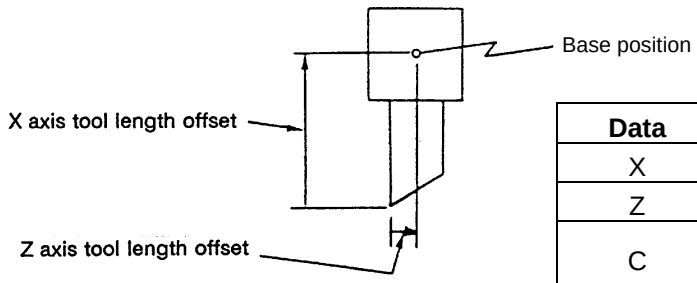
The TOOL DATA screen will appear when the menu key [T-DATA] is pressed.



#	X	Z	C
1	-12.345	23.456	0.000
2	-100.100	10.123	0.000
3	55.123	100.234	0.000
4	0.000	0.000	0.000
5	0.000	0.000	0.000
6	0.000	0.000	0.000
7	0.000	0.000	0.000
8	0.000	0.000	0.000
9	0.000	0.000	0.000
10	0.000	0.000	0.000

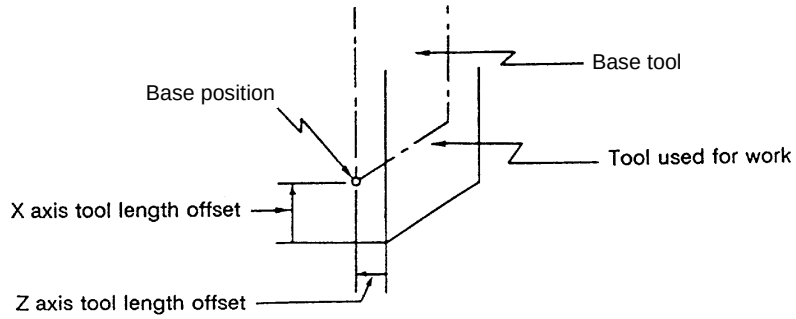
Set the tool length in respect to the programmed base position of each tool used.
When the tool compensation No. is designated by the tool command (T command), compensation is carried out matching the wear data of the previous screen. Generally, the programmed base point position is either the tool rest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



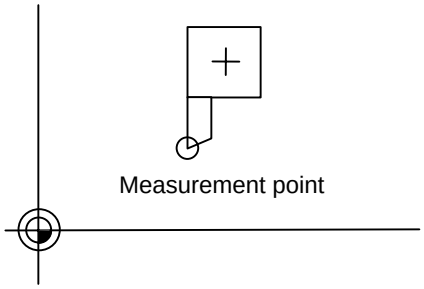
2.3.2.1 Manual Tool Length Measurement I

(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

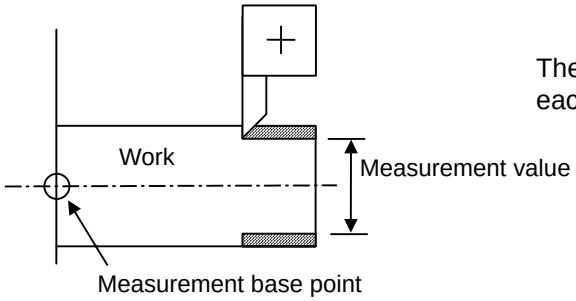
Obtain the tool length with the tool nose placed on the measurement point.



Set the measurement point in parameter "#2015 tlml" beforehand.

(b) Measurement value input method

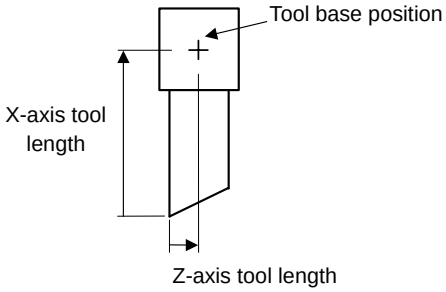
Actually cut the workpiece. Measure its dimensions, and obtain the tool length from the measured values.



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

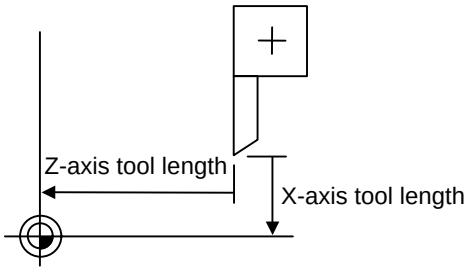
(Note) The tool length from tool length measurement I is as follows, depending on the whether the 1st reference point coordinate values have been set.

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.

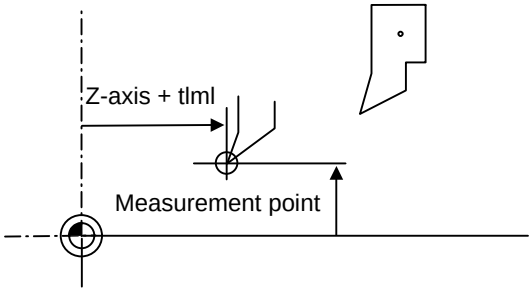
If the 1st reference point coordinate values have not been set:



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

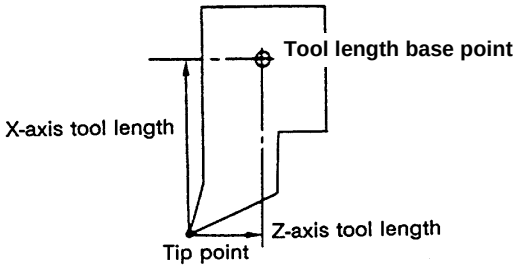
Set the type selection to the base point method. (Set “#1102 tlm” to 0).
To carry out the reference point method, a point to place the tool nose on (measurement point) is required.
Set the measurement point in parameter "#2015 tlml" beforehand.



(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)

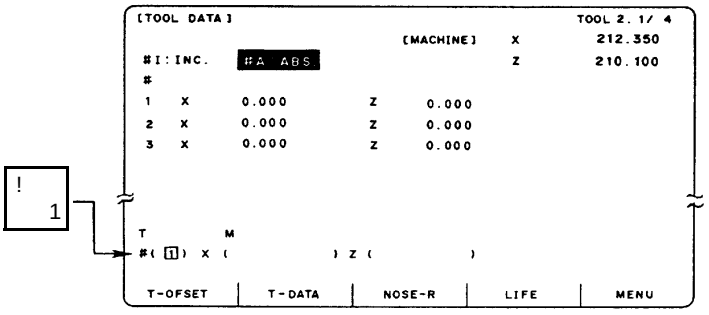
The expression above is used for automatic calculation in the base point method. When the tool nose is placed on the measuring point, the distance from the tool nose to the tool length base point is calculated.



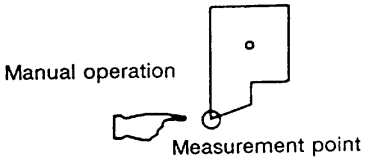
< Measuring procedure for the base point method >

(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.



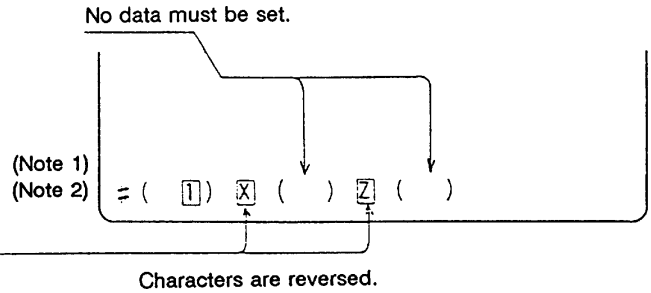
(4) Select the axis to be measured.

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]

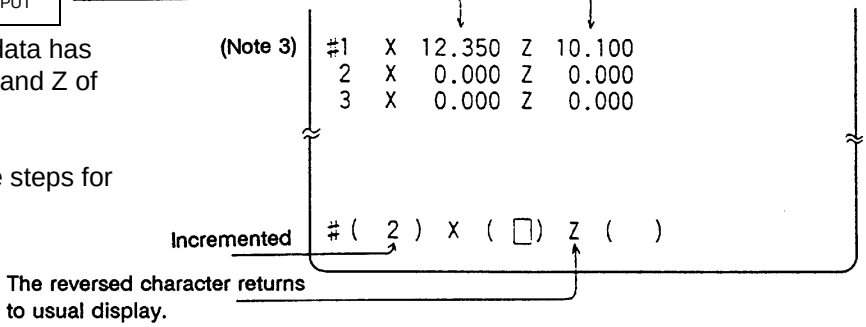


(5) The data is automatically calculated and written.
(The data is written for the axis shown in highlighted characters.)

[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).

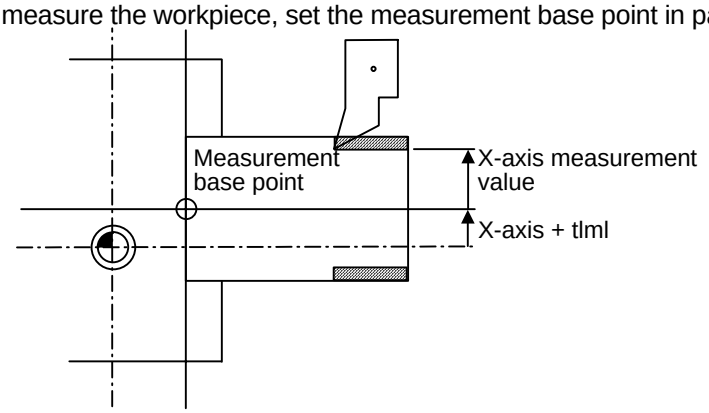
(Note 2) If an axis having an error (reference point return incomplete axis, etc.) is selected, the characters will not be highlighted. An error message will appear.

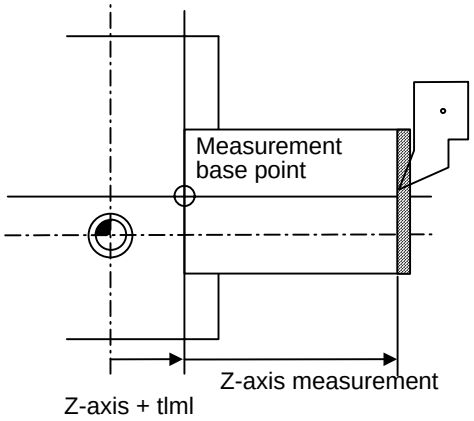
(Note 3) For a diameter command, the diameter value is written.
For a radius command, the radius value is written.

(3) Measurement value input method

Set the type selection to the measurement value input method. (Set "#1102 tlm" to 1).

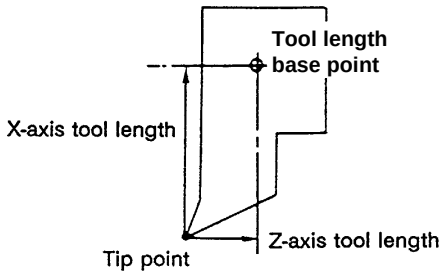
To carry out the measurement value input method, a workpiece for measuring is required. To measure the workpiece, set the measurement base point in parameter "#2015 tlml–" beforehand.





Tool length = Machine value - Measurement base point (tlml) - Measurement value

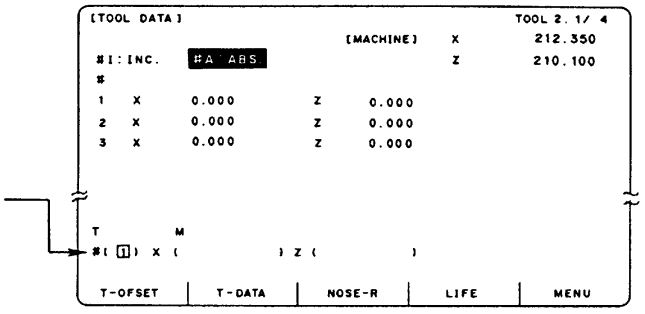
The expression above is used for automatic calculation in the measurement value input method.



< Measuring procedure for the measuring value input method >

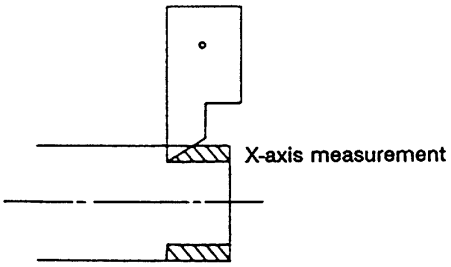
(1) Select the TOOL DATA Screen.
(2) Set the tool No. to be measured in # ( ).
(Select the tool before this step. It can be selected using a manual numerical command.)

(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.

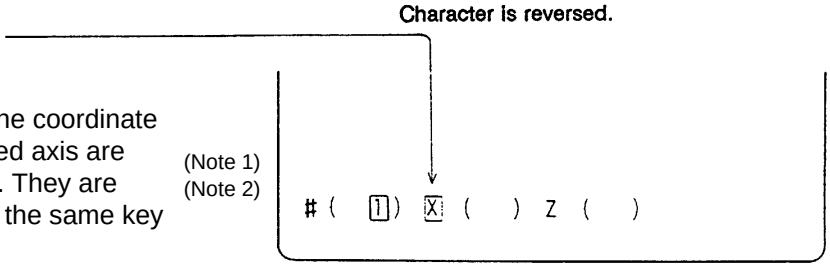
(For the Z axis, execute face turning.)



(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

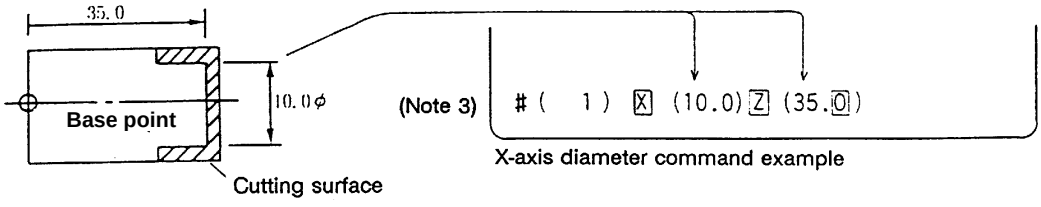
In this way the machine coordinate values of the measured axis are stored in the memory. They are canceled by pressing the same key twice.

∗ Also repeat steps (3) and (4) for the Z axis.

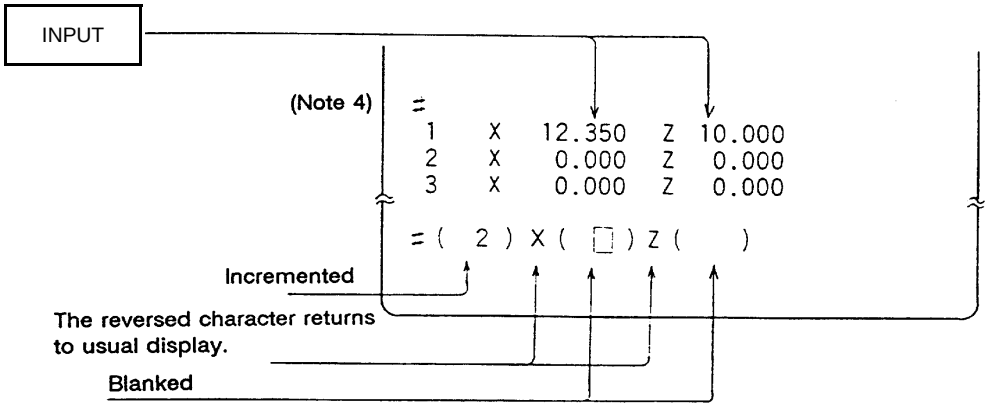


(5) Retract the tool, and stop the spindle.
(6) Measure the workpiece, and set the measurement values in the setting areas of each axis. Set the values for all axes shown in highlighted characters.

(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. The operation is as follows:

```
 T        M
#(  ) DATA (       )
```

Set [M] (manual) in the first set of parentheses in the setting area.

☞

```
 T        M
#( M) DATA (       )
```

1) This operation is the same for M or T commands.

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

☞

```
 T        M
#(       ) DATA (       )
```

1) 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.

☞

```
 T20      M6
#(  ) DATA (       )
```

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

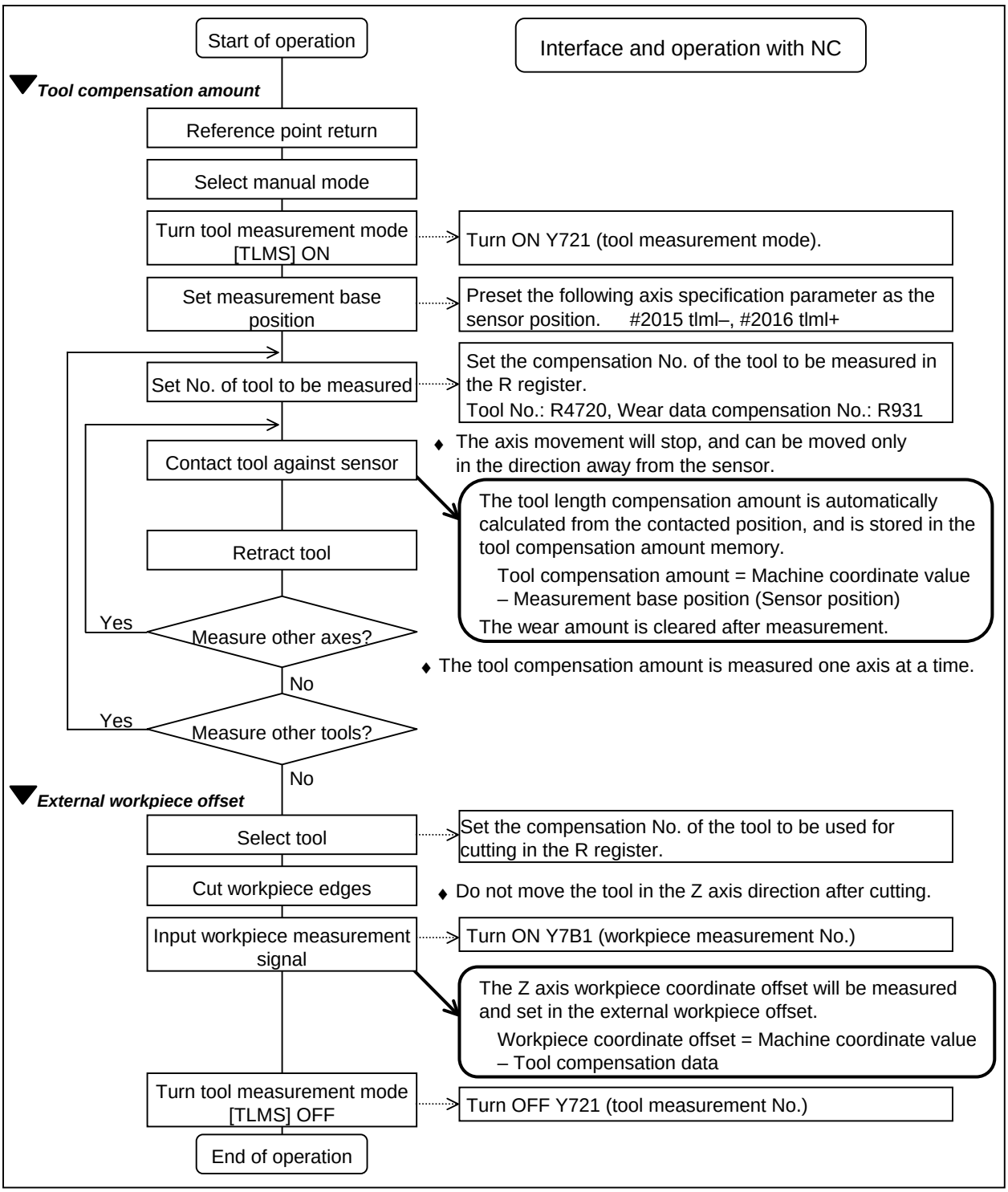
2.3.2.3 Tool Presetter

(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

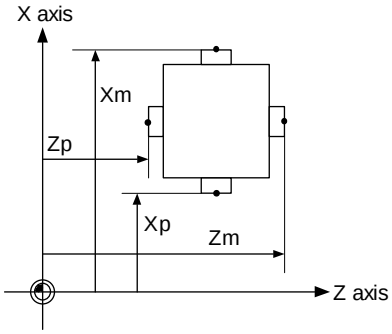
1) Zero point return
After turning the power ON, establish the coordinate system by carrying out dog-type zero point return.
When using the absolute position detection specifications, carry out initialization if the absolute position is not established.

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

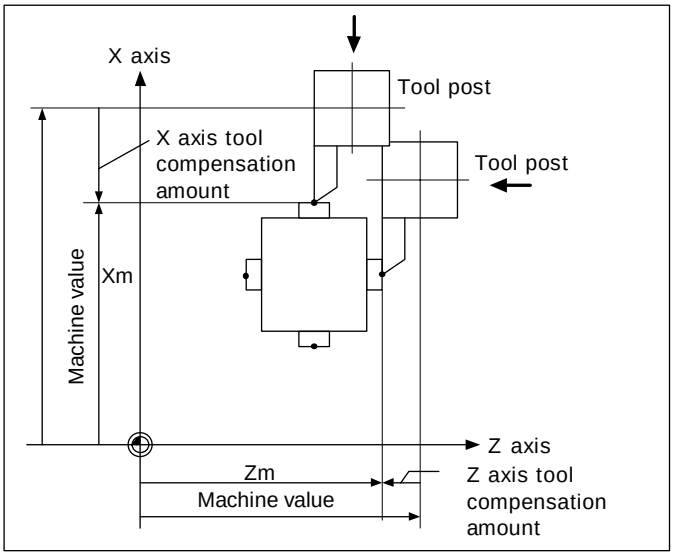
5) Select the tool
Select the tool to be measured.
Set the compensation No. of the tool to be selected as a BCD code in R4720.
Set the compensation No. of the wear data to be cleared after measurement as a BCD code in R931.
(The tool No. data is input from the PLC to the NC.)

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.
Notes) 1. Set the compensation No. of the tool to be selected in the R register (R register corresponding to the compensation No.).
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.
Notes) 1. Do not move the tool in the Z axis direction after cutting the workpiece edges.
2. If the edges do not need to be cut, position to the measurement position.

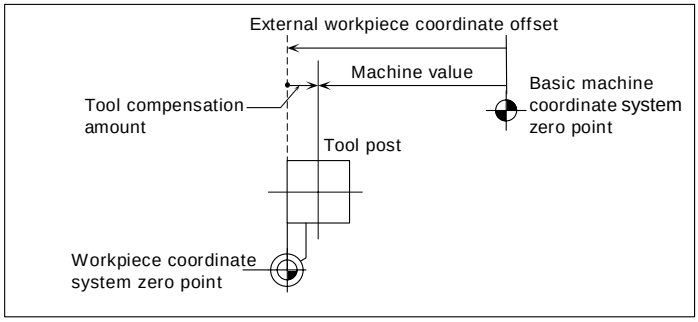
6) Set the Z axis external workpiece offset data with the workpiece measurement signal input
Turn ON the workpiece measurement signal. The Z axis external workpiece coordinate offset data will be automatically calculated from the machine value at the time the signal is turned ON and the tool compensation 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	R946, R947	R946, R947
	1	0/1		
1	0	0	R220, R221	R946, R947
		1	R948, R949	R946, R947
	1	0/1	R948, R949	R946, R947

(Notes) 1. If the compensation No. is 0, the compensation amount will be calculated as "0".
2. If the compensation No. exceeds the number of offset sets in the specifications, the "E76 TOOL No. ERROR" error will occur.
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

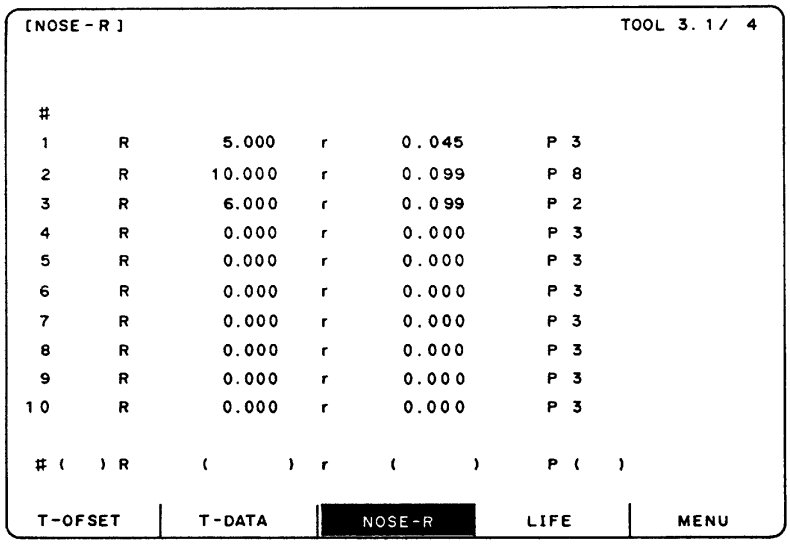
7) Turn the tool measurement mode signal OFF.
This completes the measurement of the external workpiece coordinate offset.
When carrying out this operation independently, follow steps 1) to 7), and when carrying out after measuring the tool compensation amount, carry out steps 4) to 6) between 9) and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

(4) Precautions

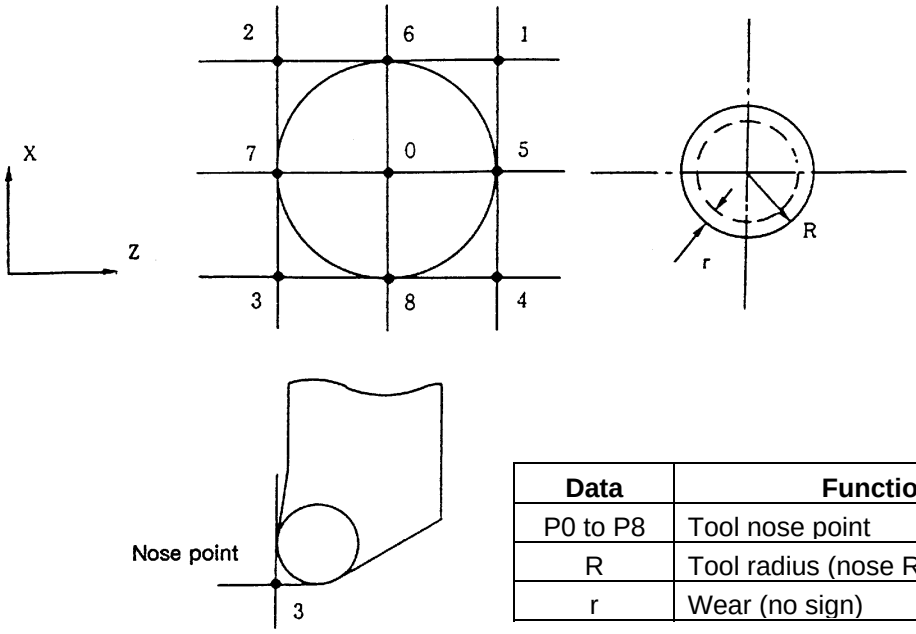
1) When entering the sensor area, the axis can move only in one direction selected from +X, –X, +Z, –Z, (+Y, –Y).
If two axes (ex. +X, –Z) are moved simultaneously, it will not be clear which contact surface was contacted, so the measurement will not be made. Note that the error "E78 AX UNMATCH (TLM)" 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 key [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.

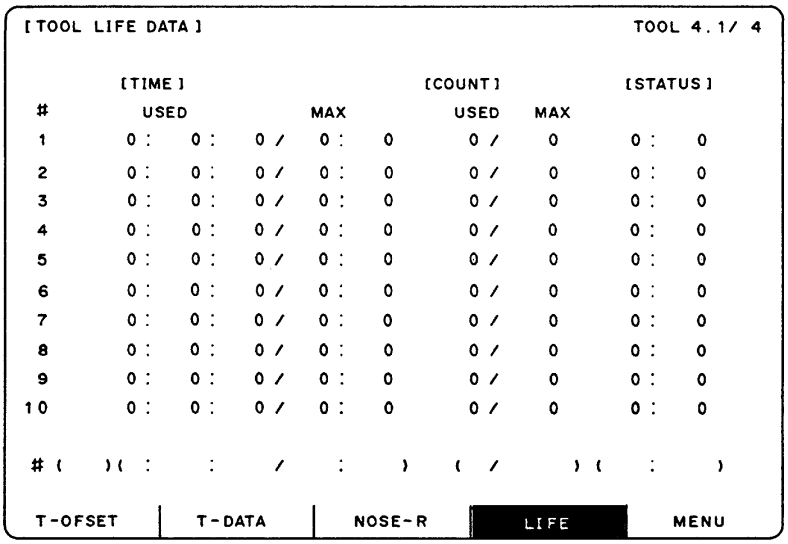


Data	Function
P0 to P8	Tool nose point
R	Tool radius (nose R) (no sign)
r	Wear (no sign)

(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 key [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 CAN C.B 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.

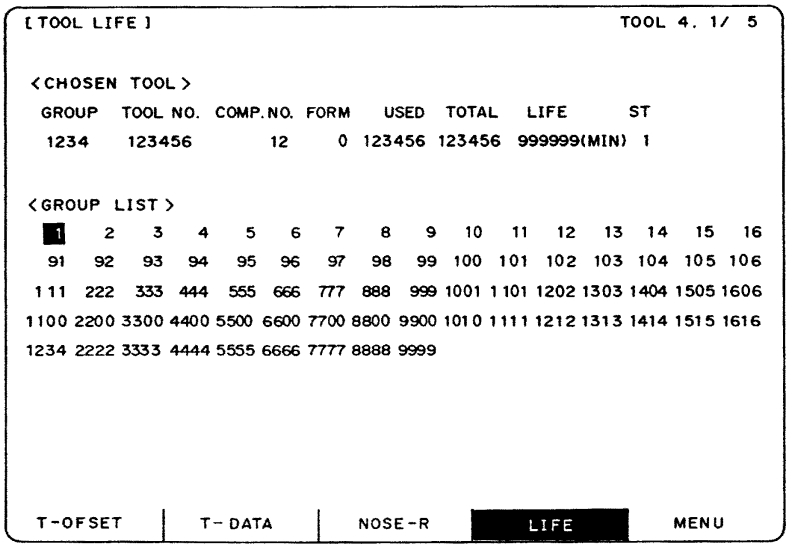
2.3.5 Tool Life Management II ("#1096 T_Ltype" is 2)

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) Page 1 of the TOOL LIFE screen displays the life management data of the tool currently being used and the list of registered groups of tools. This page is mainly used for monitoring tool life data in group units.
The monitoring of each tool's data in the group, and setting of the group No. and tool life management data are carried out on the next page.



TOOL LIFE screen Page 1

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

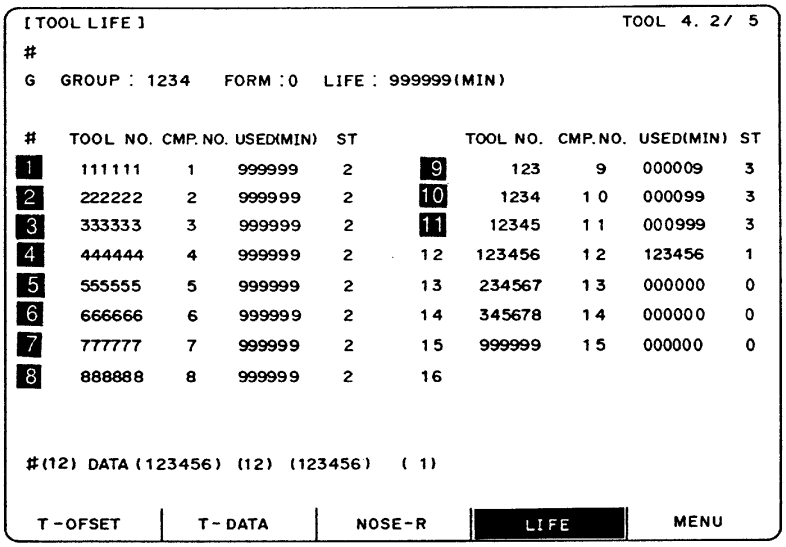
TOTAL : For tools using multiple compensation Nos., the total of the usage data for each compensation appears here. If there is only one compensation No., the data will be the same as "USED".

< 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 + CAN C.B + INPUT .

(2) Page 2 is used to set and display a group's life management information.



TOOL LIFE screen Page 2

(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 + CAN C.B + 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.

2.3.5.2 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.

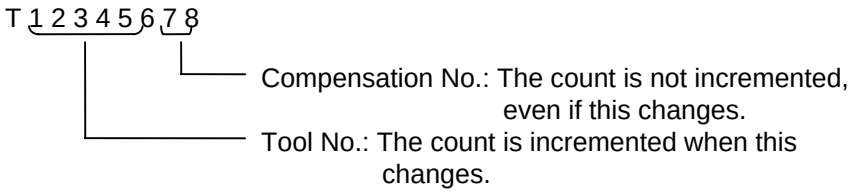
(Note) • The max. life value is 999999 min.
• The data on the TOOL LIFE screen is displayed in minute units.

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

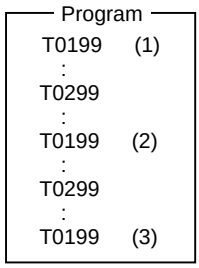
Incrementation is carried out when the No. of the tool being used changes by the execution of a tool selection command (T****99) during the cutting mode (except during machine lock, miscellaneous function lock, dry run, and single block states). (If the mode never changes to the cutting mode after the 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.

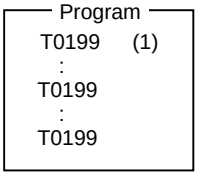
If the T code of the current tool is 12345678:



<< 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 1.

Example of the screen display when using multiple compensation Nos.

• Time method (life: 100000 min.)

#	TOOL No.	CMP.No.	USED(MIN)	ST
1	101010	1	40000	2
2	101010	2	40000	2
3	101010	3	30000	2
4	202020	4	20000	1
5	202020	5	20000	1
6	202020	6	15000	1
7	303030	7	0	0

∗ The life of tool 101010 is the total usage time of #1 to #3.

• Count method (life: 100000 times)

#	TOOL No.	CMP.No.	USED(SET)	ST
1	101010	1	50000	2
2	101010	2	50000	2
3	101010	3	0	2
4	202020	4	40000	1
5	202020	5	40000	1
6	202020	6	0	1
7	303030	7	0	0

∗ The life of tool 101010 is the total count of #1 to #3.

2.3.5.3 Parameters

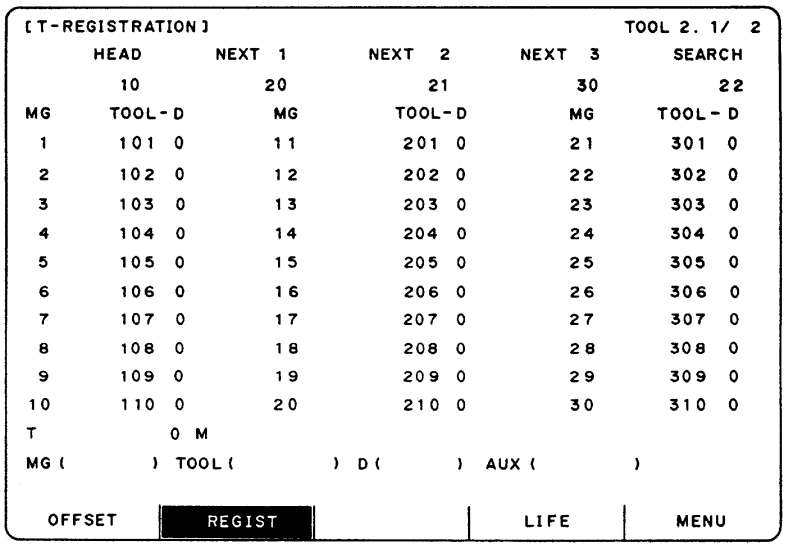
The tool life management specifications will differ according to parameter "#1096 T_Ltype" and "#1106 Tcount". Confirm the explanation for the relevant setup parameter data item.

2.3.6 Tool Registration

The T-REGISTRATION screen will appear when the menu key [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.

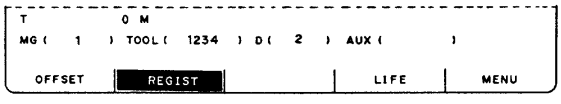
2.3.6.1 Outline of Functions

(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.

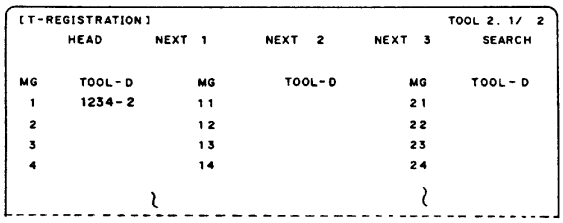


2.3.6.2 Tool Registration in the Magazine Pot

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 ( ) disappears.
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.

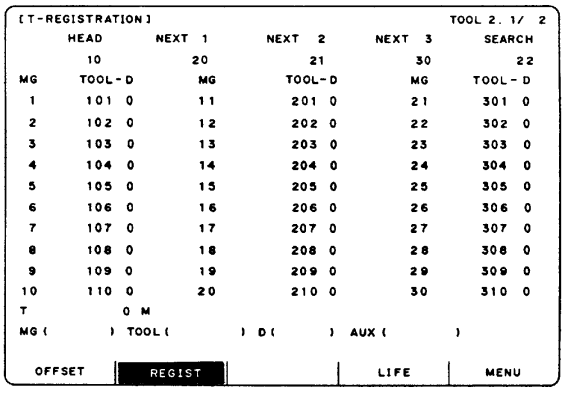
2.3.6.3 Tool Registration in the Spindle, Standby and Indexing Areas

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 ( ).

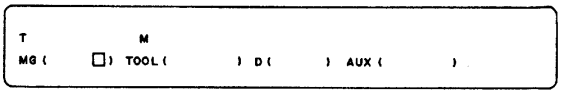
Press the [INPUT] key. ☞

All data displayed in USAGE and MG1 to MGn is cleared to 0.

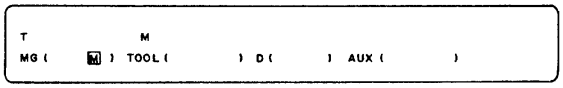
2.3.6.5 Manual Numerical Command Operation (M, T) on the TOOL REGISTRATION Screen

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. The operation is as follows:



Set [M] (manual) in the first set of parentheses in the setting area. ☞



1) This operation is the same for M or T commands.

Press the [INPUT] key. The mode changes to the manual numerical command mode. ☞

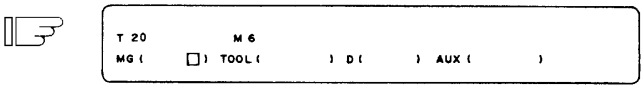
1) The data in the setting area is cleared, and the cursor disappears from the screen.

(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

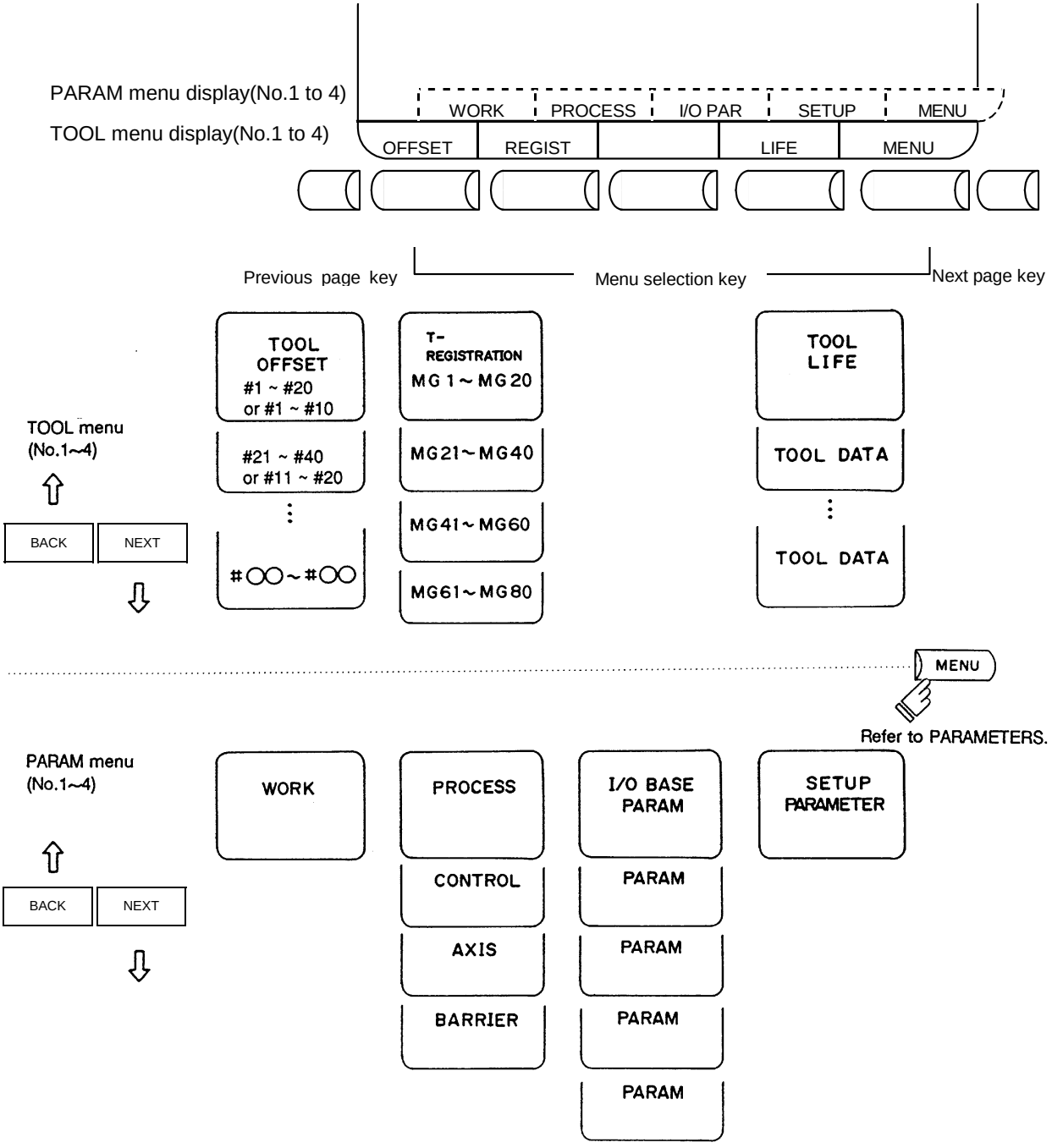
Press the [↓] key. The normal data setting mode returns.



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

2.3 (II). Tool Offset (M system)

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



⚠ CAUTION

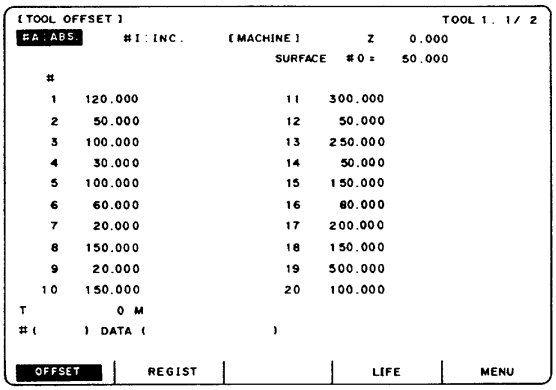
⚠ If a tool offset amount is changed during automatic operation (including during single block stop), the new offset is validated from the command of the next block or blocks onwards.

2.3.1 Tool Offset

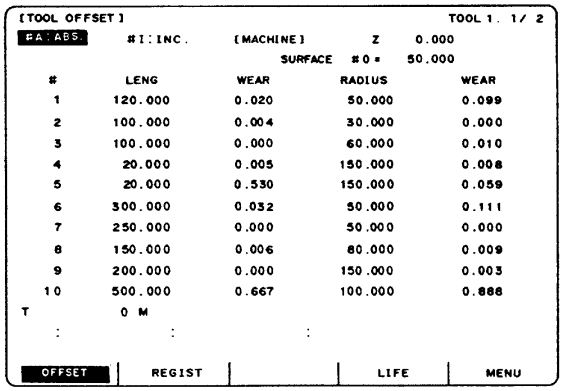
When the menu key [OFFSET] is presented, the TOOL OFFSET screen is displayed.

(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 memory type II

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 [P] 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 [O] in # ( ), then press the [INPUT] key; the absolute mode is set. (For details, see Sections 6.3.1.3.)

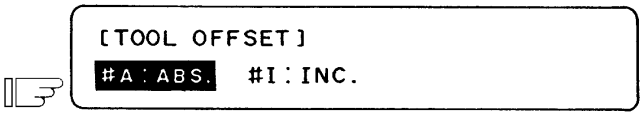
2.3.1.2 Tool Offset Data Clear

(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 [CAN C.B] and [INPUT] keys.

2.3.1.3 Tool Offset Data Setting Modes (Absolute and Incremental)

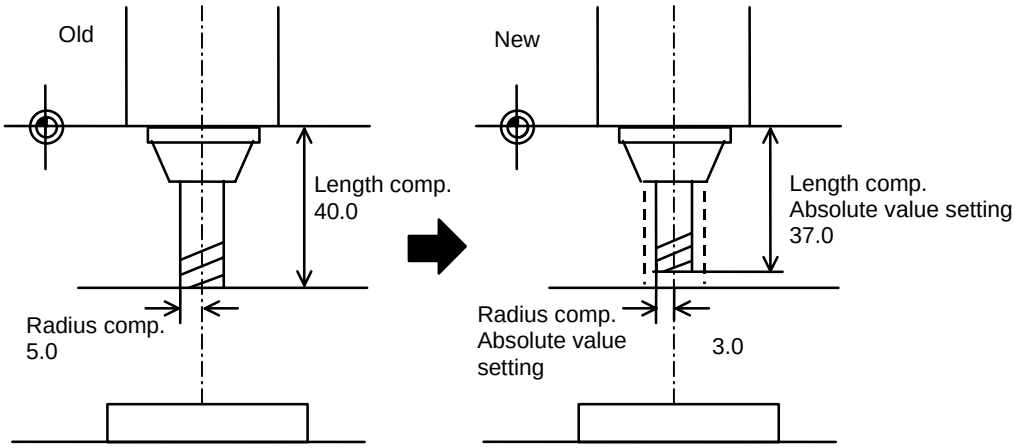
(1) Absolute value setting
Change to the absolute value setting mode as follows:

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)

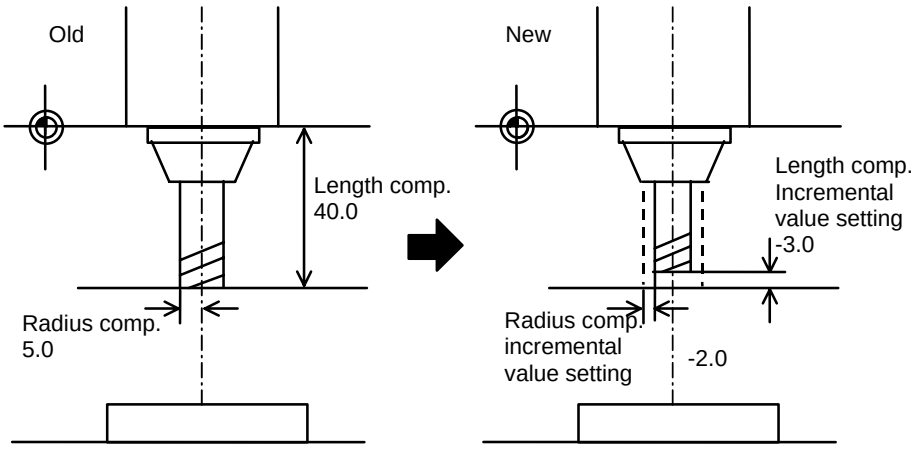
Display		Setting	Display
#3 40.000	→	# (3) (37)	#3 37.000

(2) Incremental value setting
Change to the incremental setting mode as follows:

Enter I in # ( ), then press the input key. # (I) DATA ( )	☞	[TOOL OFFSET] #A:ABS. #I:INC.

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

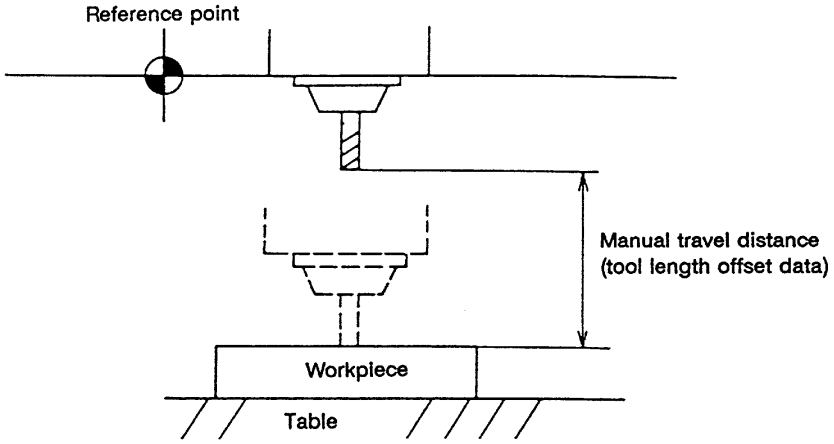
The mode thus set is retained even after the screen is changed or after power has been turned off.

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

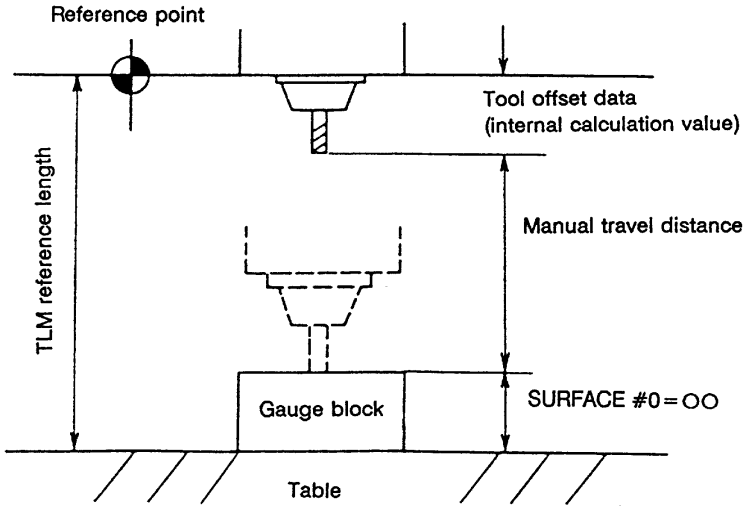
When the tool is placed in the reference point, the distance from the tool tip to measurement point (work top end) can be measured and set as tool offset data.



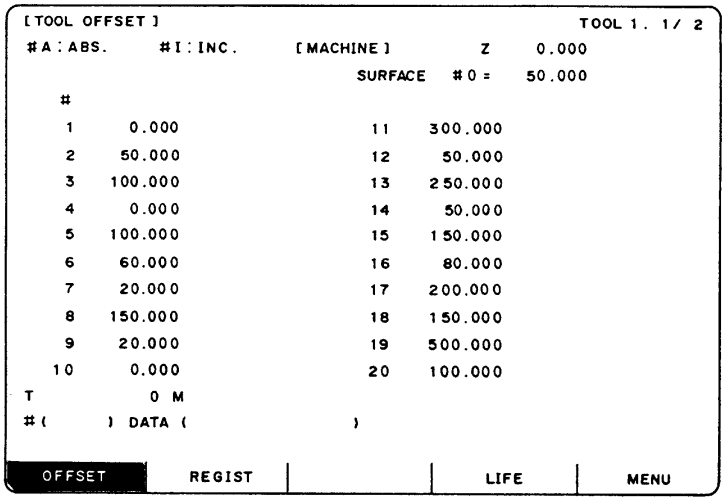
When TLM reference length (#1102 tlm) = 0 and SURFACE #0 = 0 are set, tool length measurement I mode is set.

(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 [O] 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.

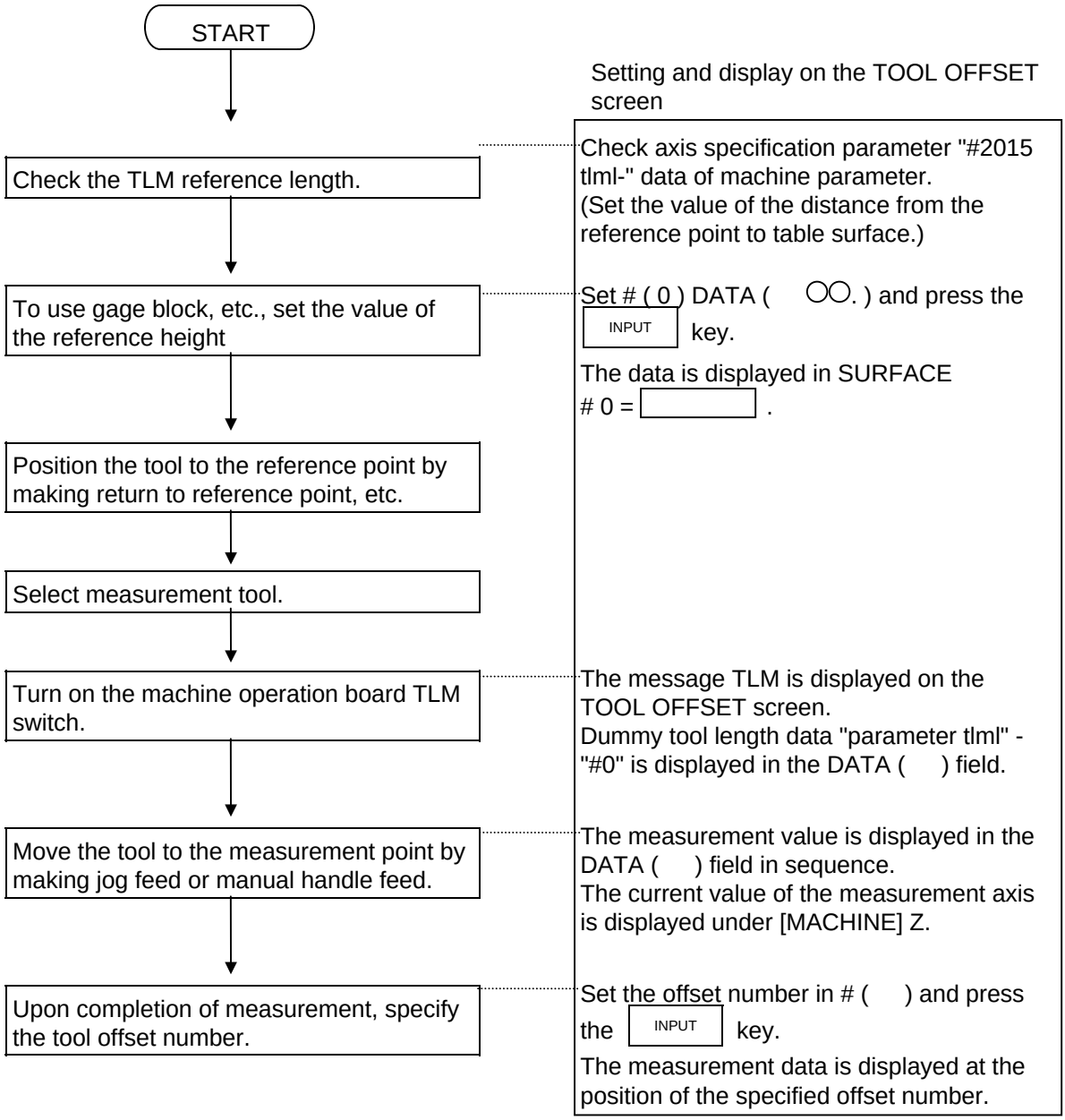
The measurement value is displayed in DATA ( ) field in sequence.
The measurement value is also displayed at [TLM].
The current value of the measurement axis is displayed at [MACHINE].

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".

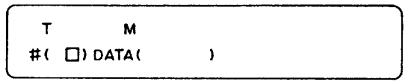
(2) Tool length measurement II



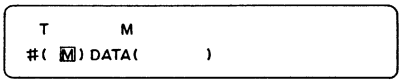
2.3.1.5 Manual Numeric Command Operation on the TOOL OFFSET Screen (M, T)

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. Change the mode by the following operations:

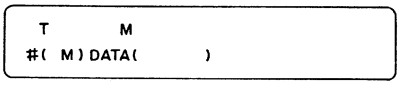


Set [M] (Manual) in the first parenthesis pair of the setting field.



1) This operation is necessary regardless of the command (M, T).

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



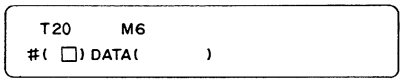
1) Data is cleared from the setting field. The cursor is also cleared from the screen.

(2) Execution the manual numeric command ... Execute this after operation (1) above.
1) 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].
2) Input the specified numerics from keys.
3) 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 "2.2.1.3 Manual Numeric Value Command" in 'POSITION' of 'MONITOR' screen for details.

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

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



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

2.3.2 Tool Registration

When the menu key) 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.

[T-REGISTRATION] TOOL 2. 1/ 2

HEAD		NEXT 1		NEXT 2		NEXT 3		SEARCH	
10		20		21		30		22	
MG	TOOL-D	MG	TOOL-D	MG	TOOL-D	MG	TOOL-D	MG	TOOL-D
1	101	0	11	201	0	21		301	0
2	102	0	12	202	0	22		302	0
3	103	0	13	203	0	23		303	0
4	104	0	14	204	0	24		304	0
5	105	0	15	205	0	25		305	0
6	106	0	16	206	0	26		306	0
7	107	0	17	207	0	27		307	0
8	108	0	18	208	0	28		308	0
9	109	0	19	209	0	29		309	0
10	110	0	20	210	0	30		310	0

T 0 M
MG ( ) TOOL ( ) D ( ) AUX ( )

OFFSET | REGIST | | LIFE | MENU

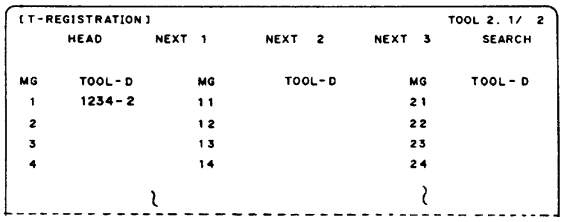
2.3.2.2 Tool Registration in Magazine Pot

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

T 0 M
MG (1) TOOL (1234) D (2) AUX ( )

OFFSET | REGIST | | LIFE | MENU

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.

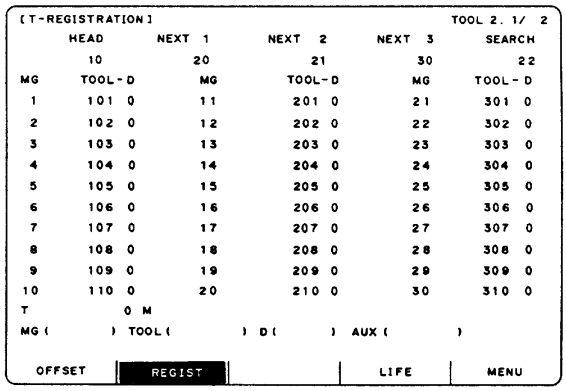
2.3.2.3 Tool Registration in HEAD, NEXT, and INDEX

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
“8” in TOOL ( ).

Press the INPUT key.



“8” 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

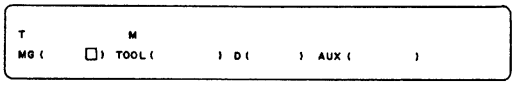
Set “CL” in MG ( ).

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.

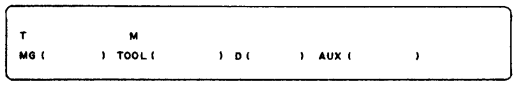
(1) Changing the mode from usual data setting to manual numeric command
In the usual data setting mode, the cursor is displayed in the data setting field. It is not displayed in manual numeric command mode. By checking this difference, make sure that the mode has changed. Change the mode by the following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



Set M (Manual) in the first parenthesis pair of the setting field. ☞

1) This operation is necessary regardless of the command (M, T).

Press the INPUT key. The mode changes to manual numeric command. ☞



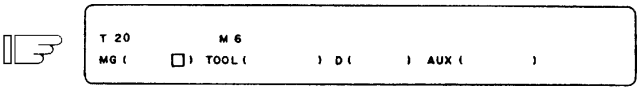
1) Data is cleared from the setting field. The cursor is also cleared from the screen.

(2) Execution of the manual numeric command ... Execute this after operation (1) above.
1) 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.
2) Input the specified numerics from keys.
3) 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 "2.2.1.3 Manual Numerical Value Command" in 'POSITION' of 'MONITOR' screen for details.

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

Press the ↓. The usual data setting mode returns.



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

2.3.3 Tool Life

When the menu key [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

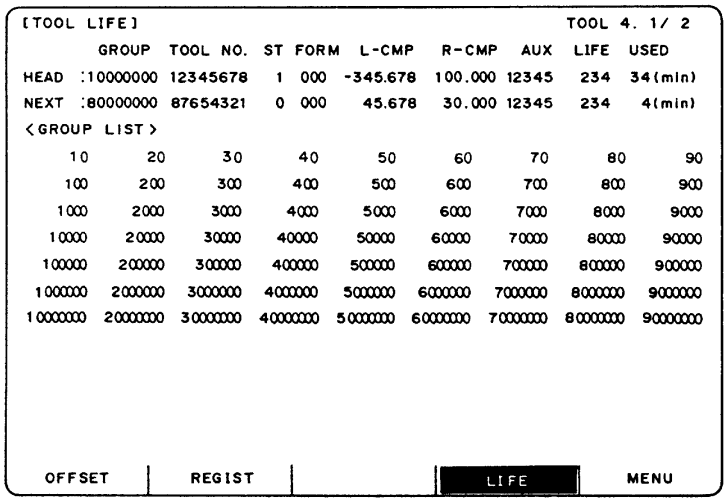
The specifications for TOOL LIFE changes depending on parameter "#1096 T_ Ltype" and "#1103 T_Life" to "#1106 T count". Confirm the explanations described in the appropriate section in setup parameter.

(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.

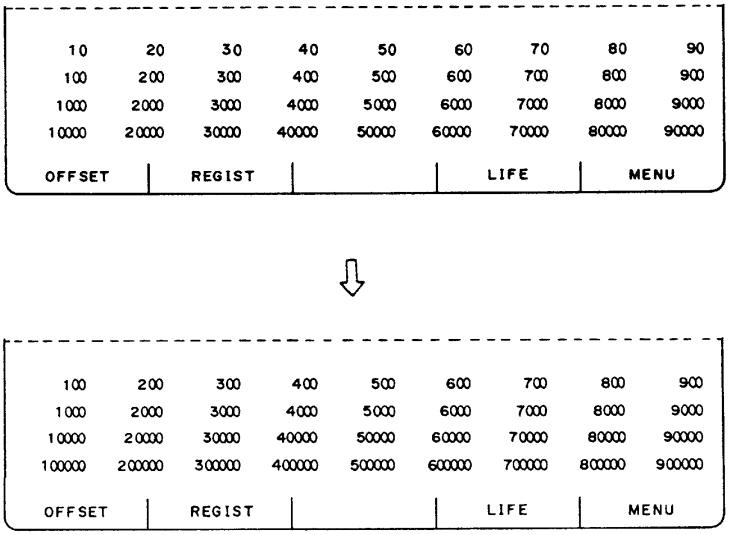
2.3.3.2 TOOL LIFE Screen Data Display

(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.

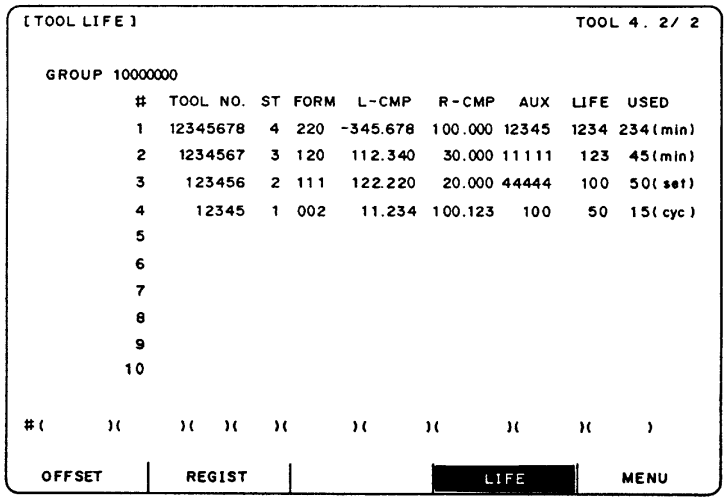


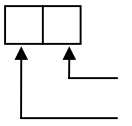
One-line scroll by pressing the ↓ key.

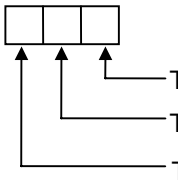
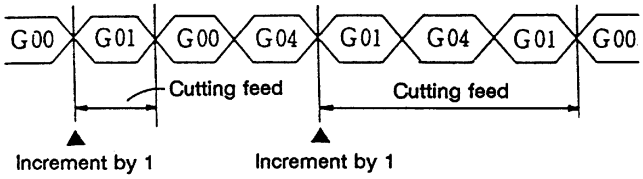
(2) TOOL LIFE data screen

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 9999.999
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.

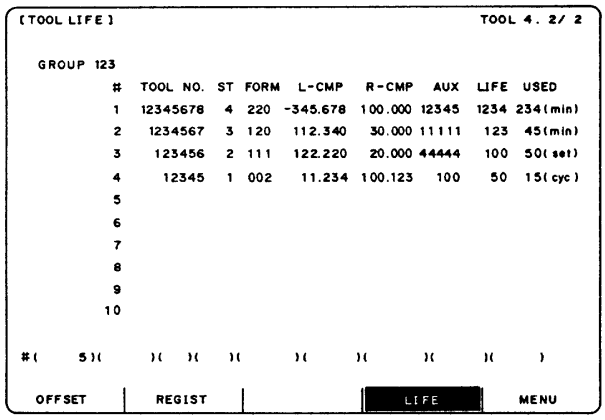
2.3.3.3 TOOL LIFE Data Display and Setting (TOOL LIFE Data Screen Page 2)

(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.

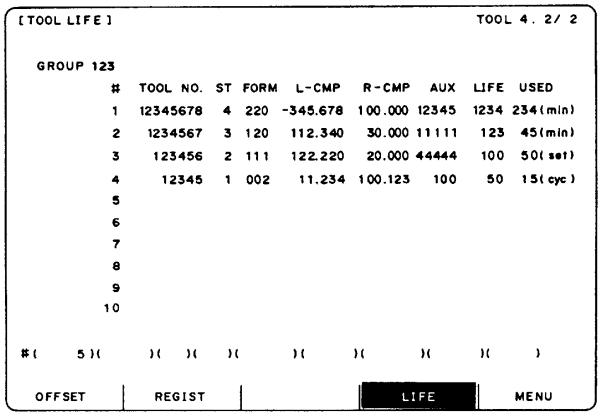


The tool data registered in group 123 is displayed in the registration order. If the data exceeds one screen, the remaining data can be seen by using the [NEXT] key.

(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.

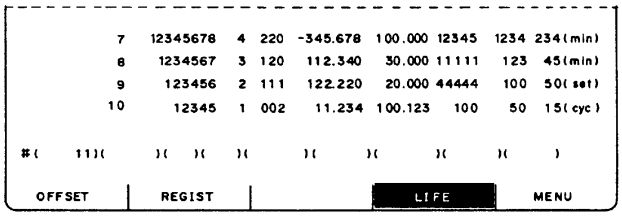


Set necessary data of TOOL NO. to USED in the corresponding parenthesis pairs in the ascending order of the (#) numbers, then press the [INPUT] key.

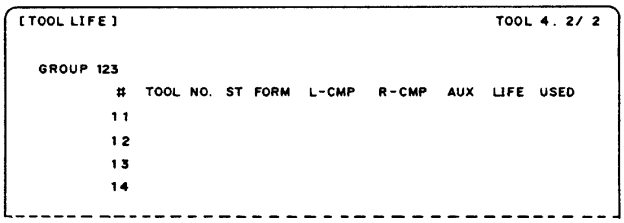
The data setting number incremented by one is set in # ( ).

(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, CAN C.B then INPUT.

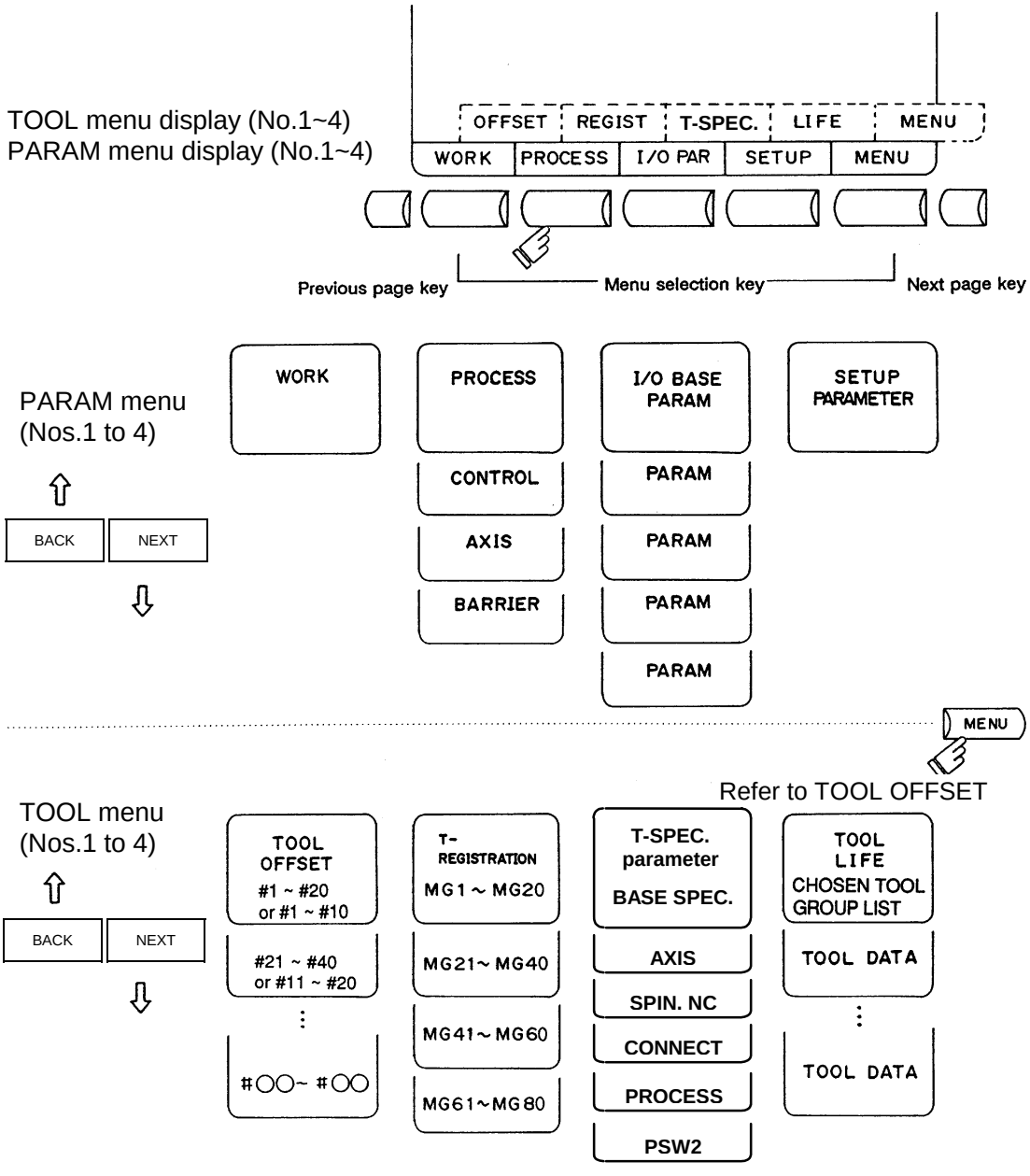
2.3.3.4 Clear of All TOOL LIFE Data (HEAD, NEXT, GROUP LIST Screen Page 1)

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

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 key [MENU] on the TOOL screen.



(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	#8052 PULL UP
	#8005 ZONE r	#8016 G71 MINIMUM	#8053 G73 U
	#8006 ZONE d	#8017 G71 DELTA-D	#8054 W
	#8009 DSC. ZONE	#8018 G84/G74n	#8056 G74 RETRACT
	#8010 ABS. MAX.	#8027 Toler-1	#8057 G76 LAST-D
	#8011 INC. MAX.	#8028 Toler-2	
	#8012 G73n	#8051 G71 THICK	
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.

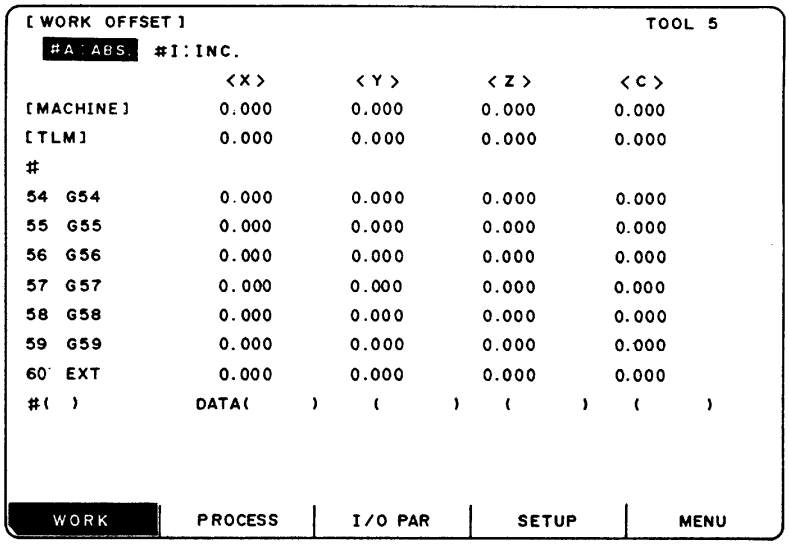
The parameter input setting units are as follows.

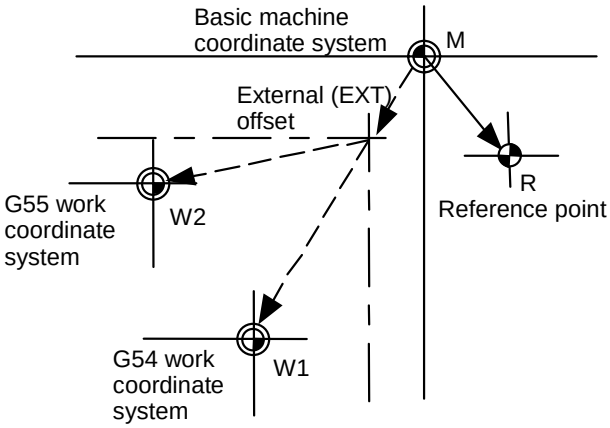
Input unit "#1003 iunit"	**Linear axis "#1017 rot"=0**		**Rotary axis "#1017 rot"=1**
	Machine constant:mm "#1040 M_inch"=0	**Machine constant:inch "#1040 M_inch"=1**	
B	0.001 mm	0.0001 inch	0.001°
C	0.0001 mm	0.00001 inch	0.0001°

2.4.1 Workpiece Coordinate

Pressing the menu key [WORK] displays the WORK OFFSET screen.

The workpiece coordinate system offset data can be set or displayed for the number of axes.



#	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)

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.

2.4.1.1 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.

2.4.1.2 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.

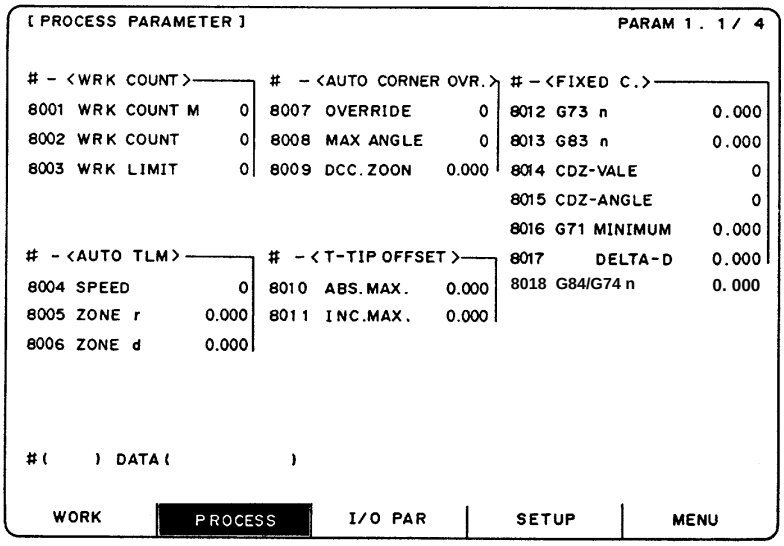
2.4.1.3 Displaying Machine Position Data

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.

2.4.2 Machining Parameters

Pressing the menu key PROCESS displays the PROCESS PARAMER screen.

2.4.2.1 Process Parameters



<WRK COUNT> (No. of workpieces machined)

#	Name	Contents	Setting range (unit)
8001	WRK COUNT M	Set the M code that counts the No. of workpiece repeated machining. The No. will not be counted when set to 0.	0 to 99
8002	WRK COUNT	The current machining No. is displayed. Set the initial value.	0 to 999999
8003	WRK LIMIT	Set the maximum No. of workpieces to be machined. A signal is output to PLC when the No. of machining times is counted to this limit.	0 to 999999

<AUTO TLM> (Automatic tool length measurement)

#	Name	Contents	Setting range (unit)
8004	SPEED	Set the feedrate during automatic tool length measurement.	1 to 1000000 (mm/min)
8005	ZONE r	Set the distance between the measurement position and deceleration start point.	0 to 99999.999 (mm)
8006	ZONE d	Set the allowable range of the measurement point. An alarm will occur if the sensor signal turns ON before zone d from the measurement point or if the sensor signal does not turn ON after zone d is passed.	

<AUTO CORNER OVR.> (Automatic corner override)

#	Name	Contents	Setting range (unit)
8007	OVERRIDE	Set the override value for the automatic corner override.	0 to 100 (%)
8008	MAX ANGLE	Set the max. corner opening angle where deceleration should start automatically. Deceleration will not occur if the angle is larger than this value.	0 to 180 (°)
8009	DSC. ZONE	Set the position where deceleration starts at the corner. Designate the distance from the corner at which deceleration should start.	0 to 99999.99 (mm)

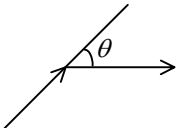
<T-TIP OFFSET> (Wear data input)

#	Name	Contents	Setting range (unit)
8010	ABS. MAX. (For L system only)	Set the max. value when inputting the tool wear compensation amount. Values exceeding this setting value cannot be input.	0 to 99.999 (mm)
8011	INC. MAX. (For L system only)	Set the max. value when inputting the tool wear compensation amount in the incremental mode.	0 to 99.999 (mm)

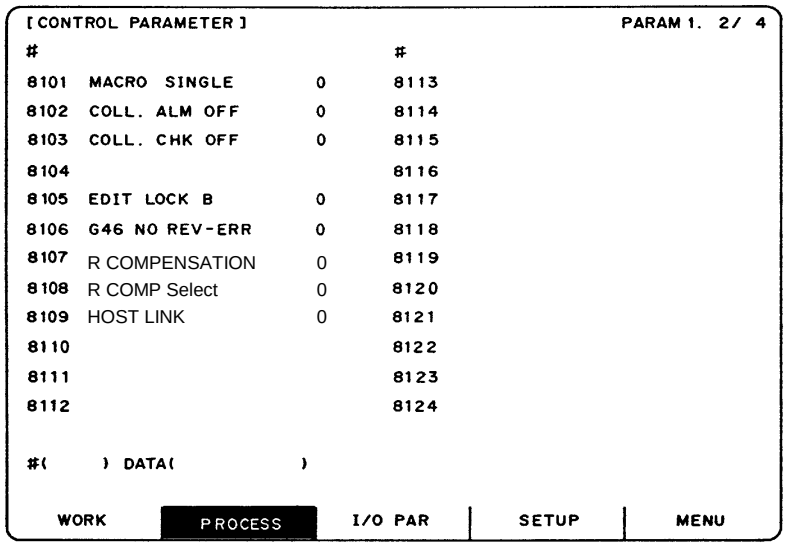
<FIXED C.> (Fixed cycle)

#	Name	Contents	Setting range (unit)
8012	G73n (For M system only)	Set the return amount for G73 (step cycle).	0 to 99999.999 (mm)
8013	G83n	Set the return amount for G83 (deep hole drilling cycle).	0 to 99999.999 (mm)
8014	CDZ-VALE (For L system only)	Set the cutting amount for the thread in G76 and 78 (thread cutting cycle).	0 to 127 (unit:0.1 lead)
8015	CDZ-ANGLE (For L system only)	Set the cutting angle for the thread in G76 and 78 (thread cutting cycle).	0 to 89 (°)
8016	G71 MINIMUM (For L system only)	Set the min. value for the final cutting amount in G71 and 72 (rough cutting cycle). If the final cutting amount is smaller than this value, the final cut will not be carried out.	0 to 99.999 (mm)
8017	DELTA-D (For L system only)	Set the change amount for the command cutting amount D in G71 and 72 (rough cutting cycle). Each cutting amount will be the value obtained by this value added to or subtracted from command D. Thus, the amount can be changed for each cut.	0 to 99.999 (mm)
8018	G84/G74n (M system dedicated)	Set the return amount "m" for the G84/G74 pecking tap cycle. **(Note)** The normal tap cycle will take place when "0" is set.	0 to 999.999 (mm)

<PRECISION> (High precision control)

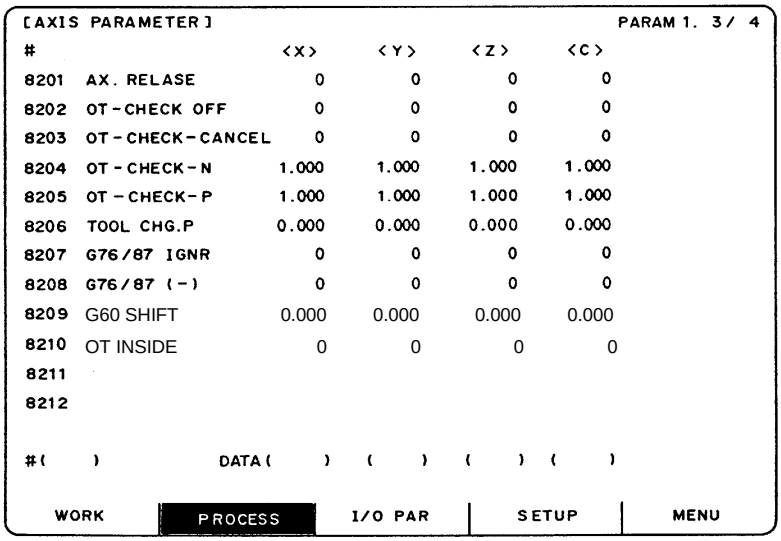
#	Item	Contents	Setup range (unit)
8019	R COMP	Set up a compensation factor for reducing a control error in the reduction of a corner roundness and arch radius. Indicate a maximum control error (mm) in parentheses. The larger the setup value, the smaller the theoretical error will be. However, since the speed at the corner goes down, the cycle time is extended.	0 to 99 (%)
8020	DCC. ANGLE	Set up 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-precision mode is larger than the set value, it is determined as a corner and the speed goes down to sharpen the edge.  If the set value is smaller than θ, the speed goes down to optimize the corner. **(Note)** If "0" is set, it will be handled as 5 degress. The standard setting value is "0".	0 to 30 (°) 0: The angle will be 5 degress.

2.4.2.2 Control Parameters



#	Items	Details	Setting range (unit)
8101	MACRO SINGLE	Select the control of the blocks where the user macro command continues. 0: Do not stop while macro block continues. 1: Stop every block during signal block operation.	0/1
8102	COLL. ALM OFF	Select the interference (bite) control to the workpiece from the tool diameter during tool radius compensation and nose R offset. 0: An alarm is output and operation stops when an interference is judged. 1: Changes the path to avoid interference.	0/1
8103	COLL. CHK OFF	Select the interference (bite) control to the workpiece from the tool diameter during tool radius compensation and nose R offset. 0: Performs interference check 1: Does not perform interference check	0/1
8105	EDIT LOCK B	Select the edit lock for program Nos. 8000 to 9999. 0: Program can be edited. 1: Editing of above program is prohibited.	0/1
8106	G46 NO REV-ERR (For L system only)	Select the control for compensation direction reversal in G46 (Nose R compensation). 0: An alarm occurs and the operation stops when the compensation direction reverses (G41 → G42, G42 → G41). 1: The compensation direction is maintained as before without an alarm occurring when the compensation direction reverses.	0/1
8107	R OMPENSATION	Not used.	0
8108	R COMP Select	Not used.	0
8109	HOST LINK	Not used.	0

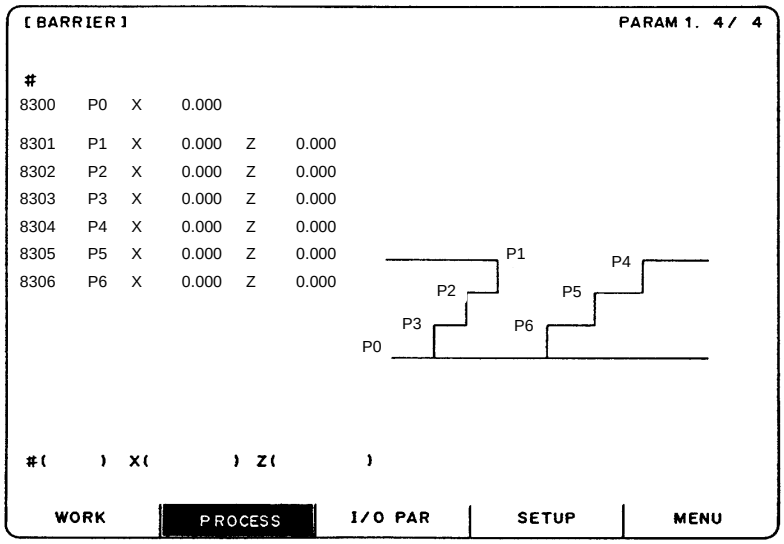
2.4.2.3 Axis Parameters



#	Items	Details	Setting range (unit)
8201	AX. RELEASE	Select the function to remove the control axis from the control target. 0: Control as normal 1: Remove from control target	0/1
8202	OT-CHECK OFF	Select the stored stroke limit II function set in #8204 and #8205. 0: Stored stroke limit II valid 1: Stored stroke limit II invalid	0/1
8203	OT-CHECK-CANCEL	When the simple absolute position method ("#2049 type" is 9), the stored stroke limit I, II (or IIB), IB will be invalid until the first reference point return is executed after the power is turned on. 0: Stored stroke limit valid (according to #8202) 1: Stored stroke limit invalid temporarily **(Note)** The setting of this parameter (#8203) affects all of the stored stroke limits.	0/1

#	Items	Details	Setting range (unit)
8204	OT-CHECK-N	This sets the coordinates of the (–) direction in the moveable 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.	-99999.999 to +99999.999 (mm)
8205	OT-CHECK-P	This sets the coordinates of the (+) direction in the moveable range of the stored stroke limit II or the upper limit coordinates of the prohibited range of stored stroke limit IIB.	-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.	-99999.999 to +99999.999 (mm)
8207	G76/87 IGNR (For M system only)	Select the shift operation at G76 (fine boring) and G87 (back boring)	0 : Shift effective 1 : No shift
8208	G76/87 (-) (For M system only)	Specifies the shift direction at G76 and G87.	0 : Shift to (+) direction 1 : Shift to (-) direction
8209	G60 SHIFT (For M system only)	Set the final positioning direction and distance per axis for the G60 (uni-directional positioning command).	-99999.999 to +99999.999 (mm)
8210	OT INSIDE	The stored stoke limit function to be set in #8204 and #8205 prevents the machine from moving to the inside or outside of the specified range. 0: Inhibits outside area. (stored stroke limit II selected.) 1: Inhibits inside area. (stored stroke limit IIB selected.)	0/1

2.4.2.4 Barrier Data



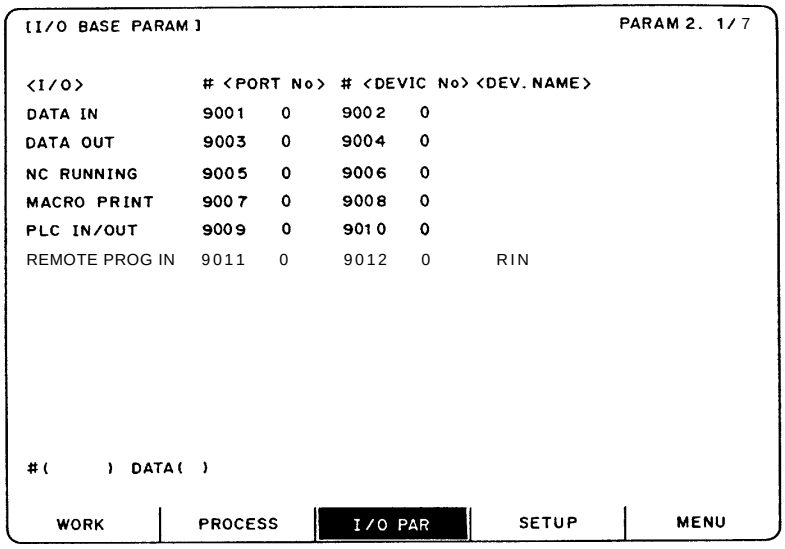
#	Items	Details	Setting range (unit)
8300	P0 (For L system only)	Set the reference X coordinate of the chuck and tailstock barrier. Set the workpiece center coordinate in the basic machine coordinate system. (Radius value)	-99999.999 to +99999.999 (mm)
8301 8302 8303 8304 8305 8306	P1 P2 P3 P4 P5 P6 (For L system only)	Set the region of the chuck and tailstock barrier. (Radius value) X axis: Set the coordinate value from the workpiece center. Z axis: Set with the coordinates in the basic machine coordinate system.	-99999.999 to +99999.999 (mm)

2.4.3 I/O Parameters

Pressing the menu key [I/O PARA] displays the I/O BASE PARAM screen.

There are basically two types of input/output parameters which must be set when inputting, outputting or referring to data, or when performing tape operation. One type is the parameters related to the input/output device. The baud rate, etc., is set according to each device. Up to five types of input/output devices can be registered. The other type of input/output parameters is the I/O base parameters which determine which device is connected to which channel per input/output application.

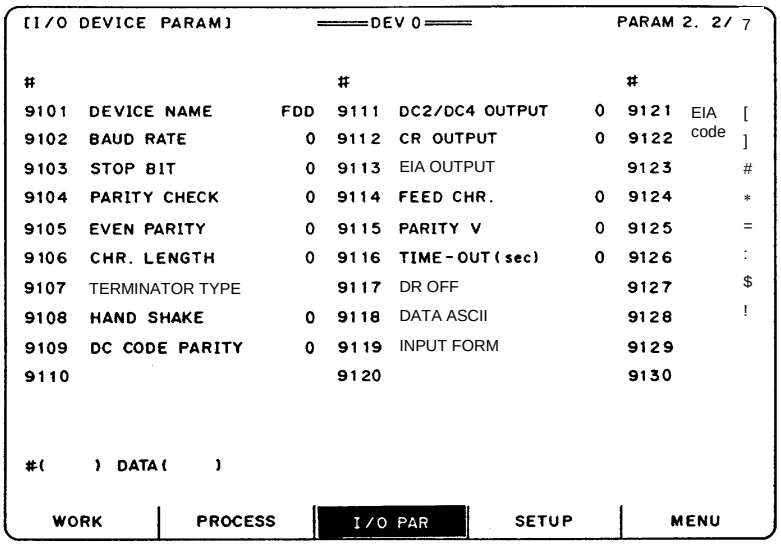
2.4.3.1 I/O Base Param



<I/O>	#	<PORT No.>	#	<DEV. No.> <DEV. NAME>
		Specify the board No. to which the serial input/output device is connected for each application. Only 0 is valid.		Set the input/output device No. for each application. The device Nos. are 0 to 4 and correspond to the input/output device parameters. The device name set in the input/ output device parameter is also displayed for identification.
DATA IN	9001	Specify the port for inputting the data such as machine program and parameters.	9002	Specify the No. of the device that inputs the data.
DATA OUT	9003	Specify the port for outputting the data such as machine program and parameters.	9004	Specify the No. of the device that outputs the data.
NC RUNNING	9005	Specify the input port for running with the tape mode.	9006	Specify the No. of the device to be run with the tape mode.
MACRO PRINT	9007	Specify the output port for the user macro DPRINT command.	9008	Specify the No. of the device for the DPRINT command.
PLC IN/OUT	9009	Specify the port for inputting/ outputting various data with PLC.	9010	Specify the No. of the device for the PLC input/output.
REMOTE PROG IN	9011	Not used.	9012	Not used.

2.4.3.2 I/O Device Param

Parameters for up to five types of input/output devices can be set in DEV <0> to <4>.



#	Items	Details
9101	DEVICE NAME	Set the device name corresponding to the device No. Set a simple name for quick identification. Use alphabet characters, numerals and symbols to set a name within 3 characters.
9102	BAUD RATE	Set the serial communication speed. 1: 9600bps 5: 600bps 2: 4800bps 6: 300bps 3: 2400bps 7: 150bps 4: 1200bps
9103	STOP BIT	Set the stop bit length used in the start-stop system. 1: 1 bit 2: 1.5 bits 3: 2 bits
9104	PARITY CHECK	Specify whether to add the parity check bit to the data during communication. 0: Parity bit not added 1: Parity bit added
9105	EVEN PARITY	Specify the odd or even parity when it is added to the data. 0: Odd parity 1: Even parity
9106	CHR. LENGTH	Set the length of the data bit 0: Data is 5 bits 1: 6 bits 2: 7 bits 3: 8 bits
9107	TERMINATOR TYPE	0: Read is ended at the EOR code. 1: Read is ended at the EOB code.
9108	HAND SHAKE	Specify the transmission control method. 1: RTS/CTS method (This method can be used only for S102.) 2: No procedure (No handshaking) 3: DC code method The method will be no procedure if a value other than the above is set.

#	Items	Details
9109	DC CODE PARITY	Specify the DC code when the DC code method is selected. 0: No parity to DC code (DC3=13H) 1: DC code with parity (DC3=93H)
9111	DC2/DC4 OUTPUT	Specify the DC code handling when outputting data to the output device. DC2 DC4 0: None None 1: Yes None 2: None Yes 3: Yes Yes
9112	CR OUTPUT	Specify whether to insert the <CR> code just before the EOB (L/F) code during output. 0: Do not add 1: Add
9113	EIA OUTPUT	0: Output with ISO code. 1: Output with EIA code.
9114	FEED CHR.	Specify the length of the tape feed to be output at the start and end of the data during tape output. The setting range is 0 to 999 characters.
9115	PARITY V	Specify whether to check the parity of the No. of characters in block during data input. 0: Do not perform parity V check 1: Perform parity V check The No. of characters is factory-set so that the check is valid at all times.
9116	TIME-OUT (sec)	Set the time out time to detect an interruption in communication. The setting range is 0 to 30 [s]. The time out is not checked when 0 is set, so the waiting time will be infinite. The screens in the setup/maintenance tool cannot be changed during the waiting time. Set this time to 10 as the standard.
9117	DR OFF	Specify whether to check the DR data during data input/output. 0: DR valid 1: DR invalid
9118	DATA ASCII	0: Output in ISO/EIA code (Depends on whether #9113, #9213, #9313, #9413, or #9513 EIA output parameter is set up) 1: Output in ASCII code
9119	INPUT FORM	Specify the method used during input (compare). 0: Standard input (Data is handled as significant information from first EOB) 1: The EOB input in succession after the first EOB in the data is skipped until data other than EOB is input.
9121 EIA CODE [9122] 9123 # 9124 * 9125 = 9126 : 9127 $ 9128 !		When outputting with the EIA codes, special ISO codes not included in EIA can be output with alternate codes. For each special code, designate a code (as a hexadecimal) that is not duplicated with existing EIA codes.

9201~	Set the same settings for device 1.
9301~	Set the same settings for device 2.
9401~	Set the same settings for device 3.
9501~	Set the same settings for device 4.

2.4.3.3 Computer Link Parameter

Computer link function is not used with this NC system.

[COMPUTER LINK PARAMETER] PARAM 2.7/7

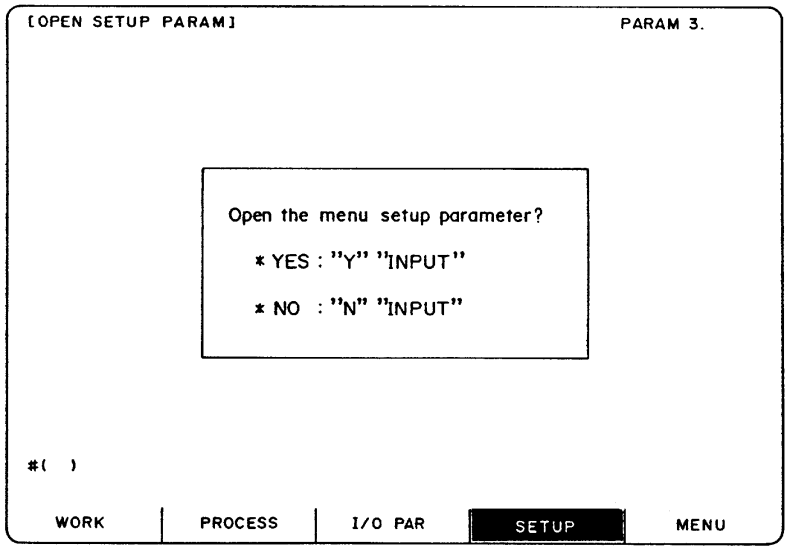
#		#		#	
9601 BAUD RATE	0	9611 LINK PARAM. 3	00	9621 DC1 OUT SIZE	0
9602 STOP BIT	0	9612 LINK PARAM. 4	00	9622 POLLING TIMER	0
9603 PARITY EFFECTIVE	0	9613 LINK PARAM. 5	00	9623 TRANS. WAIT TMR	0
9604 EVEN PARITY	0	9614 START CODE	0	9624 RETRY COUNTER	0
9605 CHR. LENGTH	0	9615 CTRL. CODE OUT	00	9625	
9606 HAND SHAKE	0	9616 CTRL. INTERVAL	0	9626	
9607 TIME-OUT SET	0	9617 WAIT TIME	0	9627	
9608 DATA CODE	0	9618 PACKET SIZE	0	9628	
9609 LINK PARAM. 1	00	9619 BUFFER SIZE	0	9629	
9610 LINK PARAM. 2	00	9620 START SIZE	0	9630	

( ) DATA ( )

2.4.4 Setup Parameters

Pressing the menu key SETUP 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.



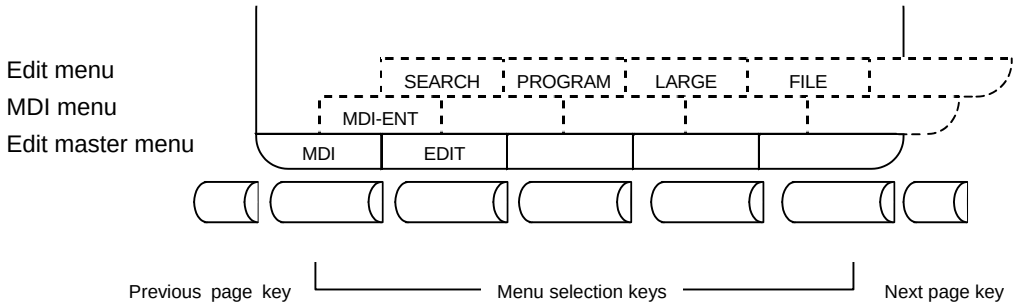
1) Select the setup parameter.
   Key-in "Y" in # ( ), and then press INPUT.
   The normally hidden setup parameter menu will display when the menu changes over.
   The required menu can be selected to display and set the setup parameters.
2) Cancel the setup parameter selection.
   Key-in "N" in # (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

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

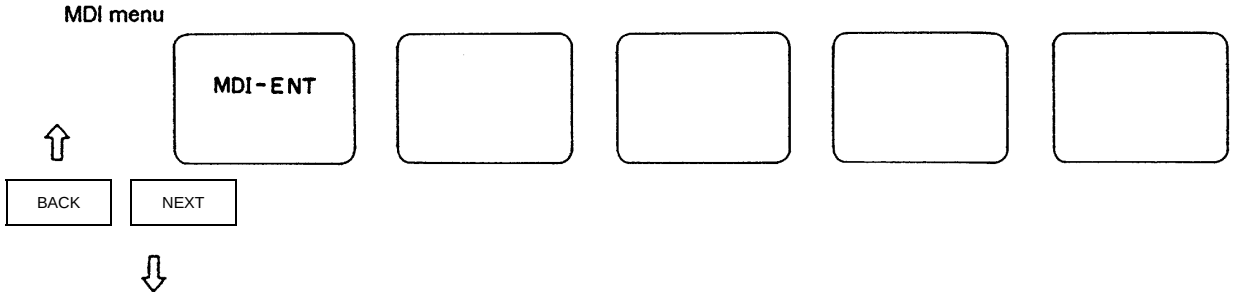
Refer to “C6/C64/C64T Parameter Manual” for details on the setup parameters.
Always turn the power OFF after selecting the setup parameters.

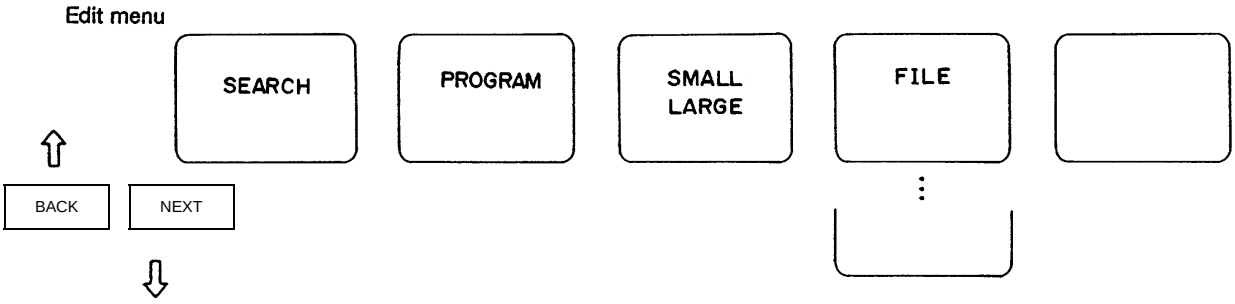
2.5 Program

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



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





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

When the [EDIT MDI] key is first pressed after the power is turned on, the MDI screen appears.

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 [SEARCH] or [MAKE] operation. To edit a program already registered in memory, perform [SEARCH] operation. To register a new program in memory, perform [MAKE] 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

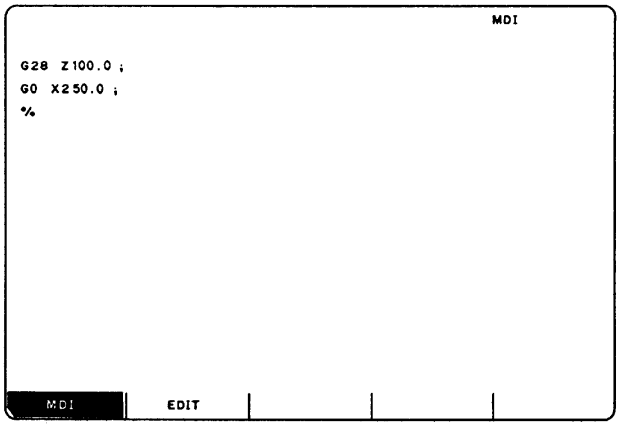
If a character string that matches a macro operator exists in the machining program (including a comment statement), it is automatically converted into the corresponding intermediate code during editing. This may cause a string different from that entered to be displayed during editing.

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

2.5.2 Menu Function

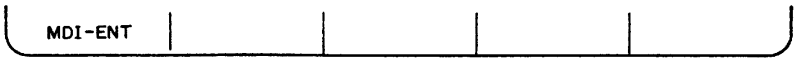
2.5.2.1 MDI Screen Menu Function

(1) Menu when [MDI] is selected



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     ;
```

```
N1G28X0Y0Z0;N2G92X0Y0Z0;N3G00X–100. Y–
100. ;N4G01X–300. F2000;N5Y–300. ;N6X–100
. ;N7Y–100. ;N8M02;
                                EDITING
```

(2) Press the [INPUT] key.

```
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 ;
%
                        MDI SETTING COMPLET
```

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.

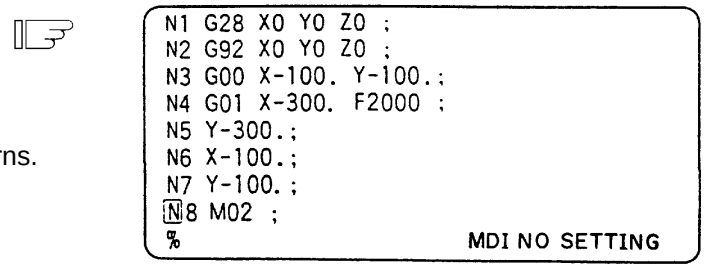
(Note 3) Check the MDI SETTING COMPLET message before starting MDI operation. If the EDITING or MDI NO SETTING message is displayed, MDI operation cannot be started. If the [INPUT] key is pressed at the time, the MDI SETTING COMPLET message is displayed.

(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.



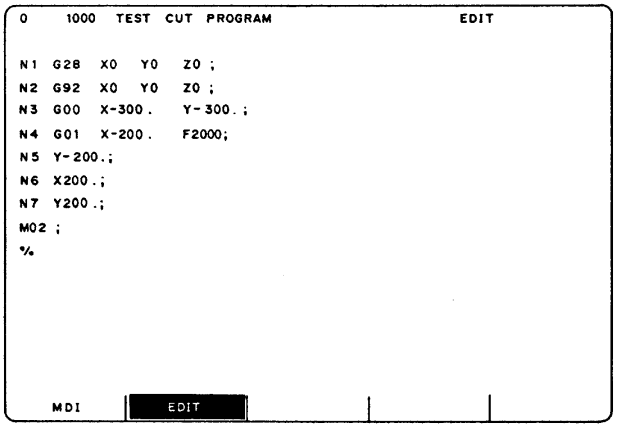
1) The "MDI NO SETTING" status returns.

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.

2.5.2.2 EDIT Screen Menu Function

(1) Menu when [EDIT] is selected



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 operation menu

SEARCH | MAKE | LARGE | FILE | or SEARCH | MAKE | SMALL | FILE |

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.

(3) Edit program call

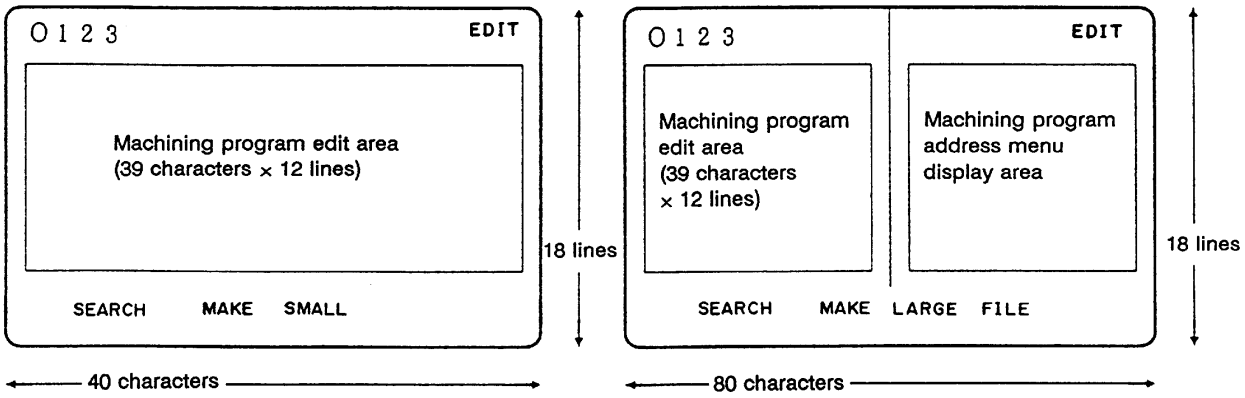
To edit a program on the EDIT screen, first press the extension operation menu key [SEARCH] or [MAKE]. To edit an already stored program in memory, press [SEARCH]. To store a new program in memory, press [MAKE]. For details, see 5.5.

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.

(4) Large-size mode/small-size mode

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



Large-size mode **Small-size mode**

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 keys [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.

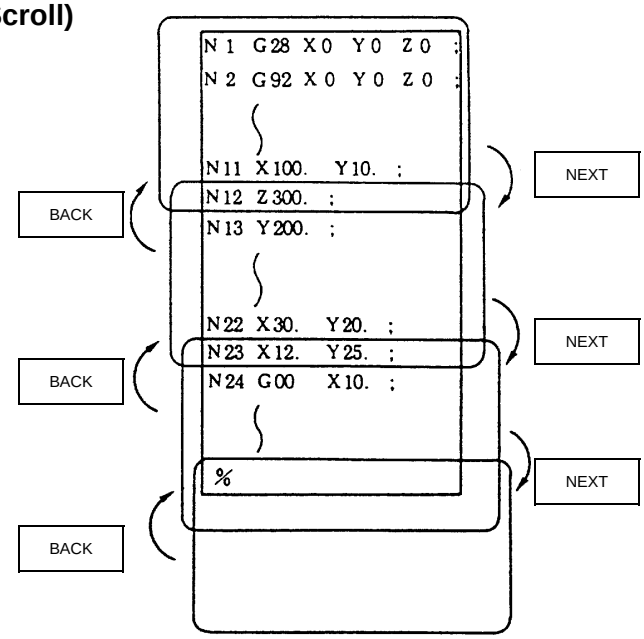
2.5.3 Program Edit Operation

Program edit operation is common to the EDIT and MDI screens.

2.5.3.1 Data Display Update (One Screen Scroll)

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.

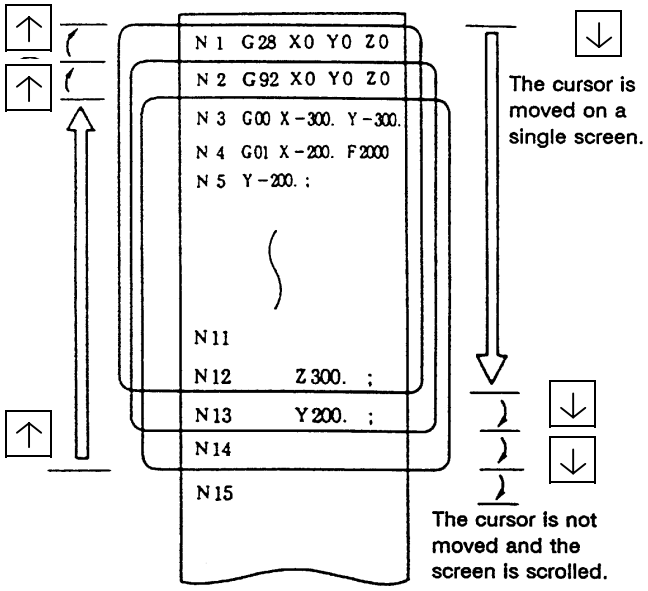
```
N1 G28 X0 Y0 Z0;
N2 G92 X0 Y0 Z0;
N3 G00 X-300. Y-400.;
N4 G01 X-200. F2000;
N5 Y-200.;
N6 X200.;
   :
N11 X100. Y10.;
                    EDITING
```

```
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)
```

2.5.3.2 Data Display Update (One Line Scroll)

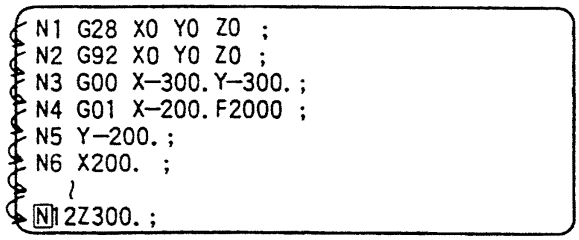
Data display on the screen can be updated in line units by using the ↑ or ↓ key.

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.



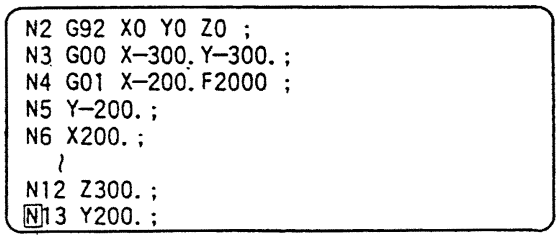
The cursor is moved downward each time the ↓ key is pressed.

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.



2.5.3.3 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 Y200. ; block to

↓

G03 Y200. J100. ;

```
                                    EDIT (REP.)
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.;
N7 Y200.;
M02;
%
```

(1) Move the cursor to the data to be replaced. ☞

```
                                    EDIT (REP.)
N1 G28 X0 Y0 Z0;
        :
N7  Y 200.;
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.

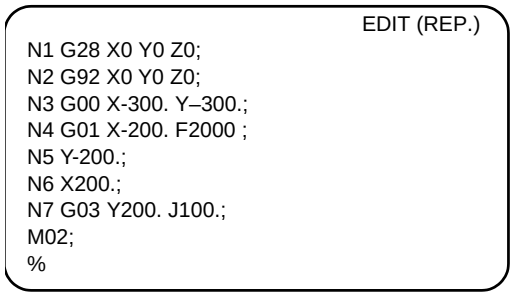
(3) After completion of correction, press the INPUT key. ☞

```
N7 G03 Y200. J100.;
M02;
%
```

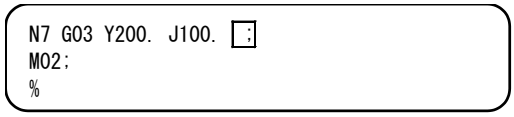
1) The new data is written into memory.
2) The new data is also displayed with each work being both preceded and followed by space code.
3) The cursor is moved to the top of the block on the screen.
4) When the data has been written into memory, the EDITING message disappears.

2.5.3.4 Data Insertion (INS)

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

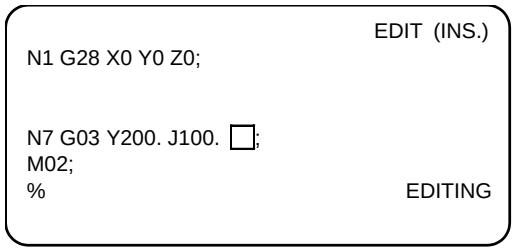


(1) Move the cursor to the character following the position in which the data is to be inserted.



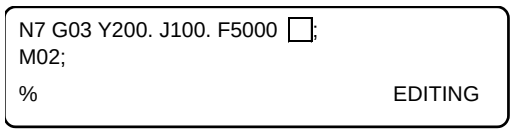
(2) Press the INS key.

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) 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;
%

2.5.3.5 Deletion of One Character (DEL)

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

```
                                    EDIT  (REP.)
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.;
N7 G03 Y200. J100. F5000;
M02;
%
```

(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.

2.5.3.6 Deletion of One Block (CAN C.B.)

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

```
                                        EDIT  (REP.)
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.;
N7 G03 Y200. J100. F500;
M02;
%
```

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

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

(2) Press the CAN C.B. key. ☞

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

M02;
%                                       EDITING
```

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

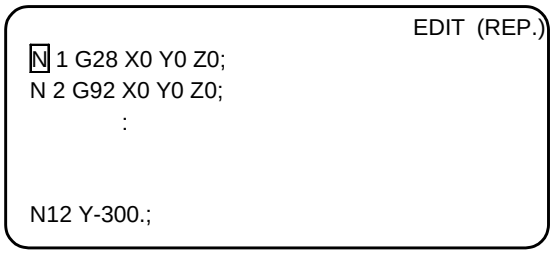
(3) After completion of correction, press the INPUT key. ☞

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

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.

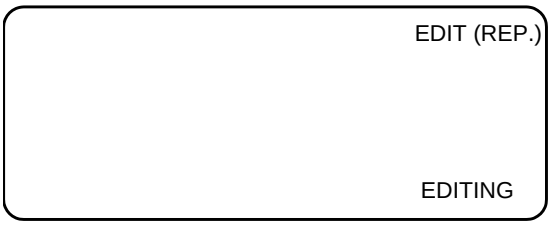
2.5.3.7 Deletion of Data on One Screen

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 CAN C.B. 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.



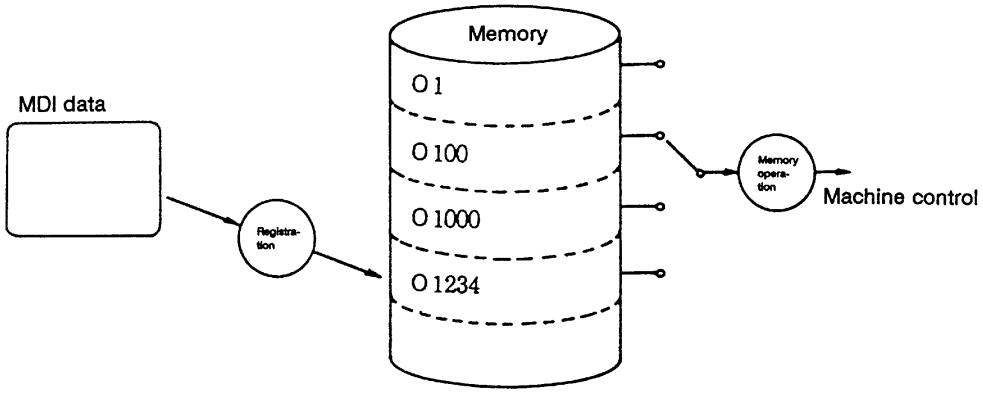
(2) Press the INPUT key.

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.

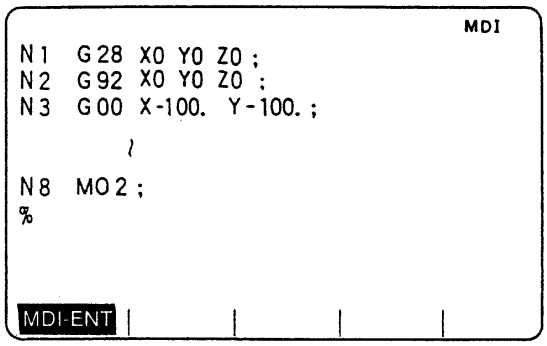
2.5.4 MDI Screen Extension Operation

2.5.4.1 MDI Data Registration in Memory (MDI-ENT)

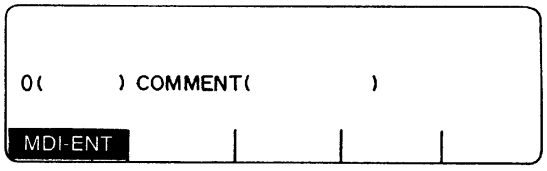
Data set on the MDI screen can be registered in memory. Comments can be added to indicate the contents of the program to be registered.



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

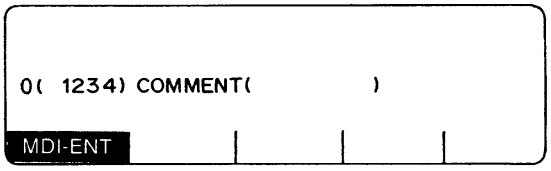


(1) Press the menu key MDI-ENT.

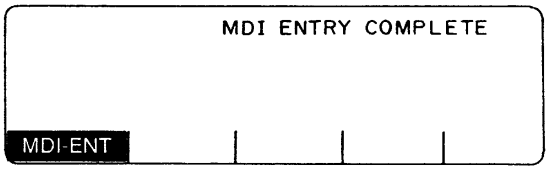


1) The setting area for MDI-ENT is displayed.

(2) Set the registered program number. A comment can also be set at the same time.
(Example)
O (1 2 3 4) 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 (SPASE) can be written in it. But, the space is ignored after registration for efficient use of memory.

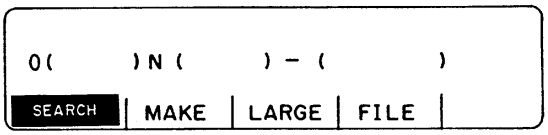
2.5.5 Edit Screen Extension Operation

2.5.5.1 Edit Data Call (SEARCH)

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.

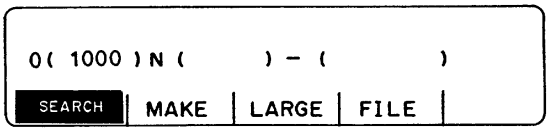
(1) Search for the program head
In the setting field, specify the program number of the program to be called.
The operating procedure is as follows:

(1) Press the menu key SEARCH.

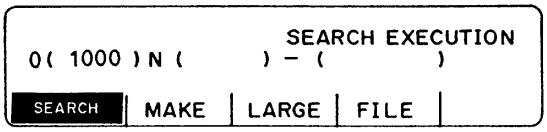


1) The setting area for SEARCH is displayed.

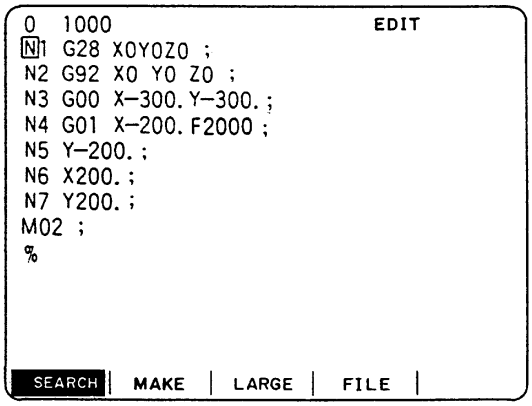
(2) Set the called program number.
(Example)
O (1 0 0 0) N ( ) - ( )



(3) Press the INPUT key.



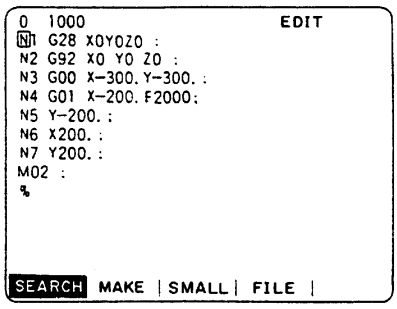
1) The message SEARCH EXECUTION is displayed during searching.
2) The specified program is displayed, beginning with top of the program.
3) The cursor is displayed at the top of the screen.
4) At normal completion of program head search, display of the setting area disappears and SEARCH menu display is restored to normal mode from reverse mode.



(2) Character string search

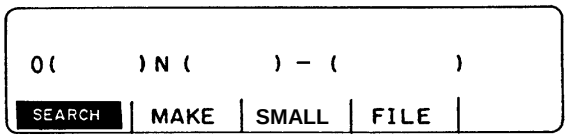
The character string search is useful particularly to search the word data to be corrected.

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.



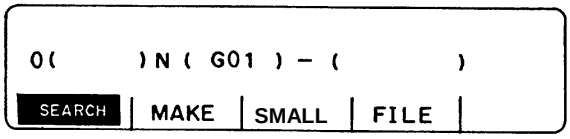
The operation procedure is described below:

(1) Press the menu key [SEARCH].

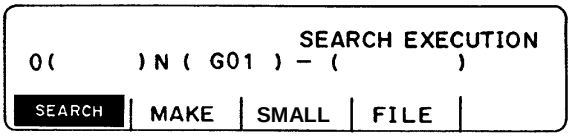


1) The setting area for SEARCH is displayed.

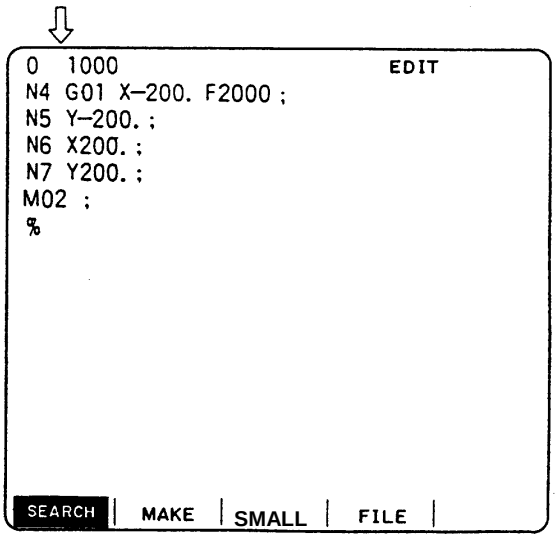
(2) Set the called program number and character string.
(Example)
O ( ) N (G 0 1) - ( )



(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 search, display of the setting area disappears and SEARCH menu display is restored to normal mode from reverse mode.



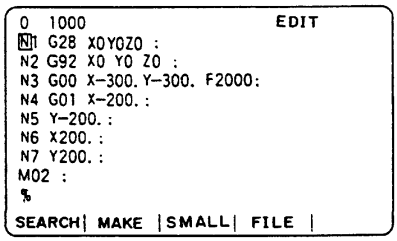
(Note 1) When a given character string is not found, a "NO CHARACTERS" message is displayed.
(Note 2) A string of up to 11 characters may be specified.
(Note 3) The specified character string is searched and identified in the specified number of character strings regardless of the preceding and subsequent 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.

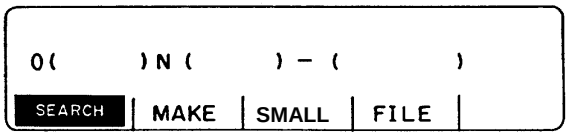
(3) Sequence number, block number search

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.



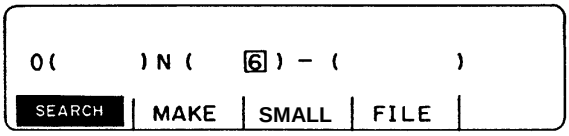
The operation procedure is described below.

(1) Press the menu key [SEARCH].

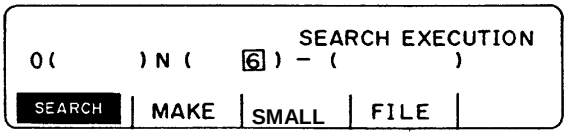


1) The setting area for SEARCH is displayed.

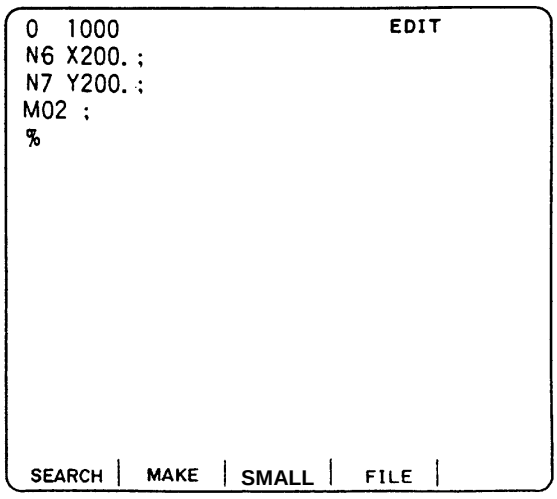
(2) Set the called program number, sequence number, and block number.
(Example)
O ( ) N (6) - ( )



(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.

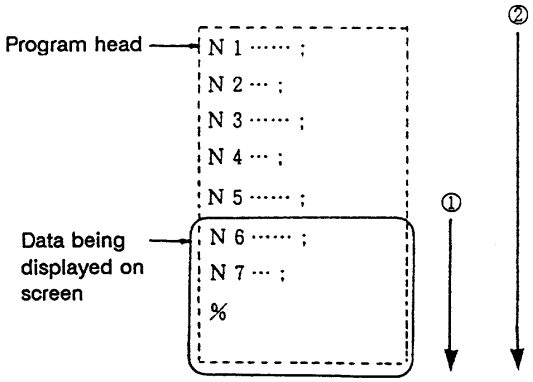


(Note 1) When a given N number is not found, an "NB NOT FOUND" message is displayed.
(Note 2) If a given program number is not found, a "PROG NOT FOUND" message is displayed.
(Note 3) The sequence number can be specified in a maximum of five digits.

(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)



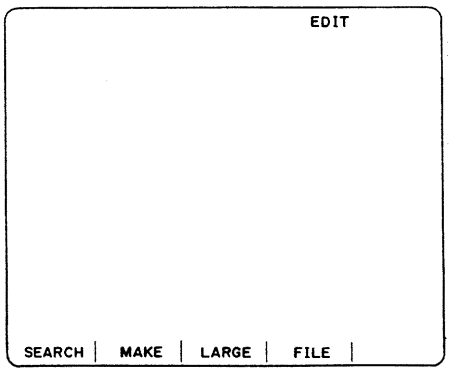
(Example 1) For search for N4:
(1) First search ... Error "NB NOT FOUND"
(2) Second search ... N4 can be found.

(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"

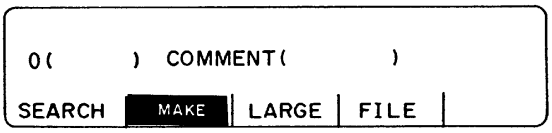
2.5.5.2 New Program Registration and Preparation

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

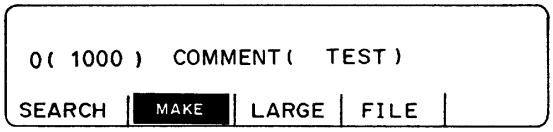


(1) Press the menu key [MAKE].

1) The setting area for PROGRAM is displayed.

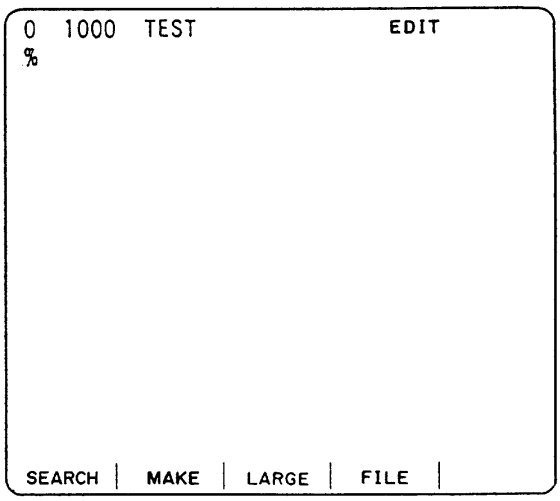


(2) Set the new registered program number. A comment can also be set at the same time if necessary.
(Example)
O ([1][0][0][0])
COMMENT ([T][E][S][T])



(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.



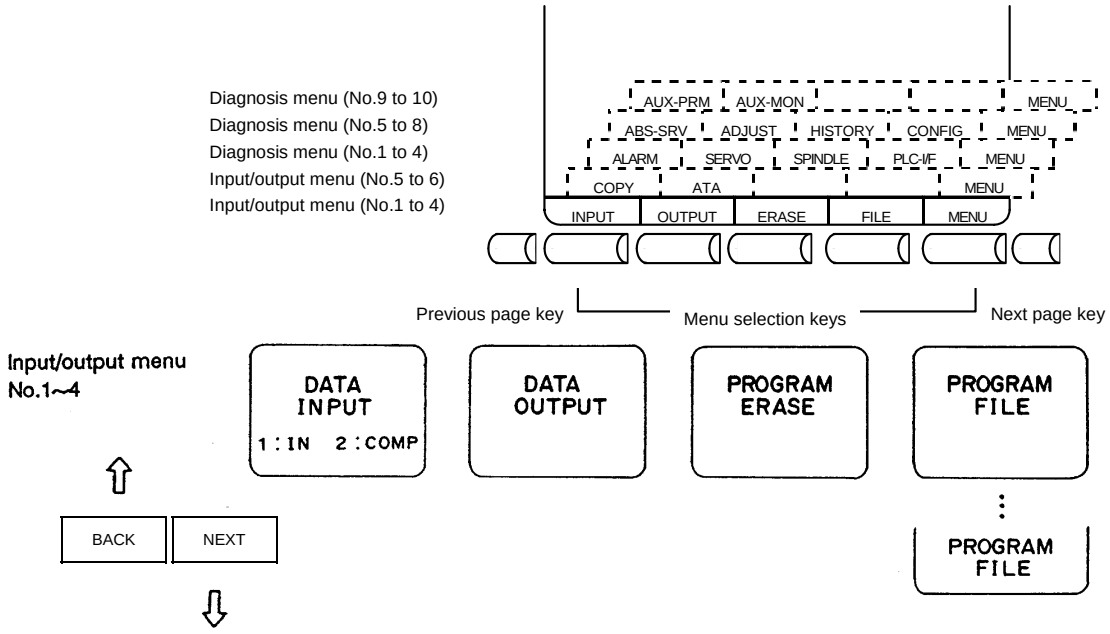
(4) Enter the work program in sequence by using the keys. Key operation is the same as normal program edit operation.

(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 ([SPASE]) can be written in it. But, the space is ignored after registration for efficient use of memory.

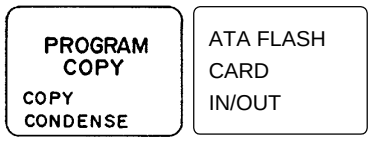
2.6 Data In/Out

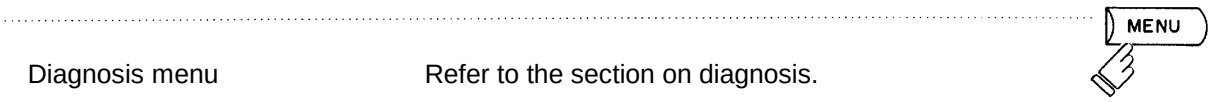
When the function selection key [MONITOR] is pressed, the following menu is displayed:



(Note) Use [I/O PARA] of [TOOL PARAM] to define the data input/output parameter.

Input/output menu No.5





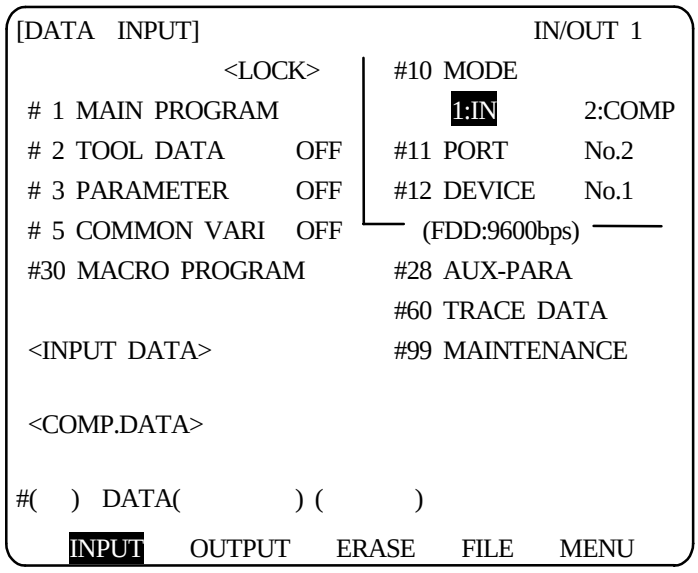
Diagnosis menu — Refer to the section on diagnosis.

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

2.6.1 Data Input

Pressing the menu key [INPUT] displays the DATA INPUT screen.

The DATA INPUT screen allows the operator to input user-created machining programs (main program and subprogram), tool offsets, parameters and common variables.



#	Item	Explanation
1 2 3 5 28 30 60	MAIN PROGRAM (Note 1) TOOL DATA PARAMETER COMMON VARI AUX-PARA MACRO PROGRAM TRACE DATA (Note 2)	The types of data that can be input are indicated. (1) Used when inputting the machining program. (2) Used when inputting the tool data. (3) Used when inputting the parameter data. (4) Used to input a common variable. (5) Used to input an auxiliary axis parameter data. (6) Used to input a macro program.
10	MODE 1: IN 2: COMP	The operation mode on the DATA INPUT screen is changed between IN (input) and COMP (comparison). After power is turned on, IN mode is initiated. Since indication in active mode is highlighted, make sure that correct mode is selected before input or comparison operation.
11 12	PORT NO. DEVICE NO.	The I/O port number and device number required to input data are set. If they are already set on the I/O BASE PARAM screen, the setup values are displayed. They may be changed on either screen.

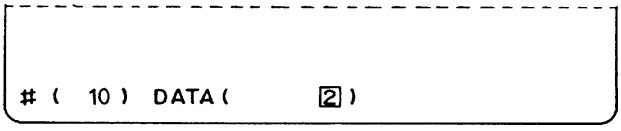
(Note 1) To input a fixed cycle program, set the parameter.
Refer to the Appendix "Registering and Editing Fixed Cycle Programs".
(Note 2) The history data (#60 TRACE DATA) cannot be input.
(Note 3) To input the parameters related to MELSECNET/10, set as “#(99) DATA (ALL2)”.

2.6.1.1 Change of Input and Comparison

To perform data input operation, select the IN mode; to perform data comparison operation, select the COMP mode. Before performing input or comparison operation, check the MODE display to ensure that the appropriate mode is set.
To change the mode between input and comparison, perform the following:
For example, if the IN mode is selected, "IN" is highlighted.

(Example) Change to the comparison mode.

(1 0)
DATA (2)



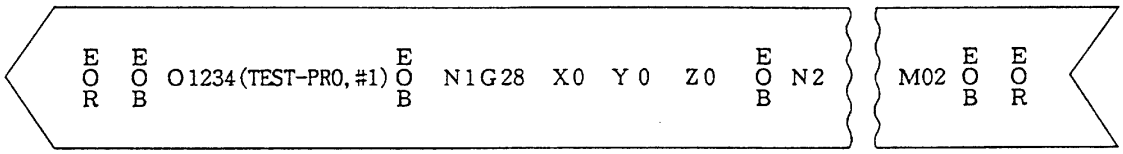
Press the INPUT key.

1) A change is made to the comparison mode and the word COMP is highlighted.
2) To change to the input mode, set 10 in # ( ) and 1 in DATA ( ), then press the INPUT key.

2.6.1.2 Machining Program Input

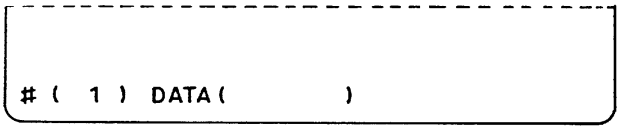
To input a user-prepared machining program, perform the following:

(1) To input the program stored in the external memory onto the CNC without changing its number. The program can be input simply by specifying machining program data type #1. Even if the program number is specified, the number on the external memory takes precedence over that number.

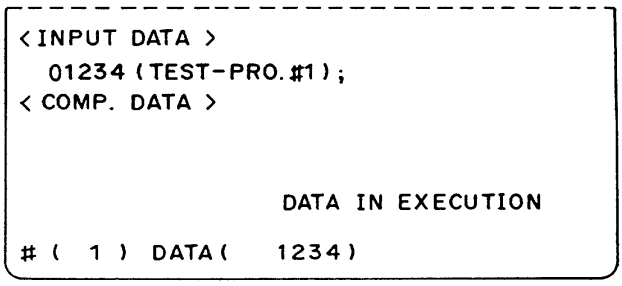


Program number and comment　　Program main unit

Set the data type.
(1) DATA ( )

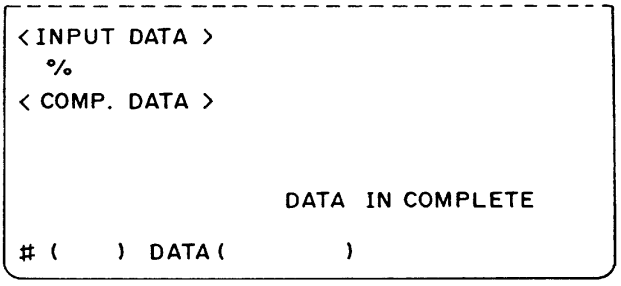


Press the [INPUT] key.



1) Data input is started.
The program number being entered is displayed in the setting area.

2) When normal data input is executed to the end and the EOR code is read, data input is terminated.

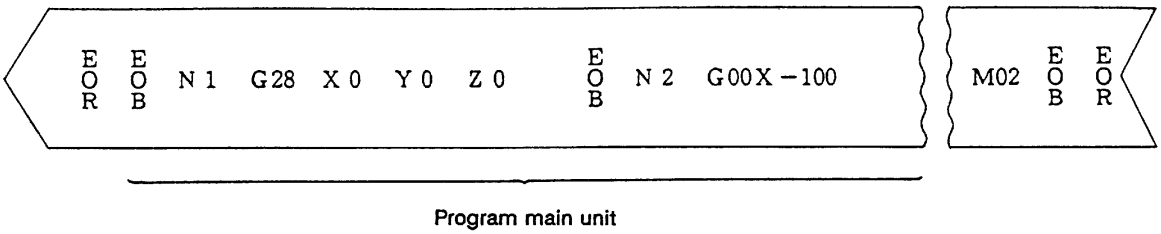


CAUTION

" ; " "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)".

To prevent influence from data omission and data transformation in the communication circuit, always verify the data after inputting and outputting machining programs.

(2) When program number does not exist on the external memory
Specify machining program data type #1 and the program number to be registered.



Program main unit

Set the data type and program number.
(Example)
To register the program with O1000
(1) DATA (1000)

☞

```
# ( 1 ) DATA (    1000 )
```

Press the [INPUT] key.

☞

```
<INPUT DATA >
  N1 G28 X0 Y0 Z0 ;
< COMP. DATA >


                    DATA IN EXECUTION
# ( 1 ) DATA (    1000 )
```

1) Data input is started.
2) The program is registered in memory with the specified program number.

⇩

3) When normal data input is executed to the end and the EOR code is read, data input is terminated.

```
<INPUT DATA >
  %
< COMP. DATA >


                    DATA IN COMPLETE
# (    ) DATA (        )
```

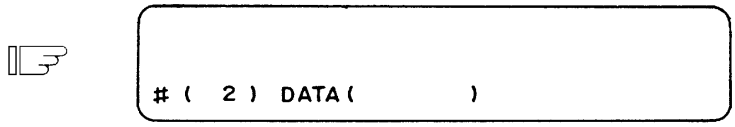
2.6.1.3 Inputting Tool Offset Data

Data which is output by tool offset data output operation can be input.
Data which is created in the same format as output data can be input as tool offset data and workpiece coordinate offset data.

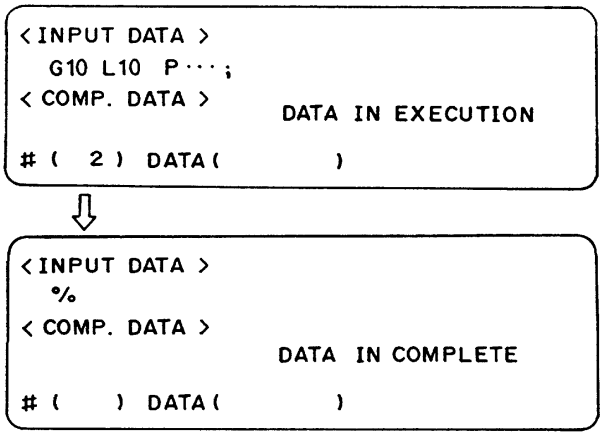
(Note) Tool offset data input operation cannot be made during automatic operation.

(1) Inputting tool offset data

Set the data type to tool data.
(2) DATA ( )



Press the INPUT key.



1) Data input is started. The contents of input data and message "DATA IN EXECUTION" are displayed.
2) When all data to the end has been input and the end code % (EOR) is read, data input ends with message "DATA IN COMPLETE" displayed.

(2) When an error occurs during offset input:

If an error occurs during offset input, the error number and error message will be displayed on the screen. At this time, input operation stops. (E02, E25, E71, or E86 error)

In this case, data input can resume by repressing the INPUT key while the input screen is being displayed. The data input can resume, beginning with the block next to the erroneous block, which is not input in this case.

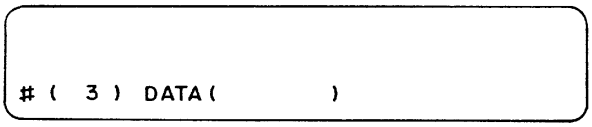
2.6.1.4 Inputting Parameter Data

Parameter data which has been output by parameter output operation can be input.
The input parameter may go effective immediately after it is input or after the power is once turned off/on. (This is the same as when setting from the screen.) After data is input, turn off/on the power.

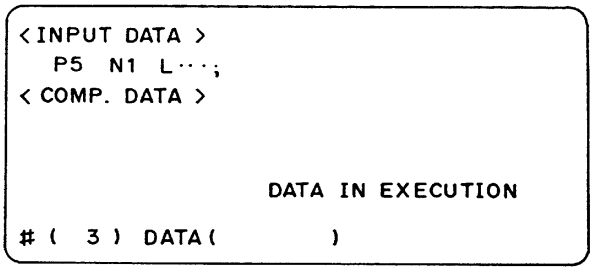
(Note) Parameter data input operation cannot be made during automatic operation.

(1) Inputting parameter data

Set the data type to parameter.
(3) DATA ( )

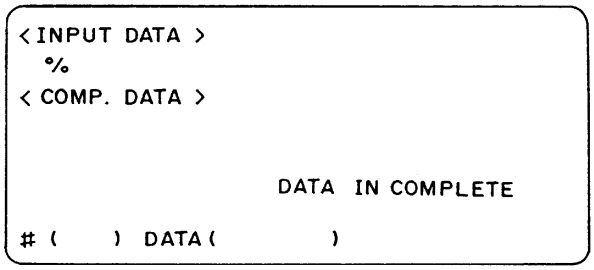


Press the INPUT key.



1) The parameter data input is started. The contents of input data and message "DATA IN EXECUTION" are displayed.

2) When writing is completed, the message "DATA IN COMPLETE" will display.



Turn the power OFF and ON once.

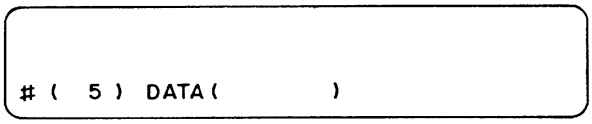
2.6.1.5 Inputting Common Variables

Common variable data that has been output by common variable output operation can be input.

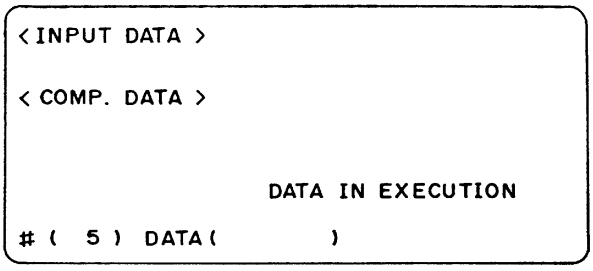
(Note) Common variable data input operation cannot be performed during automatic operation.

(1) Inputting common variable data

Set the data type to common variable.
(5) DATA ( )

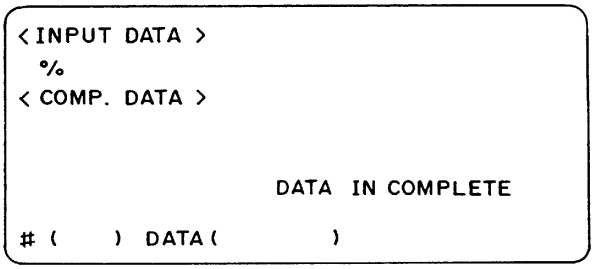


Press the INPUT key.



1) Data input is started.
The contents of input data and message "DATA IN EXECUTION" are displayed.

2) When all data through the end has been input and the end code % (EOR) is read, data input ends with message "DATA IN COMPLETE" displayed.

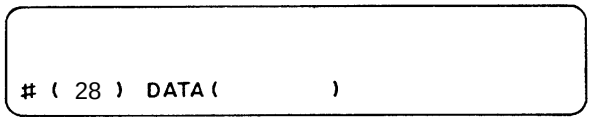


2.6.1.6 Inputting Auxiliary Axis Parameter Data

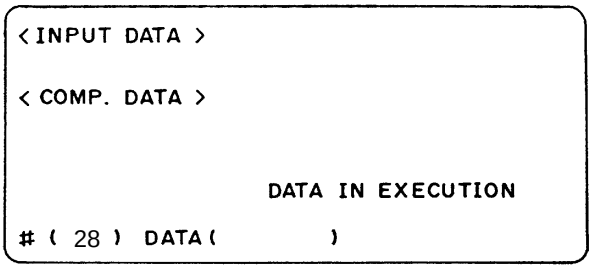
Auxiliary axis parameter data that has been output by auxiliary axis parameter output operation can be input.

(1) Inputting auxiliary axis parameter data

Set the data type to auxiliary axis parameter.
(2 8) DATA ( )

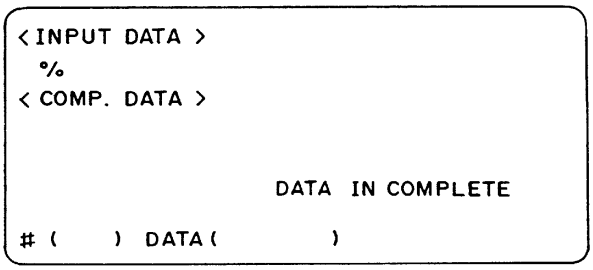


Press the INPUT key.



1) Data input is started.
The contents of input data and message "DATA IN EXECUTION" are displayed.

2) When all data through the end has been input and the end code % (EOR) is read, data input ends with message "DATA IN COMPLETE" displayed.



(Note 1) If MR-J2-CT is not connected, "E01 SETTING ERROR" will occur and input will not be carried out.

(Note 2) Whether to input the auto-tuning parameters depends on the “#7 ATU” parameter settings in the input data and NC data.

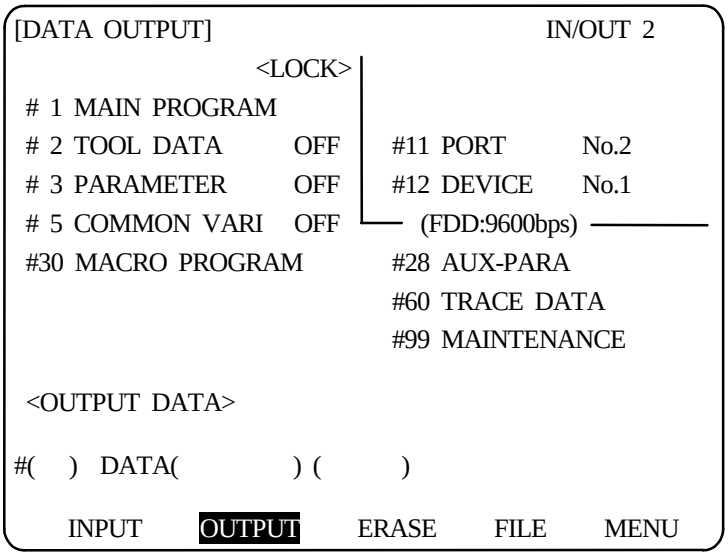
#7 ATU in input data	#7 ATU set in NC	Auto-tuning parameter input
Auto-tuned (0 or 1)	Auto-tuned (0 or 1)	Not input
Not auto-tuned (2)	Auto-tuned (0 or 1)	Input (Note 3)
Auto-tuned (0 or 1)	Not auto-tuned (2)	Input (Note 3)
Not auto-tuned (2)	Not auto-tuned (2)	Input (Note 3)

(Note 3) Which parameters can be input depends on the setting of “#7 ATU” parameter.
Refer to "2.6.2.7 MR-J2-CT Parameters and N No. Correspondence Table" for details on auto-tuning target parameters.

2.6.2 Data Output

Pressing the menu key [OUTPUT] displays the DATA OUTPUT screen.

The DATA OUTPUT screen allows the operator to output user-created machining programs (main program and subprogram), tool offset data, parameters, common variables and history data that have been stored in memory.



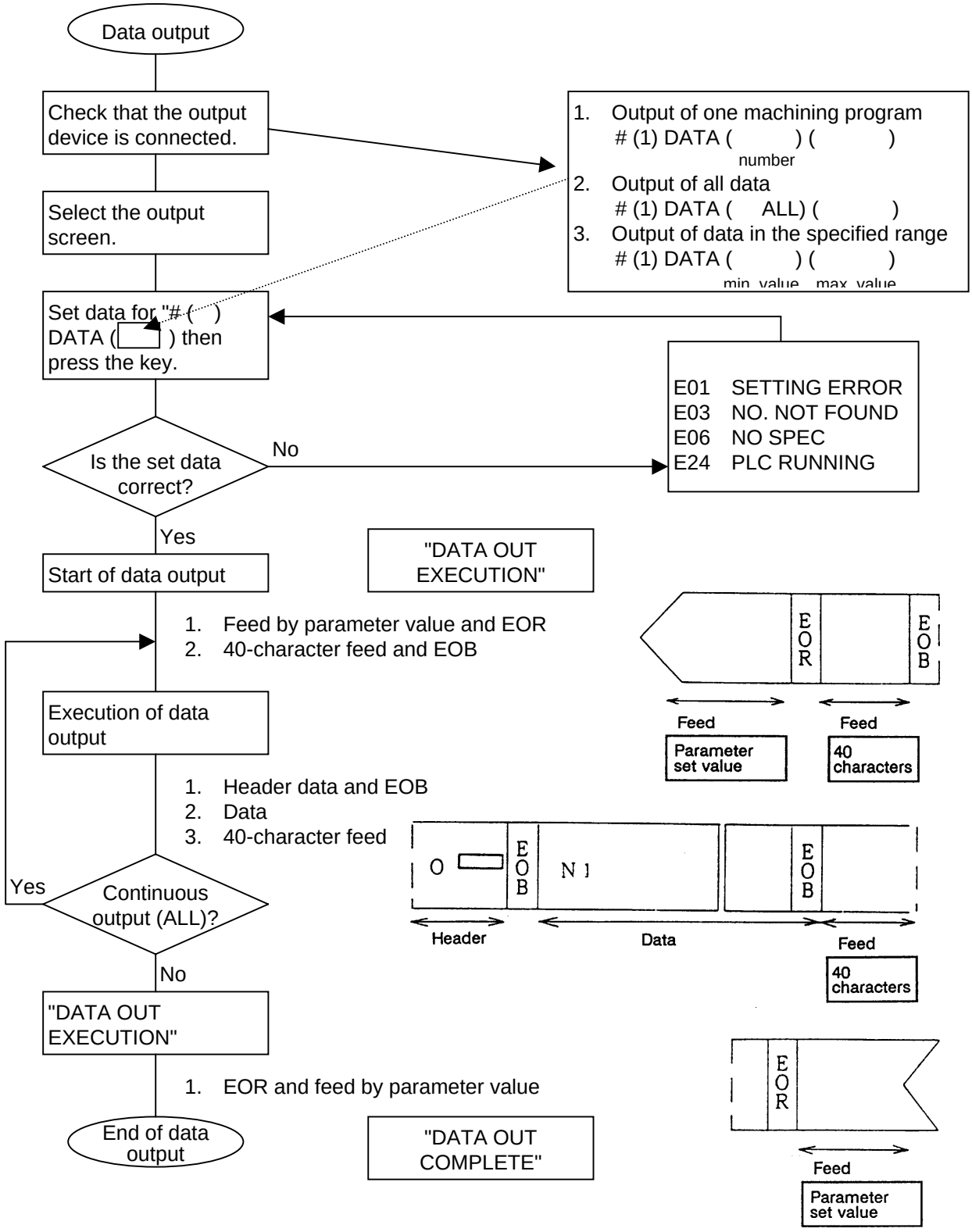
#	Item	Explanation
1 2 3 5 28 30 60	MAIN PROGRAM (Note 1) TOOL DATA PARAMETER COMMON VARI AUX-PARA MACRO PROGRAM TRACE DATA	The types of data that can be output are as follows. (1) Used to output a machining program. (2) Used to output tool data. (3) Used to output parameter data. (4) Used to output a common variable. (6) Used to output an auxiliary axis parameter data. (6) Used to output a macro program. (7) Used to output history data.
11 12	PORT NO. DEVICE NO.	The I/O port number and device number required to output data are set. If they are already set on the I/O BASE PARAM screen, the setup values are displayed on the DATA OUTPUT screen. They may be changed on either screen.

(Note 1) To output a fixed cycle program, set the parameter.
See Appendix 2 "Registering and Editing Fixed Cycle Programs".

(Note 2) If the data protection, edit lock B, or edit lock C condition is set, data may not be output. For the details, see the descriptions in "Data protection" and "Edit lock".

Output inhibit condition / Output method	Data protection key ON Machining programs, tool data, parameters	Edit lock B ON Machining programs 8000 to 9999	Edit lock C ON Machining programs 9000 to 9999
Specifying individual machining programs	No data is output.	Machining programs O8000 to 9999 are not output.	Machining programs O9000 to 9999 are not output.
Specifying ALL Specifying a range	No data is output.	Machining programs other than O8000 to 9999 are output.	Machining programs other than O9000 to 9999 are output.

Operation procedure for outputting data

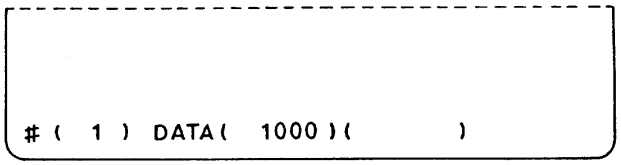


2.6.2.1 Machining Program Output

To output user-prepared machining programs, perform the following:

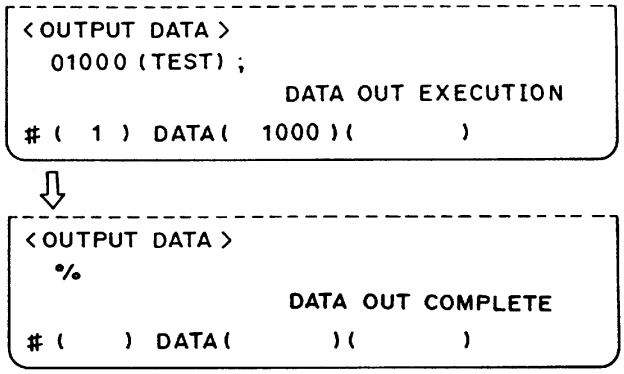
(1) When only one machining program is output
Specify machining program data type #1 and the number of the program to be output.

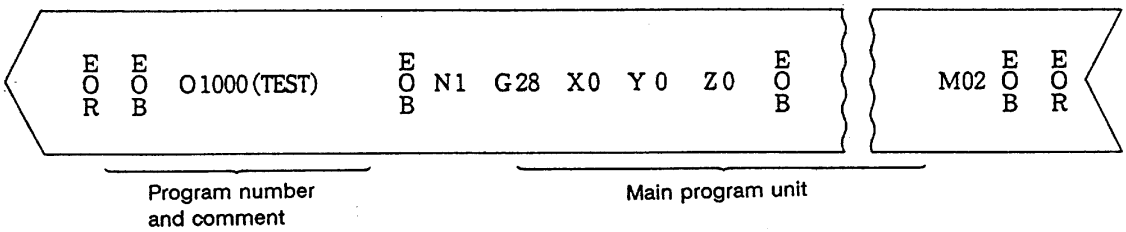
Set the data type and program number.
(Example)
To output program O1000
(1) DATA ( )



Press the INPUT key.

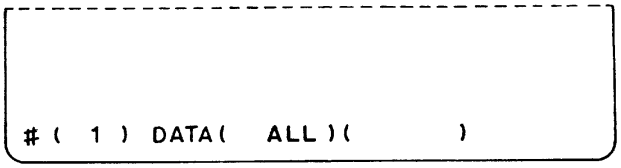
1) Data output is started.





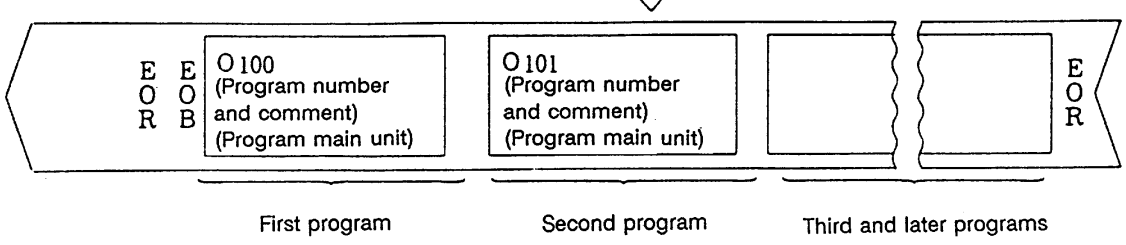
(2) When all machining programs are output
To output all machining programs registered in memory in batch, specify machining program data type #1 and ALL in DATA ( ).

Set 1 in # ( ) and ALL in DATA ( ).
(1) DATA (ALL)



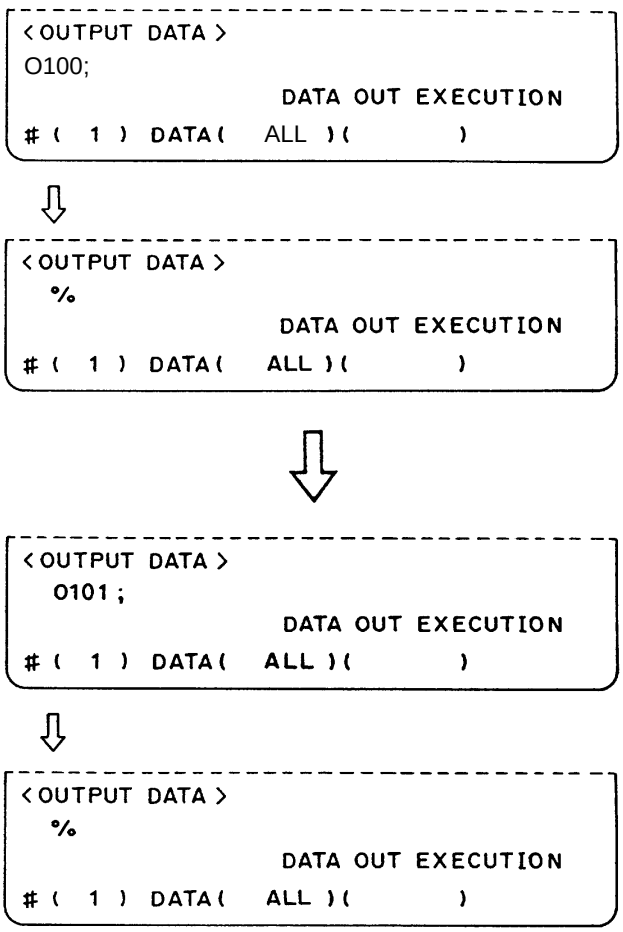
Press the INPUT key.

All programs are output in the program number ascending order.



(Note 1) When all data of one machining program is output, % is displayed. Note that % is not displayed each time individual data items are output.

When output of the first program is completed, % is displayed before indicating the next program.



Also for each of the second and succeeding programs, % is displayed each time one complete program is output.

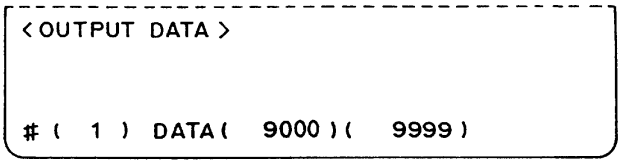
After all the specified machining programs are output, EOR is output. EOR is not output for individual program output.

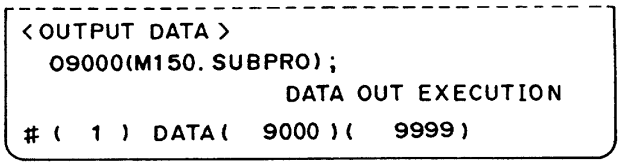
⚠ CAUTION

To prevent influence from data omission and data transformation in the communication circuit, always verify the data after inputting and outputting machining programs.

(3) When the machining programs in the specified range are to be output
A group of programs can be output by specifying a range of program numbers. To specify the range, set the largest and smallest numbers of the machining programs to be output in the data setting area. The machining programs in the specified range are output sequentially in order of their program numbers.

Specify the data type, and then the smallest and largest program numbers.
(Example)
To output program numbers O9000 to O9999, specify:
(1) DATA (9000)
(9999)

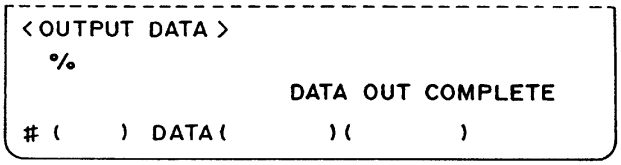




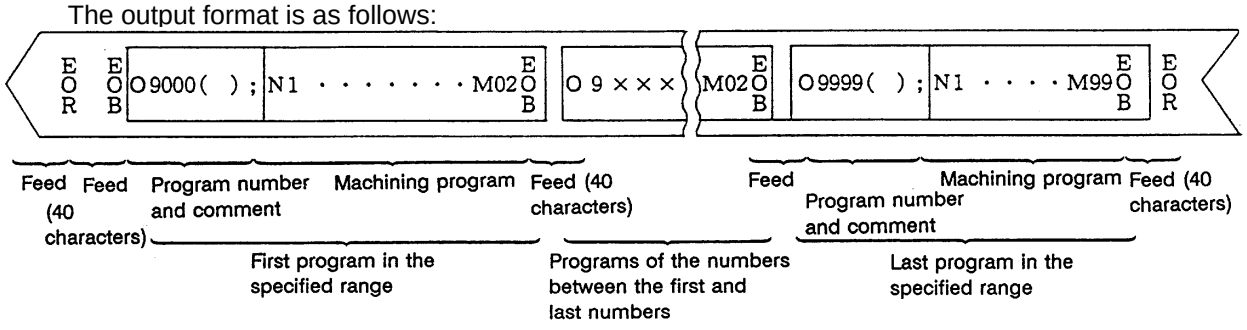
Press the INPUT key.

The machining programs are output sequentially.

1) Data output is started.
2) The number and the contents of the machining program being output are displayed in the OUTPUT DATA display area.
3) When all the specified machining programs are output, data set in the data setting area disappears and instead message "DATA OUT COMPLETE" is displayed.



The output format is as follows:



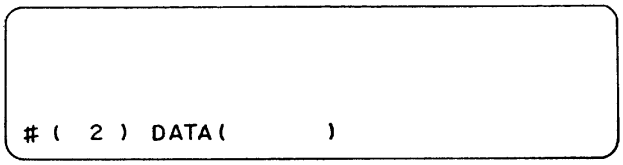
(Note) 1. If the number specified as the smallest number is not found, output starts with the machining program with the number nearest to that number.
Likewise, if the number specified as the largest number is not found, output ends with the machining program with the number nearest to that largest number.
2. Specify the smallest number first, then the largest number. If the numbers are specified reversely, program error E01 SETTING ERROR occurs.

2.6.2.2 Outputting Tool Offset Data

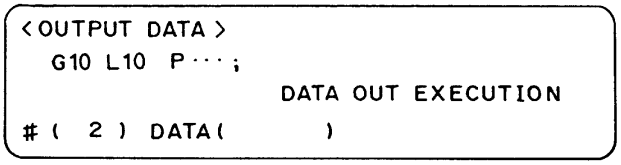
Tool offset data which is set and displayed on the screen can be output. The output operation can be made also during automatic operation.
The output tape length varies with the tool offset type, the number of sets, and the offset data numeric. For the 40-set specification, this length is 3 to 4m for type I and 12 to 16m for type II.

(1) Tool offset data is output as follows:

Set the data type to tool data.
(2) DATA ( )

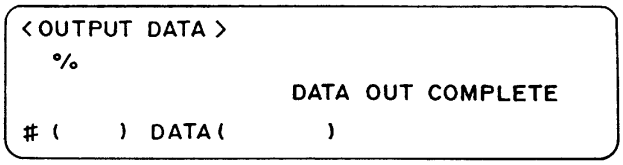


Press the INPUT key.

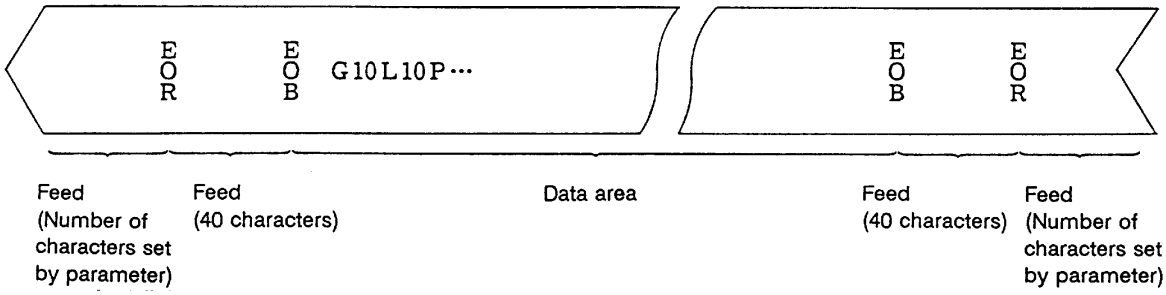


1) Data output is started.
The contents of output data and message "DATA OUT EXECUTION" are displayed.

2) After the end code % (EOR) is output, data output ends with message "DATA OUT COMPLETE" displayed.



(2) Output format



The data area format is the same as tool offset input (G10) and work offset input (G10) by the program.
The data is output in the order of the tool offset and workpiece coordinate offset data.

2.6.2.3 Outputting Parameter Data

Parameter data which is set and displayed on the screen can be output. The output operation can be made also during automatic operation.
The output tape length varies with the number of axes and parameter numerics. For the three-axis specification, this length is 40 to 50m. The following data is output:

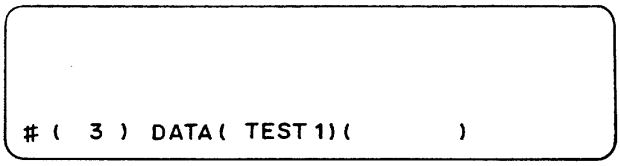
- User parameters (Machining parameters, control parameters, axis parameters)
- Data input/output parameters (I/O BASE PARAM, I/O DEVICE PARAM)
- All setup parameters
- Internal parameter data (absolute position internal data)

(Note 1) The TOOL OFFSET, TOOL REGISTRATION, TOOL LIFE, and WORK OFFSET data are not output.

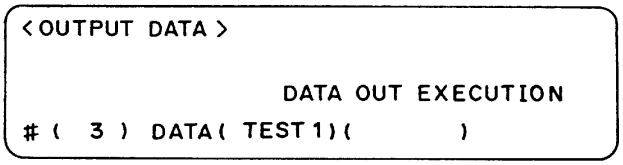
(Note 2) To output the parameters related to MELSECNET/10, set as “#(99) DATA (ALL2)”.

(1) Outputting parameter data

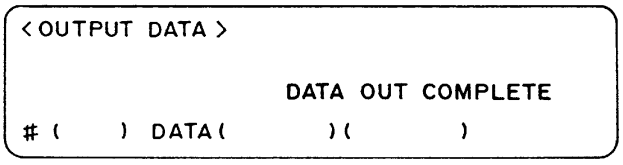
Set the data type to parameter.
(3) DATA (T E S T 1)



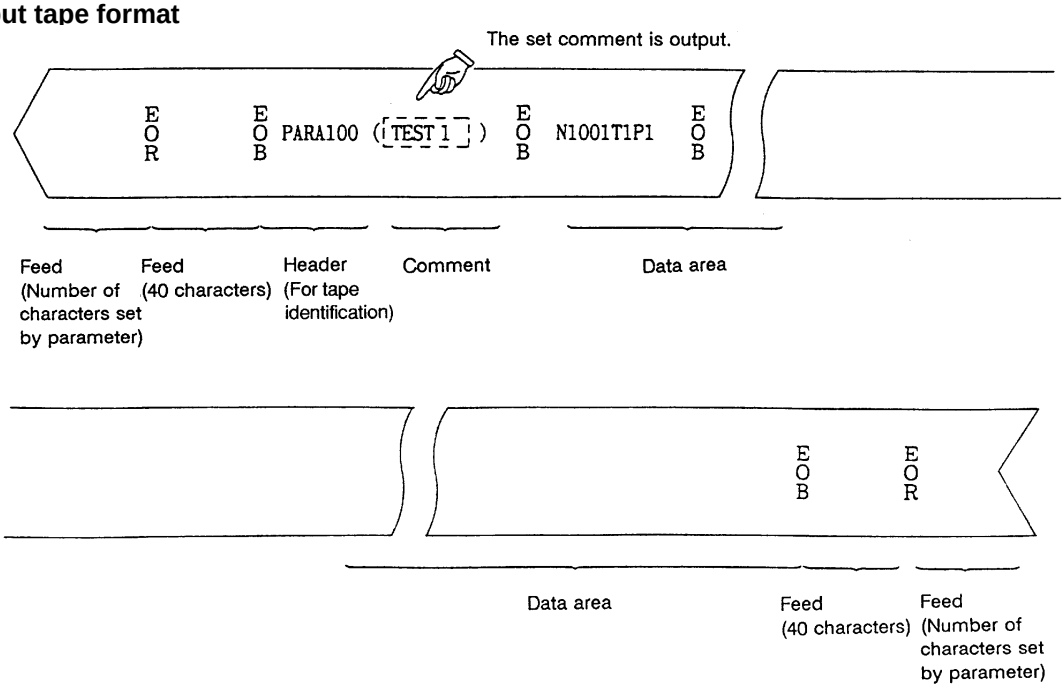
Press the INPUT key.



1) Output to parameter data is started. The message "DATA OUT EXECUTION" are displayed.
2) After the end code % (EOR) is output, data output ends with the message "DATA OUT COMPLETE" displayed.



(2) Output tape format



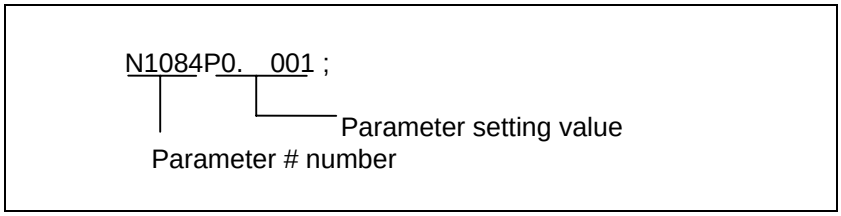
(3) Data format

The data format is as follows:

Address	Definition	Details
N	Parameter number	The parameter # number is indicated by the value following “N.”
A	Axis number	For axis data, the axis number is indicated by the value following “A”. The 1st axis is indicated as "A1".
T	Part system number	For data per part system, the part system number is indicated by the value following “T”. (1st part system: T1, 2nd part system: T2, PLC axis: T3)
P	Parameter data	The parameter data is indicated by the value following “P”.
C	Spindle Number	For spindle data, the spindle number is indicated by the value following “C”. (ex.) 1st spindle: C1 Note that this expression is available only when two or more spindles are used.

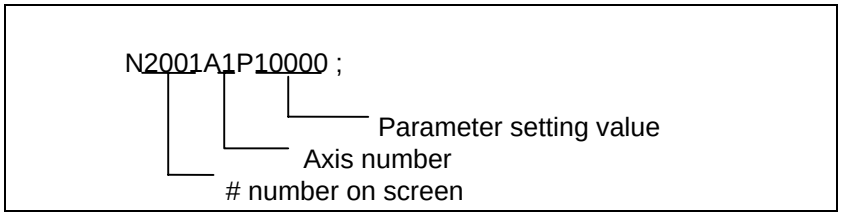
The following types of data format are used according to parameter type and display method. (The address order in one block must use the following format.)

1) Common parameter (one data item per one # number)



(1) The output parameter setting value is the same format as the screen display.

2) Axis parameter



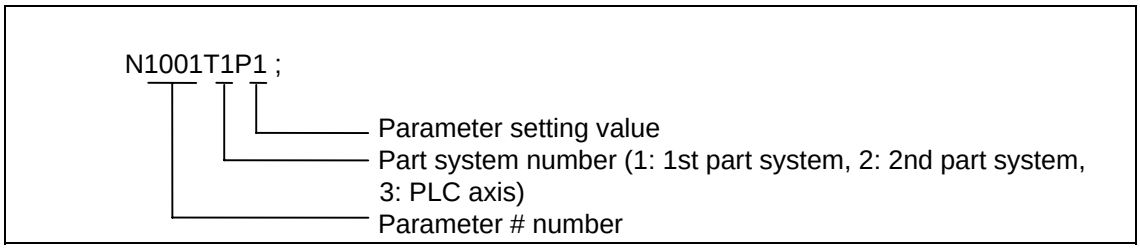
(a) When multiple axes are displayed on one screen

The parameter data for when the parameters for multiple axes are displayed on one screen are output per axis.

[Output example]

```
    :
N2001A1P12000 ;
N2002A1P4000 ;          ) Axis 1 data
N2003A1P21 ;
    :
N2001A2P12000 ;
N2002A2P4000 ;          ) Axis 2 data
N2003A2P21 ;
    :
```

3) Part system parameter



(a) The parameter data on the screen when the parameters are displayed per part system are output as follows.

[Output example]

```
    :
N1001T1P1 ;
N1001T2P1 ;
N1001T3P0 ;
N1002T1P2 ;
N1002T2P1 ;
N1002T3P0 ;
    :
```

(b) The parameter data per part system displayed by changing over the part system ([SHIFT] and [$ 4] key) is output per part system screen.

[Output example]

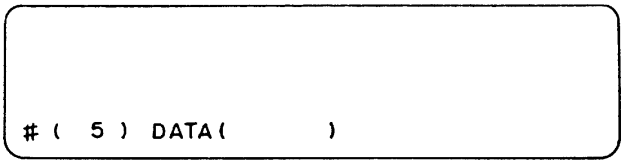
```
    :
N8001T1P99 ;
N8002T1P0 ;
N8003T1P10000 ;
    :
N8001T2P30 ;
N8002T2P1 ;
N8003T2P20000 ;
    :
```

2.6.2.4 Outputting Common Variable Data

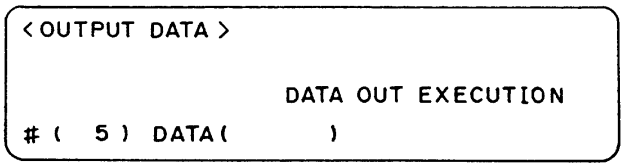
Common variable data can be output. The output operation can be performed even during automatic operation.

(1) Common variable data output operation

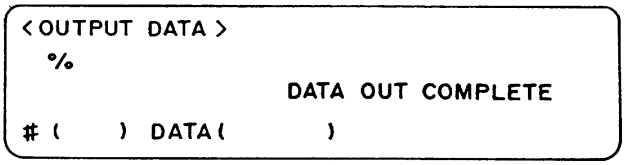
Set the data type to the common variable.
(5) DATA ( )



Press the INPUT key.



1) Data output starts and the contents of the output data and message "DATA OUT EXECUTION" are displayed.
2) When tape end code % (EOR) is output, the data output completes with message "DATA OUT COMPLETE" displayed.



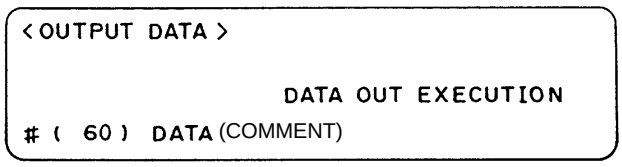
2.6.2.5 Outputting History Data

History data can be output. The output operation can be performed even during automatic operation.

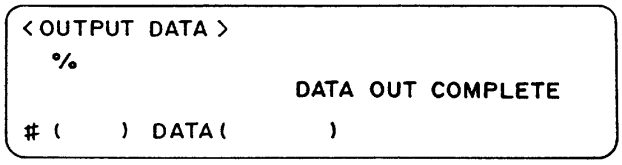
(1) History data output operation

Set the data type to the history data.
(6 0) DATA (COMMENT)

Press the INPUT key.



1) Data output starts and the contents of the output data and message "DATA OUT EXECUTION" are displayed.
2) When tape end code % (EOR) is output, the data output completes with message "DATA OUT COMPLETE" displayed.



Format of data output to RS-232C

(Note) The output data is ASCII or Shift_JIS code.

(Example)

```
%
TRACE (COMMENT    )  LF
00/01/01 SP 13:59:58 SP CAN LF
00/01/01 SP 13:59:56 SP Y220 LF
00/01/01 SP 13:59:55 SP !Y220 LF
00/01/01 SP 13:59:02 SP P153 I.F ERROR $1 LF
                :
00/01/01 SP 13:00:56 SP Y218 LF
%
```

* Comment is set in the parentheses of TRACE ( ).

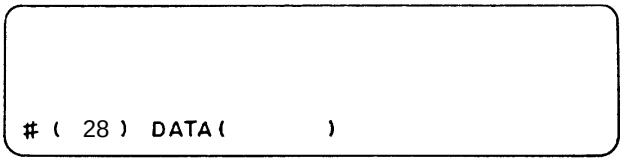
* If the input/output device parameter "#9112 CR OUTPUT" is set to 1, the history data can be output with a CR attached before LF.

2.6.2.6 Outputting Auxiliary Axis Parameter Data

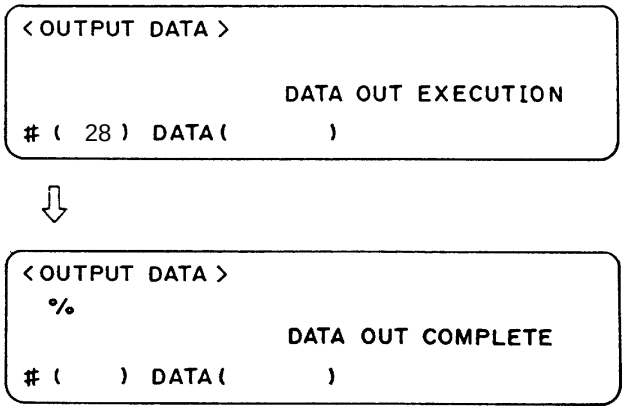
Auxiliary axis parameter data can be output.

(1) Auxiliary axis parameter data output operation

Set the data type to the auxiliary axis parameter.
(2 8) DATA ( )



Press the INPUT key.



1) Data output starts and the contents of the output data and message "DATA OUT EXECUTION" are displayed.
2) When tape end code % (EOR) is output, the data output completes with message "DATA OUT COMPLETE" displayed.

(Note) If MR-J2-CT is not connected, "E01 SETTING ERROR" will occur and output will not be carried out.

2.6.2.7 MR-J2-CT Parameters and N No. Correspondence Table

No.	Symbol name	N No.	Remarks
1	∗MSR	50001	Automatic setting∗2
2	∗RTY	50002	
3	∗PC1	50003	
4	∗PC2	50004	
5	∗PIT	50005	
6	INP	50006	
7	ATU	50007	
8	PG1	50008	Auto-tuning
9		50009	
10	EMG	50010	
11		50011	
13	MBR	50013	
14	NCH	50014	
16	JIT	50016	
19	PG2	50019	Auto-tuning
20	VG1	50020	Auto-tuning
21	VG2	50021	Auto-tuning
22	VIS	50022	Auto-tuning
23	VDC	50023	Auto-tuning
24	DG2	50024	Auto-tuning
30	∗MTY	50030	∗1
31	∗TMX	50031	∗1
32	∗PMS	50032	∗1
33	∗BAS	50033	∗1
34	∗MAX	50034	∗1
35	∗AMR	50035	∗1
36	∗JMK	50036	∗1
37	∗KCM	50037	∗1
38	∗KVI	50038	∗1
39	∗VGM	50039	∗1
40	∗MLD	50040	∗1
41	∗KEC	50041	∗1
42	∗IQG	50042	∗1
43	∗IDG	50043	∗1
44	∗IQI	50044	∗1
45	∗IDI	50045	∗1
50	MD1	50050	Automatic setting∗2
51	MO1	50051	Automatic setting∗2
53	MD2	50053	Automatic setting∗2
54	MO2	50054	Automatic setting∗2
56	sty02	50056	

No.	Symbol name	N No.	Remarks
100	*∗station	50100	
101	Cont1	50101	
102	∗Cont2	50102	
103	∗EmgCont	50103	
104	∗tleng	50104	
105	Axis nam	50105	
110	ZRNspeed	50110	
111	ZRNcreep	50111	
112	grid mask	50112	
113	∗grspc	50113	
114	ZRNshift	50114	
115	ST.ofset	50115	
116	ABS Base	50116	
117	Limit(+)	50117	
118	Limit(–)	50118	
120	ABS Type	50120	
123	ABScheck	50123	
130	backlash	50130	
132	yobi16a	50132	
133	yobi16b	50133	
134	yobi32a	50134	
135	yobi32b	50135	
150	Aspeed1	50150	
151	Mspeed1	50151	
152	time1.1	50152	
153	time1.2	50153	
154	TL1	50154	
155	OD1	50155	
156	just1	50156	
157	near1	50157	
158	Aspeed2	50158	
159	Mspeed2	50159	
160	time2.1	50160	
161	time2.2	50161	
162	TL2	50162	
163	OD2	50163	
164	just2	50164	
165	near2	50165	
166	Aspeed3	50166	
167	Mspeed3	50167	
168	time3.1	50168	
169	time3.2	50169	
170	TL3	50170	
171	OD3	50171	
172	just3	50172	
173	near3	50173	
174	Aspeed2	50174	
175	Mspeed4	50175	

No.	Symbol name	N No.	Remarks
176	time4.1	50176	
177	time4.2	50177	
178	TL4	50178	
179	OD4	50179	
180	just4	50180	
181	near4	50181	
190	stpos1	50190	
191	stpos2	50191	
192	stpos3	50192	
193	stpos4	50193	
194	stpos5	50194	
195	stpos6	50195	
196	stpos7	50196	
197	stpos8	50197	
200	PSWcheck	50200	
201	PSW1dog1	50201	
202	PSW1dog2	50202	
203	PSW2dog1	50203	
204	PSW2dog2	50204	
205	PSW3dog1	50205	
206	PSW3dog2	50206	
207	PSW4dog1	50207	
208	PSW4dog2	50208	
209	PSW5dog1	50209	
210	PSW5dog2	50210	
211	PSW6dog1	50211	
212	PSW6dog2	50212	
213	PSW7dog1	50213	
214	PSW7dog2	50214	
215	PSW8dog1	50215	
216	PSW8dog2	50216	
220	push.L	50220	
221	push.t1	50221	
222	push.t2	50222	
223	push.t3	50223	

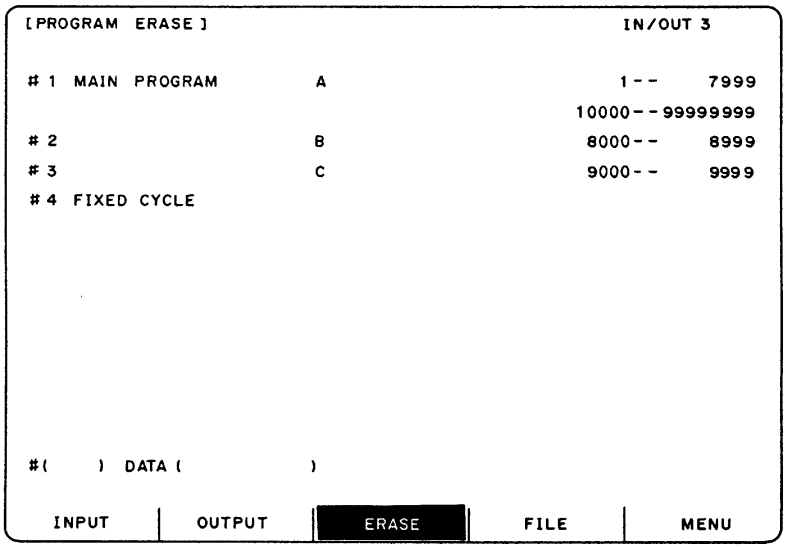
(Note 1) The parameters marked with *1 cannot be set from the screen. (Setting is possible only from the optional setup software.)
Note that these parameters can be input/output or backed up to SRAM same as the other parameters.

(Note 2) The items marked with *2 are automatically set, but these parameters can be input/output or backed up to SRAM same as the other parameters.

2.6.3 Program Erase

When the menu key 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.

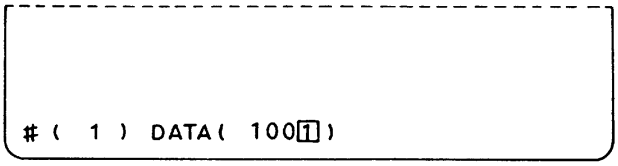


#	Item	Explanation	Data setting range		
			Program number specification	Program group erase	All program erase
1	MAIN PROGRAM <A>	This is specified to erase data in the range of work program numbers 1 to 7999 and 10000 to 99999999.	1 to 7999 and 10000 to 99999999	ALL	CLR
2	MAIN PROGRAM <B>	This is specified 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.			

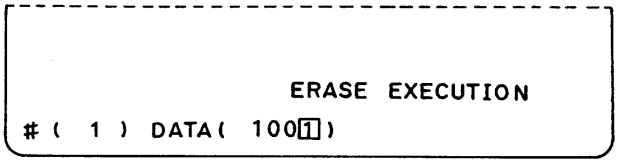
(1) To erase one machining program

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

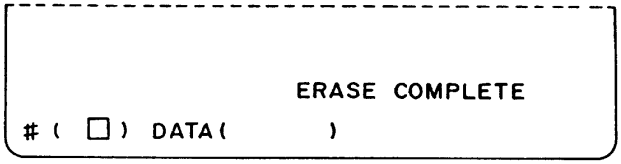
Set the data type and program number of the machining program to be erased.
(Example)
To erase machining program O1001
(1)
DATA (1 0 0 1)



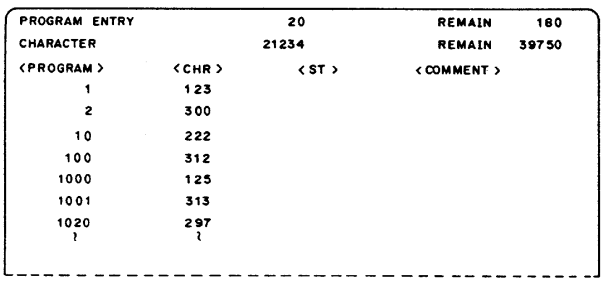
Press the INPUT key.

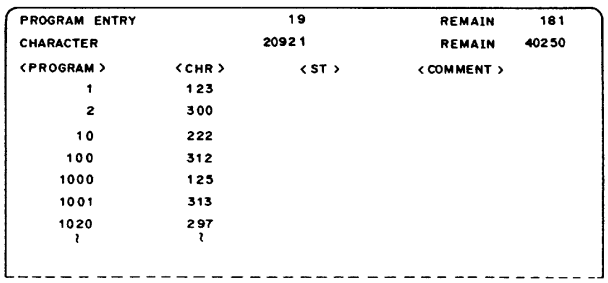


1) Program erase is started and the message ERASE EXECUTION is displayed in the message field. Usually program erasure is terminated in a moment.
2) When the erase is complete, the message ERASE COMPLETE is displayed in the message field and the data setting area becomes blank.



PROGRAM FILE before erase execution — Erase execution → PROGRAM FILE after erase execution



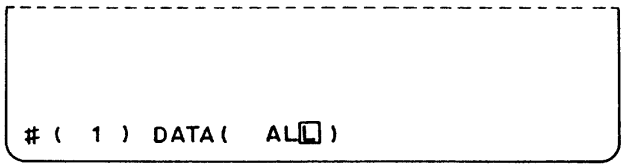


Program O1001 for which the erase function is executed disappears from the PROGRAM FILE screen. The new values are displayed in PROGRAM ENTRY, CHARACTER, and REMAIN.

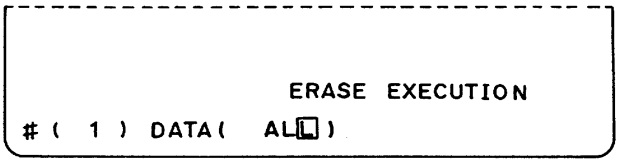
(2) To erase machining program group

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:

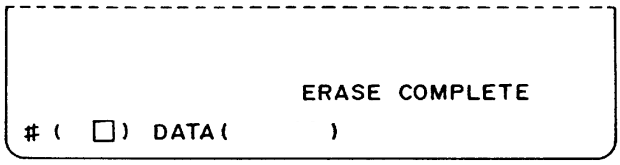
Set the data type to be erased and "ALL" in DATA ( ).
(Example)
To erase the entire machining program A group
(1) DATA (A L L)



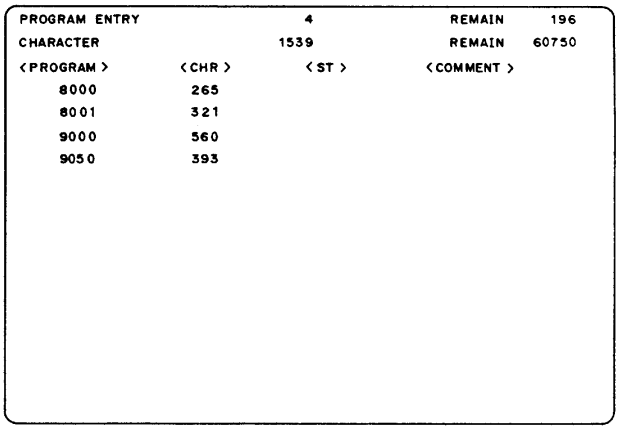
Press the INPUT key. ☞



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.



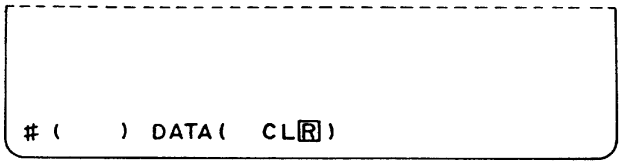
Check REMAIN, etc. on the PROGRAM FILE screen. ☞



(3) To erase all machining programs registered in memory

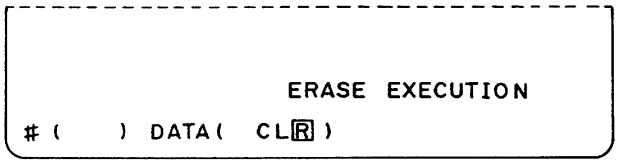
To erase all of machining programs 1~99999999 registered in memory, perform the following:

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

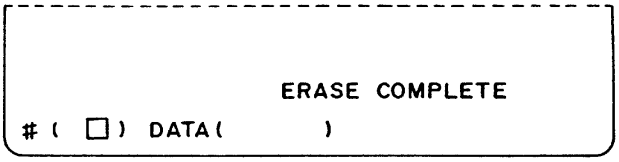


1) No value needs to be entered in # ( ).

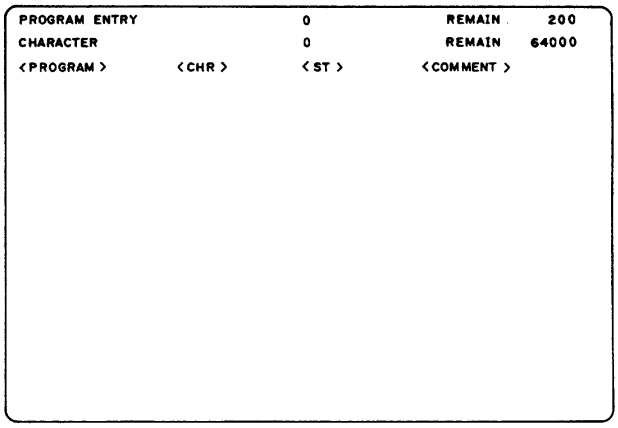
Press the INPUT key.



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:

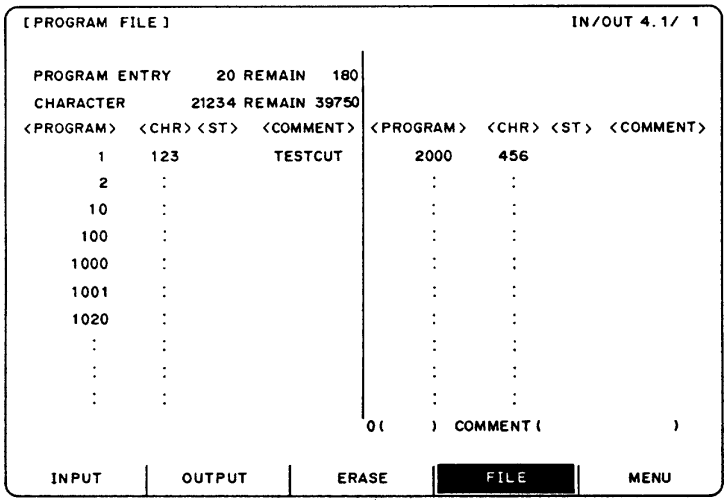
Tape storage length	Number of programs	Number of characters
40m	64	17500
80m	128	32000
160m	200	64000
320m	200	128000
600m	400	236250
1280m	1000	512000
2560m	1000	1024000
5120m	1000	2048000

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

2.6.4 Program File

When the menu key [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.

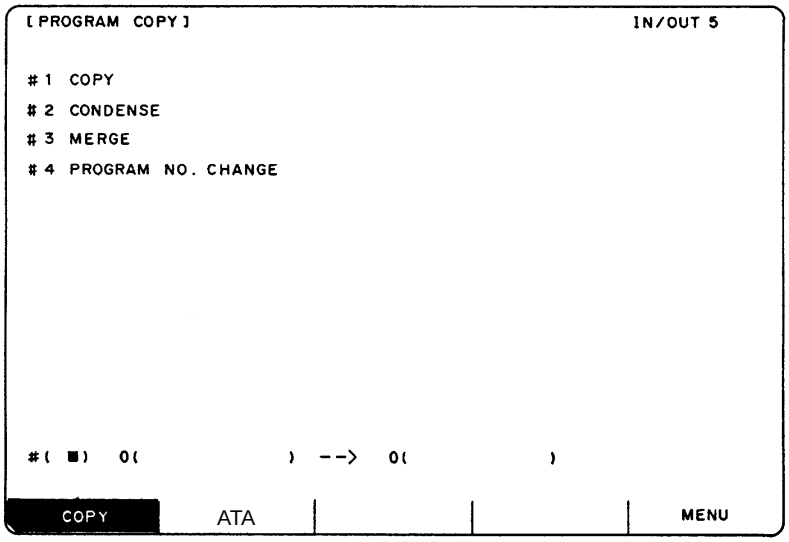
(Note 1) If a large number of short programs are registered or edit operation is performed frequently, the memory use efficiency lowers because of the memory configuration and (CHARACTER + REMAIN) becomes less than the maximum number of available characters that are stored. If the difference between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

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

2.6.5 Program Copy

When the menu key) COPY is pressed, the PROGRAM COPY screen is displayed.

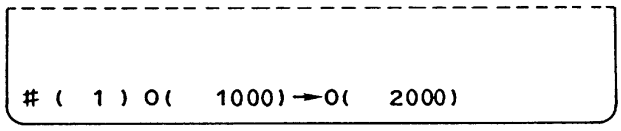
Copy, condense, merge, and number change of user-prepared machining programs (main program and subprogram) can be performed on the PROGRAM COPY screen.



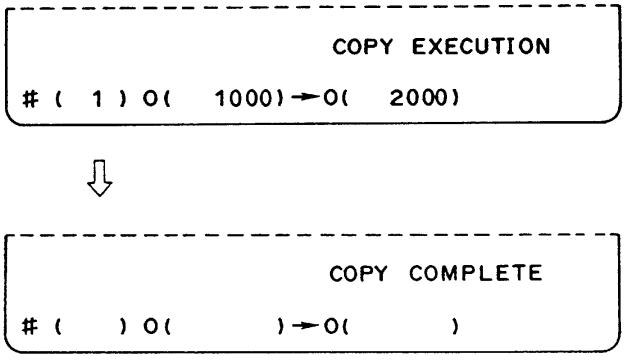
2.6.5.1 Machining Program Copy

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.



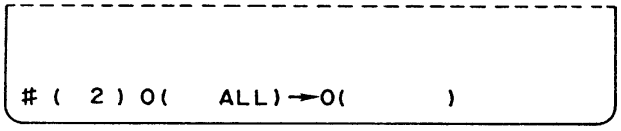
1) A copy is started and the message COPY EXECUTION is displayed in the message field. It may be terminated in a moment (when there is less copy data).

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

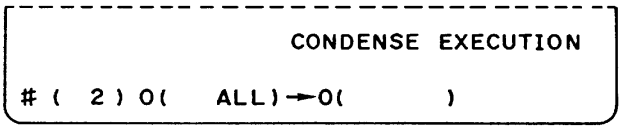
2.6.5.2 Machining Program Condense

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 ( ).

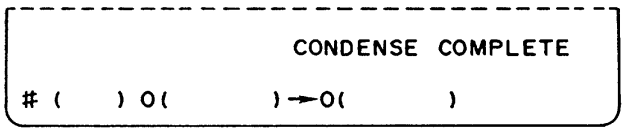
To condense all work programs registered in memory:
(2) O (ALL) → O ( )



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.



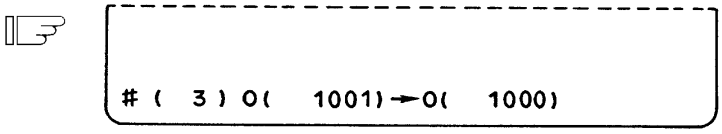
2.6.5.3 Machining Program Merge

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.

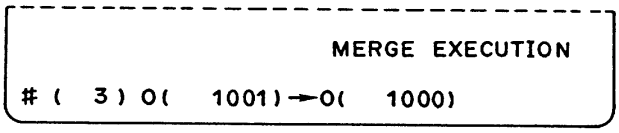
(Example 1)
To move program O1001 to the area following program O1000 for merge

(3)

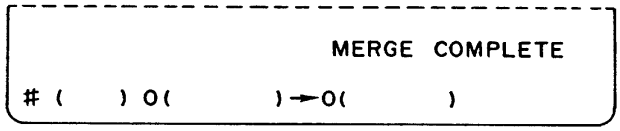
O (1 0 0 1) → O (1 0 0 0)



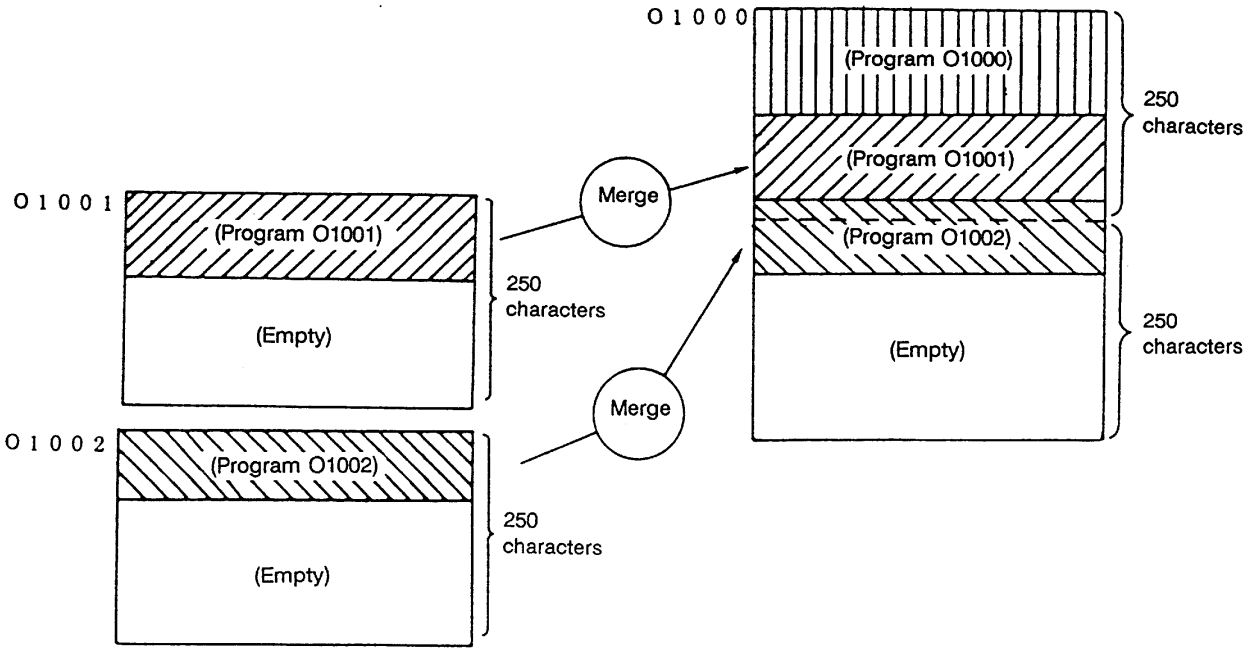
Press the INPUT key.



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.



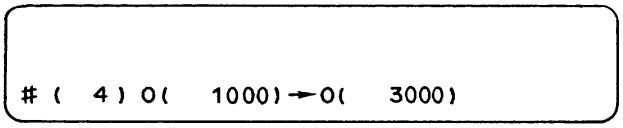
2.6.5.4 Changing the Machining Program Number

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.

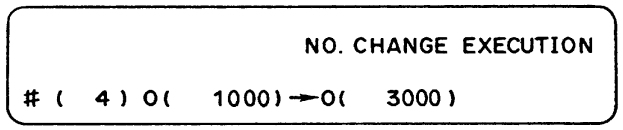
(Example)
When O1000 is changed to O3000:

(4)

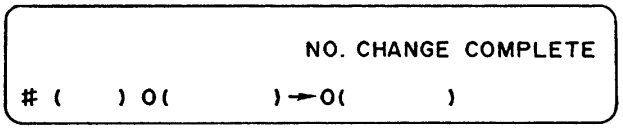
O (1 0 0 0) → (3 0 0 0)



Press the INPUT key.

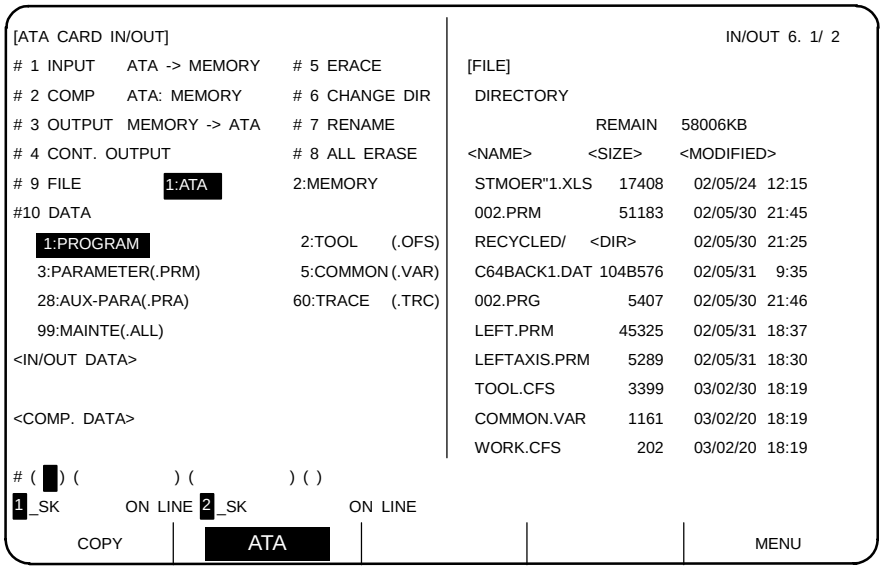


1) Number change processing is initiated with "NO. CHANGE EXECUTION" displayed in the message field. This processing is completed in a moment. Then, the message "NO. CHANGE COMPLETE" is displayed.

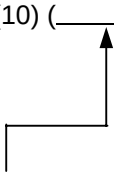


2.6.6 Flash ATA Card I/F

With this screen, the data such as the machining programs, parameter data and tool data can be input to or output from the flash ATA card mounted in the CNC.
This data is stored as the DOS format in the flash ATA card, so the data can be referred to or edited with a computer compatible with the flash ATA card.
A 2GB or smaller flash ATA card can be used.
If the flash ATA card is not mounted, an error will occur and all input/output operations will be disabled.



#	Display items	Details
1	INPUT ATA CARD -> MEMORY Set # (1) to enter the above mode.	The files in the flash ATA card are input into the NC memory. Designate the name of a file in the card to be input.
2	COMP ATA : MEMORY Set # (2) to enter the above mode.	The files in the flash ATA card and the contents of the NC memory are compared. Designate the name of the file in the card to be compared.
3	OUTPUT MEMORY -> ATA Set # (3) to enter the above mode.	The contents of the NC memory are output to the flash ATA card as a file. The output destination file name can be designated. When outputting machining programs, set the machining program No. in the NC and the name of the file in the output destination card.
4	CONT. OUTPUT Set # (4) to enter the above mode.	Machining programs in the NC memory are continuously output. By setting the range of program Nos. to be output in the setting area, the machining programs with the Nos. in the designated range will be continuously output.
5	ERASE Set # (5) to enter the above mode.	The files and directories in the flash ATA card are deleted. Designate the name of the file or directory in the card to be deleted.
6	CHANGE DIR Set # (6) to enter the above mode.	The directory in the flash ATA card is changed or a new directory is created. Set the directory name with up to 8 alphanumeric characters. The name is not case sensitive.
7	RENAME Set # (7) to enter the above mode.	The name of a file in the flash ATA card is changed. The directory name cannot be changed.
8	ALL ERASE Set # (8) to enter the above mode.	All contents of the flash ATA card, including the directories, are deleted.
9	FILE 1: ATA 2: MEMORY Set # (9) to enter the above mode.	Select whether the files in the flash ATA card or the machining programs in the NC memory are listed in the display on the right side of the screen.

#	Display items	Details
10	DATA # (10) (_____) ( )  1 MACHINING PROGRAM 2 TOOL (.OFS) 3 PARAMETER (.PRM) 5 COMMON (.VAR) 28 AUX-PARA (.PRA) 60 TRACE DATA (.TRC) 99 MAINTE	• Select the data to be input or output. The file name extension is shown in the parentheses at the right of the data type. • As shown on the left, set "10" in the first setting area and the number of the data to be selected in the second setting area. 1: Input/output machining programs (Fixed cycle programs can be input/output when the base specifications parameter "#1166 fixpro" is valid.) 2: Input/output tool offset data 3: Input/output parameter data 5: Input/output common variable data 28: Input/output auxiliary axis parameter data 60: Input history data 99: Input/output maintenance data # (10) (99) (_____) ↑ Refer to table below
	<IN/OUT DATA> <COMP DATA>	When inputting, comparing or outputting the data, the symbols "→", "→→" and "→→→" will appear sequentially in <IN/OUT DATA> If the comparison mismatches, the data will appear in <IN/OUT DATA> and <COMP DATA>.

Selecting the input/output maintenance data

3rd setting area	Type of input/output maintenance data		Extension
ALL1	System data	No.100 to 149	.AL1
ALL2	File data	No.200 to 231	.AL2
ALL3	PLC program	No.250, No.252 to 299	.AL3
Individual data No.	Individual data (including No.150 to 199 and No.251)		.MNT
Blank	All ALL1 to ALL3		.ALL

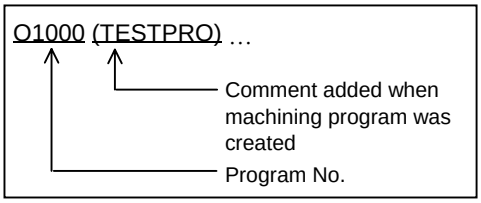
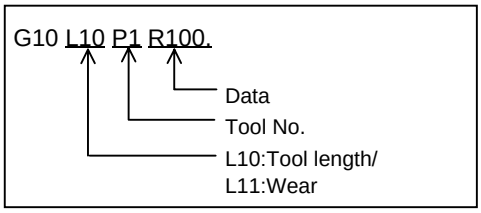
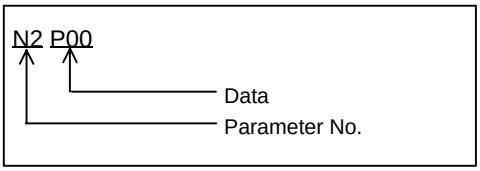
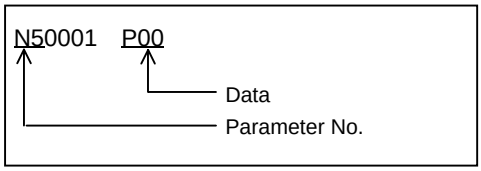
Files are saved in the flash ATA card with the following formats.

Data type	File name		Extension
Machining program **(Note 1)**	8 characters or less	Alphabet and number **(Note 2)**	None **(Note 3)**
Tool data		Alphabet and number	.OFS (automatically assigned)
Parameter			.PRM (automatically assigned)
Common variable			.VAR (automatically assigned)
Auxiliary axis parameter			.PRA (automatically assigned)
History data		Alphabet and number	.TRC (automatically assigned)
Maintenance data			
System data No.100 to 149			.AL1 (automatically assigned)
File data No.200 to 231			.AL2 (automatically assigned)
PLC program No. 250, No.252 to 299			.AL3 (automatically assigned)
Individual data (including No.150 to 199 and No.251)		**(Note 4)**	.MNT (automatically assigned)

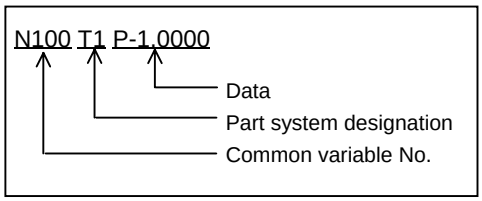
(Note 1) A parameter must be set to input and output fixed cycle programs.
(Note 2) The program No. in the memory is a number with eight or less digits.
(Note 3) When assigning the file name, the user can assign the name randomly with three or less digits.
(Note 4) For independent data, if the data type No. is set to n, "n.MNT" will be the independent data file name.

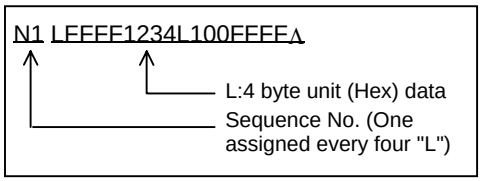
<Output format>

The files in the flash ATA card are output with the following format.
The end of the line is "CR+LR" instead of ";".
"%" is output at the end of the file.

Data type	Output example
Machining program **(Note 1)**	O1000 (TEST PROGRAM) CRLF : Header "0" + "program No." + "(" + "comment" + ")" N010G28X0. CRLF N200G00X-100. CRLF : : M02 CRLF : Machining program % 
Tool data	$1 CRLF : Part system selection G10L10P1R100. CRLF G10L10P2R123. CRLF : : G10L2P6X0.000Z0.000 CRLF : Tool data % 
Parameter data	PARA200 ( ) CRLF : Header "PARA100" or "PARA200" N1 P74 CRLF N2 P00 CRLF N3 P00 CRLF : Parameter data : : % 
Auxiliary axis parameter data	PARA200 ( ) CRLF : Header "PARA100" or "PARA200" N50001A1P0000 CRLF N50002A1P0000 CRLF N3 P00 CRLF : Parameter data : : : % 

(Note 1) One file contains one machining program.
Normally eight or less digits are set as the file name in the same manner as the machining program number saved in the memory.
A random "eight or less alphanumeric characters, three or less character extension" can be used for the file name.
Note that if an extension unique to the above data ("OFS", "PRM", etc.) is used for the extension, an error will occur.
If the file name contains characters other than numbers or an extension, the machining program No. must be designated when the file is to be saved in the NC memory again.

<table>
<tr><th>Data type</th><th>Output example</th></tr>
<tr><td>Common variable data</td><td>
COMN ( ) CRLF : Header "COMN"
N100 T1 P-1.0000 CRLF
N101 T1 P CRLF
:
:
:
N549P549.000 CRLF
%
: Common variable data


</td></tr>
<tr><td>History data</td><td>
TRACE ( ) CRLF : Header "TRACE"
03/06/24 23:01:30 INP CRLF
03/06/24 22:38:45 M01 Operation error 0101 $2
:
:
03/06/24 22:38:45 !X6B3 CRLF
%
: Operation history data
</td></tr>
<tr><td>Maintenance data</td><td>
O250 CRLF : Data type No. (O:100 to 149, 200 to 230, 250 to 239)
N0 LFFFF1234 L42FFFF CRLF
N1 L520100 L520500 CRLF
:
:
:
%
: Maintenance data


</td></tr>
</table>

2.6.6.1 Outline of Operation Procedure

The outline of the operation procedures is explained below.
Refer to section "2.6.6.2 List of Input/Output Operations" for details on the setting format for each operation.
Refer to the section following "2.6.6.3 Input/Compare" for examples of actual operations.

(Example) Input the file "10" in the flash ATA card directory "BACKUP" into the NC memory as the No. 10 machining program.

(1) Select the directory in the flash ATA card. (#6)

Change the directory to "BACKUP". # (6) (BACKUP) ( ) INPUT	☞	The files in the directory "BACKUP" will appear in the list.

To select data in the NC memory, change the list display with "#9 FILE".

(2) Select the target data. (#10) -> Refer to “2.6.6.2 List of Input/Output Operations”.

Select the target data (machining program) # (10) (1) ( ) INPUT	☞	"1: PROGRAM" at "#10 DATA" will be highlighted.

(3) Select the function. (#1 to 8) -> Refer to “2.6.6.3 Input/Compare” and following sections.

Input file "10" as machining program "10". (flash ATA card → memory) # (1) (10) ( ) INPUT	☞	Input from the flash ATA card will start.

(4) Check that the operation has ended correctly.

List the machining programs in the NC memory and check. # (9) (2) ( ) INPUT	☞	The No. of the machining programs registered in the NC memory will appear. When the input is completed, machining program "10" will be added to the list.

2.6.6.2 List of Input/Output Operations

The input, compare and output (continuous output) setting methods differ slightly according to the data that is to be input or output.
The methods of setting each operation are shown below.

(1) Machining program (When #(10) (1) is set)

n, m: Eight or less digit setting
s : Eight or less alphanumeric setting, extension (may not include extension in some cases)

Operation		Setting method	
Input (#1)	**Basic setting**	**# (1) (File name in ATA card) (Program No. in NC memory)**	
	Single file designation	#(1) (n) ( )	Inputs ATA card's file n as n in the NC memory.
		#(1) (n) (m)	Inputs ATA card's file n as m in the NC memory.
		#(1) (s) (m)	Inputs ATA card's file s as m in the NC memory.
	Batch designation	#(1) (∗) ( )	Inputs all files with a name eight digits or shorter in the ATA card's designated directory into the NC memory.
		#(1) (n++) ( )	Inputs all files with a name starting with a number (n) and having the designated number of digits. The designated number of digits is indicated with "+". An error will occur if "+" is input in between, such as "7+9+" or "+7++", or if the name does not start with a number. **(Example)** "78++": Input all files with the names 7800 to 7899.
Compare (#2)	**Basic setting**	**# (2) (File name in ATA card) (Program No. in NC memory)**	
	Single file designation	Same as input. However, when comparing, change the #(1) setting area to #(2).	
	Batch designation		
Output (#3)	**Basic setting**	**# (3) (Program No. in NC memory) (File name in ATA card)**	
	Single file designation	#(3) (m) ( )	Outputs the NC memory m as the ATA card's file m.
		#(3) (m) (n)	Outputs the NC memory m as the ATA card's file n.
		#(3) (m) (s)	Outputs the NC memory m as the ATA card's file s.
	Batch designation	#(3) (ALL) ( )	Outputs all programs in the NC memory as the ATA card file "ALL.PRG".
		#(3) (ALL) (s)	Outputs all programs in the NC memory as the ATA card file "s".
Continuous output (#4)	**Basic setting**	**# (4) (Program No. in NC memory) (File name in ATA card)**	
	Single file designation	#(4) (m) ( )	Outputs the NC memory m as the ATA card's file m.
	Range designation	#(4) (n) (m)	Outputs programs from n to m in the NC memory as the ATA card files n to m.
	Batch designation	#(4) (ALL) ( )	Outputs all programs in the NC memory to the ATA card.

(Note) The same operation is used for the output batch designation and continuous output batch designation.

(2) Tool data (When #(10) (2) is set)

s : Eight or less alphanumeric setting, extension
(Extension unique to each data is automatically added)

Operation		Setting method
Input (#1)	**Basic setting**	**# (1) (File name in ATA card) ( )**
	Single file designation	#(1) (s) ( ) Inputs file s from the ATA card to the NC memory.
Compare (#2)	**Basic setting**	**# (2) (File name in ATA card) ( )**
	Single file designation	#(2) (s) ( ) Compares file s in the ATA card with the NC memory data.
Output (#3)	**Basic setting**	**# (3) (File name in ATA card) ( )**
	Single file designation	#(3) (s) ( ) Outputs the NC memory data to the ATA card as file s.

(3) Parameter, common variable, auxiliary axis parameter, history data (When #(10) and (3), (5), (28), or (60) are set)

s : Eight or less alphanumeric setting, extension (may not include extension in some cases)
t : Twelve or less alphanumeric setting

Operation		Setting method
Input (#1)	**Basic setting**	**# (1) (File name in ATA card) ( )**
	Single file designation	#(1) (s) ( ) Inputs file s from the ATA card to the NC memory.
Compare (#2)	**Basic setting**	**# (2) (File name in ATA card) ( )**
	Single file designation	#(2) (s) ( ) Compares file s in the ATA card with the NC memory data.
Output (#3)	**Basic setting**	**# (3) (File name in ATA card) (Comment character string)**
	Single file designation	#(3) (s) (t) Outputs the NC memory data to the ATA card as file s. Outputs comment t to output file header.

(Note) The history data cannot be input or compared.

(4) Maintenance data (When #(10) (99) is set)

s : Eight or less alphanumeric setting, extension (may not include extension in some cases)
n : Data type No. (100 to 149, 150 to 199, 200 to 231, 250 to 299)
k : Character string set in 3rd setting area at "#10 DATA" (ALL1, ALL2, ALL3, data type No.)
(10) (99) (k)

Operation	Setting method		
Input (#1)	**Basic setting**	**When k is "ALLx" : # (1) (File name in ATA card) ( )** **When k is "data type No." : # (1) (File name in ATA card) ( )**	
	Single file designation	#(1) (s) ( )	When K is "ALLx", the files s.ALx in the ATA card are input to the NC memory.
		#(1) (n) ( )	When K is "data type No.", the files n.MNT in the ATA card are input to the NC memory.
		#(1) ( ) ( )	When K is "data type No.", the files k.MNT in the ATA card are input to the NC memory.
Compare (#2)	**Basic setting**	**When k is "ALLx" : # (2) (File name in ATA card) ( )** **When k is "data type No." : # (2) (File name in ATA card) ( )**	
	Single file designation	#(2) (s) ( )	When k is "ALLx", the files s.ALx in the ATA card are compared with the data in the NC memory
		#(2) (n) ( )	When k is "data type No.", the files n.MNT in the ATA card are compared with the data in the NC memory.
		#(2) ( ) ( )	When k is "data type No.", the files k.MNT in the ATA card are compared with the data in the memory.
Output (#3)	**Basic setting**	**When k is "ALLx" : # (3) (File name in ATA card) ( )** **When k is "data type No." : # (3) (File name in ATA card) ( )**	
	Single file designation	#(3) (s) ( )	When k is "ALLx", the maintenance data in the NC memory is output as the file s.Alx in the ATA card.
		#(3) (n) ( )	When k is "data type No.", the maintenance data with the data type is output as the file n.MNT in the ATA card.
		#(3) ( ) ( )	When k is "data type No.", the maintenance data with the data type is output as the file k.MNT in the ATA card.

(Note) An “x” of “ALLx” or “.ALx” indicates one of 1, 2, or 3.

(5) Common operations

n, m: Eight or less digit setting
s, t : Eight or less alphanumeric setting, extension (may not include extension in some cases)

Operation	Setting method		
Delete (#5)	**Basic setting**	**# (5) (File name in ATA card) ( )**	
	Single file designation	#(5) (n) ( )	Deletes file n from the ATA card. (Extension corresponds to the data designated with "#10 DATA")
		#(5) (s) ( )	Deletes file s from the ATA card. (Extension corresponds to the data designated with "#10 DATA")
	Batch designation	#(5) (∗) ( )	Deletes all files that do not have extensions in the designated directory of the ATA card.
		#(5) (∗.∗) ( )	Deletes all files in the designated directory of the ATA card.
		#(5) (A1++) ( )	Deletes all alphanumeric file names having the designated number of characters starting with an alphanumeric character. The designated number of characters is indicated with "+". If "+" is inserted such as "A+1+" or "+A++", or if the designation does not start with an alphabetic character, an error will occur. All the files having extensions corresponding to the data designated with "#10 DATA" are deleted. **(Example)** All file names having four characters starting with A1 (ex., "A1X1") are deleted.
		#(5) (∗.XXX) ( )	Deletes all files with the extension ".XXX" in the designated directory of the ATA card.
DIR change and create (#6)	**Basic setting**	**# (6) (Name of directory in ATA card) ( )**	
	Relative designation	#(6) (. . /) ( )	Changes to one directory higher.
	Absolute designation	#(6) (/xyz) ( )	Changes to directory "xyz".
Rename (#7)	**Basic setting**	**# (7) (file name to be changed) (New file name)**	
	Single file designation	#(7) (n) (m)	Changes name of file n in ATA card to m. Changes extension corresponding to data designated in "#10 DATA".
		#(7) (s) (t)	Changes name of file s in ATA card to t. Changes extension corresponding to data designated in "#10 DATA".
All delete (#8)	Single file designation	#(8) (CLR) ( )	Deletes all contents of the ATA card.

(Note 1) Rename is valid only for files. Directories cannot be renamed.

(Note 2) "CLR" set for all delete is a password used to delete all contents of the ATA card.
All of the files and directories in the ATA card cannot be deleted with any other setting.

(Note 3) When carrying out delete or rename, if the extension is set in the setting area in addition to the file name, delete and rename will be executed to the set files regardless of the "#10 DATA" setting.

2.6.6.3 Input/Compare (#(1)/ #(2))

- When the input mode is selected, the files in the flash ATA card will be input to the NC memory as machining programs.
- When the compare mode is selected, the files in the flash ATA card will be compared with the machining programs in the NC memory.
- Designate the file name of the machining program to be input or compared.
- Files having a name containing characters other than numbers or having an extension can be input and compared as a program. Note that when inputting or comparing, a number with eight or less digits must be designated as the NC memory's machining program number.
  Batch designated files can also be input and compared.
- If the NC memory's open capacity reaches 0 during input, the data input up to that point will be registered as a program, and then the "E10 MEMORY OVER" will occur.
- When inputting fixed cycle programs (when base specifications parameter "#1166 fixpro" is valid), data is input with the file name from which the unique extension (.FIX) has been deleted as the program number.
- If the flash ATA card files and NC memory data differ, the error "E35 COMPARE ERROR" will occur.
  The data will appear at <IN/OUT DATA> and <COMP DATA>
  The comparison can be continued by pressing the INPUT key.
  The data will not appear if the comparison is proceeding without problem.
- Input and comparison end when reset is executed.
  If reset is executed during input, the data input up to that point will be valid.
- If an extension is not designated, an extension will be automatically assigned to the data designated with "#10 DATA", and the file will be read out.

(1) Example of setting input/compare

	Example of conditions	Setting
Example 1	Input file name "100" as machining program No. "200", or compare file name "100" with machining program No. "200".	# (1) (100) (200) For comparison, #(2)
Example 2	Input file name "PROG.PRG" as machining program No. "20", or compare file name "PROG.PRG" with machining program No. "20".	# (1) (PROG.PRG) (20) For comparison, #(2)
Example 3	Batch input/compare all files (without extension) in the current directory.	# (1) (∗) ( ) For comparison, #(2)
Example 4	Batch input/compare all files in the current directory having a 4-digit file name starting with 78. Files between 7800 and 7899 will be the target.	# (1) (78++) ( ) For comparison, #(2)
Example 5	When one file has several O No. lines. Input file "ALL.PRG" as the machining program No. "200". Assume that the file has the line "O100" and "O300".	# (1) (ALL.PRG) (200) 1st O No. line: This will become machining program "200" as designated. 2nd and following O No. Line: The O No. in the file is validated, and is displayed at the right of <IN/OUT DATA>. In this example, the machining programs "200" (from O100) and "300" (from O300) will be created.

(2) Precautions

(Note 1) If the NC memory's machining program number is not designated when inputting or comparing, the same No. as the file name will be used.
Note that this setting is valid only when the file name consists of only the numbers of eight or less digits.
An error will occur if the file name contains alphabetic characters or has an extension.

(Note 2) If a file with the same No. as a program existing in the NC memory is input, the error "E65 PROG. No. DUPLI" will occur.
Note that during batch input, a message confirming whether to overwrite will appear.

(Note 3) The first O No. line in the machining program is ignored.

(Note 4)
- When machining program is designated with "#10 DATA":
  If "PROG" is designated for the file name, the file name "PROG" (no extension) will be read out.
- If tool data or parameter data, etc., is designated with "#10 DATA":
  If "PROG" is designated for the file name, the file with a unique extension automatically assigned, such as "PROG.OFS" (tool data) or "PROG.PRM" (parameter data) will be read out.

(Note 5) Files batch input or compared with "*" or "+" are limited to files having a name of eight digits or lower numbers. Even when there is no extension, if the file name contains alphabetic characters, it cannot be batch input or compared with the above operations. (It will be ignored.)

(Note 6) If the program to be overwritten is running, the error "E21 PROGRAM RUNNING" will occur, and the batch input will end.

2.6.6.4 Output (#(3))

- Machining programs in the NC memory are output as flash ATA card files.
- The name of the file to be output is designated with eight or less alphanumeric characters.
- Machining programs can be output as files with extensions. Batch designated output is also possible.
- If the flash ATA card capacity is filled during the output, the data output up to that point will be registered as a file, and then the error "E113 MEMORY OVER" will occur.
- When outputting tool data, parameter, history data, maintenance data or fixed cycle programs, a unique extension will be automatically added to the designated file name.
- If reset is executed during the output, the output data up to that point will be valid.

(1) Example of output setting

	Example of conditions	Setting
Example 1	Output machining program No. "200" as file name "100".	# (3) (200) (100)
Example 2	Output machining program No. "20" as file name "FILE.PRG".	# (3) (20) (FILE.PRG)
Example 3	Batch output all machining programs in the NC memory as file "ALL.PRG".	# (3) (ALL) ( )
Example 4	Batch output all machining programs in the NC memory as file "ALPRG.PRG".	# (3) (ALL) (ALPRG.PRG)

(2) Precautions

(Note 1) If the file name is not designated when outputting, the same name as the machining program will be used.

(Note 2) The error "E111 FILE DUPLICATE" will occur if a machining program with the same No. as a file name existing in the flash ATA card is designated.
Note that during batch output, a message confirming whether to overwrite will appear.

(Note 3) When outputting the machining programs with an extension, if an extension unique to the data (.ofs, .prm, etc.) is used, the error "E01 SETTING ERROR" will occur.

(Note 4) Do not designate extensions for tool data or parameter data, etc., to which unique extension are automatically assigned. If an extension is assigned, and that designated extension differs from the automatically assigned extension, the error "E01 SETTING ERROR" will occur.

2.6.6.5 Continuous Output (#(4))

- Machining programs in the NC memory are continuously output as flash ATA card files. Single file designation and batch designation output are also possible.
- If reset is executed during the output, the output data up to that point will be valid.

(1) Example of continuous output setting

	Example of conditions	Setting
Example 1	Output data in the range of machining program No. "100" to "200" as file name "100" to "200".	# (4) (100) (200)

(2) Precautions

(Note 1) If the range is not designated, the files will be output with single file designation.
(Note 2) An error will occur if edit lock is ON.
(Note 3) If a machining program having the same No. as a file name in the flash ATA card is found in the designated range, a message confirming whether to overwrite will appear.

2.6.6.6 Delete (#(5))

- Files in the designated directory in the flash ATA card or the directory are deleted. Add "/" to the end to designate a directory.
  Batch designated delete is also possible.
- A message confirming the deletion process will appear.

(1) Example of delete setting

	Example of conditions	Setting
Example 1	Delete file name "100" from the current directory.	# (5) (100) ( )
Example 2	Batch delete files without extension from the current directory.	# (5) (∗) ( )
Example 3	Batch delete all files in the current directory regardless of whether an extension is assigned.	# (5) (∗.∗) ( )
Example 4	Batch delete files with the extension ".DAT" from the current directory.	# (5) (∗.DAT) ( )
Example 5	Batch delete all files having a 4-digit name starting with A1 from the current directory.	# (5) (A1++) ( )
Example 6	Delete the directory "DATA" and all data contained from the flash ATA card.	# (5) (DATA/) ( )

(Note) Refer to "2.6.6.9 All Delete" to delete all data in the flash ATA card.

2.6.6.7 Directory Change and Create (#(6))

- The directory in the flash ATA card is changed or created.

  When the designated directory name already exists:
  The directory is changed.
  The directory can be changed with either an absolute path or relative path designation.

  When the designated directory name does not exist:
  The message "CREATE? Y/N" will appear. Answer with Y or N in the 3rd setting area.
  Y : Creates a directory
  N : Does not create a directory

- In addition to the /(route) designation, a relative designation (../) is possible.

(1) Example of directory change and creation setting

	Example of conditions	Setting
Example 1	Change from directory "/" to directory "/A".	# (6) (A) ( )
Example 2	Newly create directory "AB".	# (6) (AB) ( )

(2) Precautions

(Note 1) The numbers 0 to 9 and alphabet characters A to Z can be used for the directory name.

(Note 2) Designate the directory path and name with 12 or less characters from /(route) to the end. 13 or more characters cannot be set.

Directory path and name / 1 2 3 4 5 / 6 7 8 / 9
12 or less characters

(Note 3) One directory name can have up to 8 characters.

2.6.6.8 Rename (#(7))

- The file names in the flash ATA card can be renamed.
- File names with an extension can also be renamed.
- If the file is renamed to the same name as an existing file, the error "E111 FILE DUPLICATE" will appear, and the file will not be renamed.
- If a file that does not exist is designated for renaming, the error "E115 FILE NOT FOUND" will occur.
- Only files can be renamed. Directories cannot be renamed.

(1) Example of rename setting

	Example of conditions	Setting
Example 1	Rename file name "100" to "200".	# (7) (100) (200)
Example 2	Rename file name with extension "100.DAT" to "200.DAT".	# (7) (100.DAT) (200.DAT)

2.6.6.9 All Delete (#(8))

- All files and directories in the flash ATA card are deleted.
  A message confirming the deletion will appear.

(1) Setting all delete

	Example of conditions	Setting
Example 1	Delete all files and directories in the flash ATA card.	# (8) (CLR) ( )

2.6.7 Device Connection

2.6.7.1 Connection of Tape Reader, Tape Puncher, Printer, FLD

Refer to Appendix "RS-232C I/O Device Parameter Setting Examples and Cable Connections" for the parameter settings of each I/O device.

2.6.7.2 Connection of Flash ATA Card

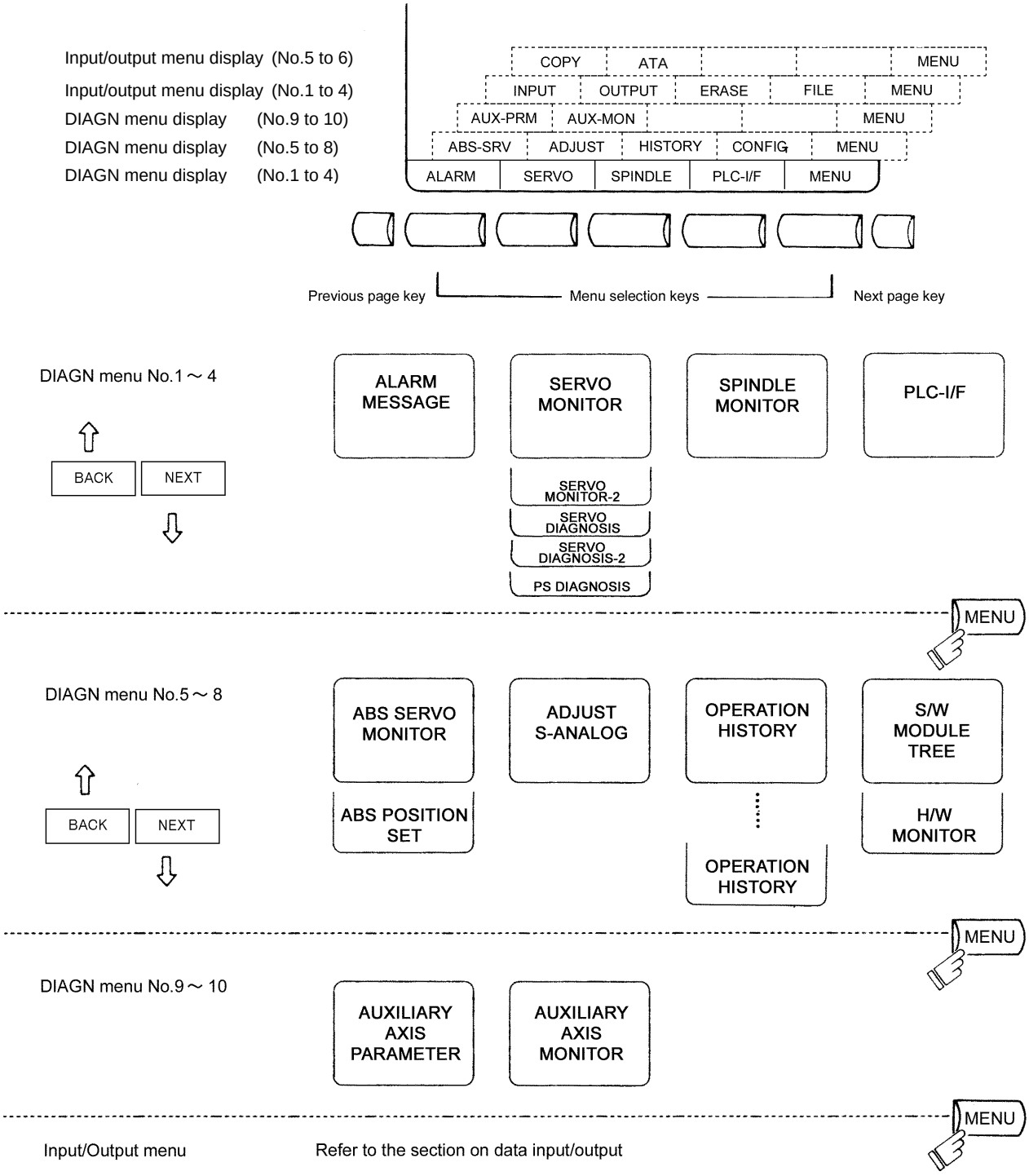
Refer to “C6/C64 Connection and Maintenance Manual”(BNP-B2255).

⚠ CAUTION

Always turn the power OFF before connecting/disconnecting the input/output device cable. Failure to do so could damage the I/O device and control unit.

2.7 Diagnosis

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



2.7.1 Alarm Message

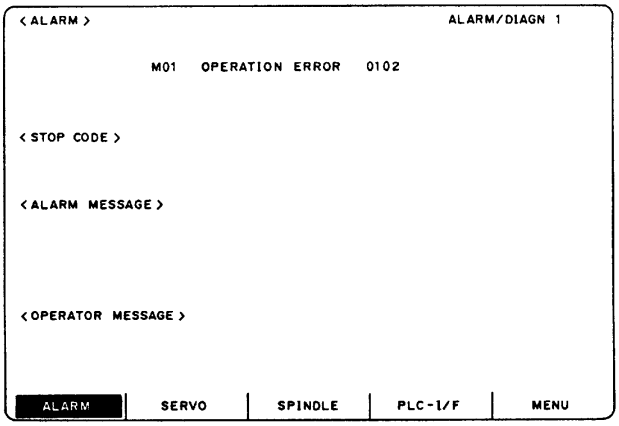
When the menu key ALARM is pressed, the ALARM/DIAGN screen is displayed.

(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.

(3) Alarm message
The alarm messages specified by the user PLC (built-in) are displayed.

(4) Operator message
The operator messages specified by the user PLC (built-in) are displayed.



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] [CAN C.B.] keys.

☞

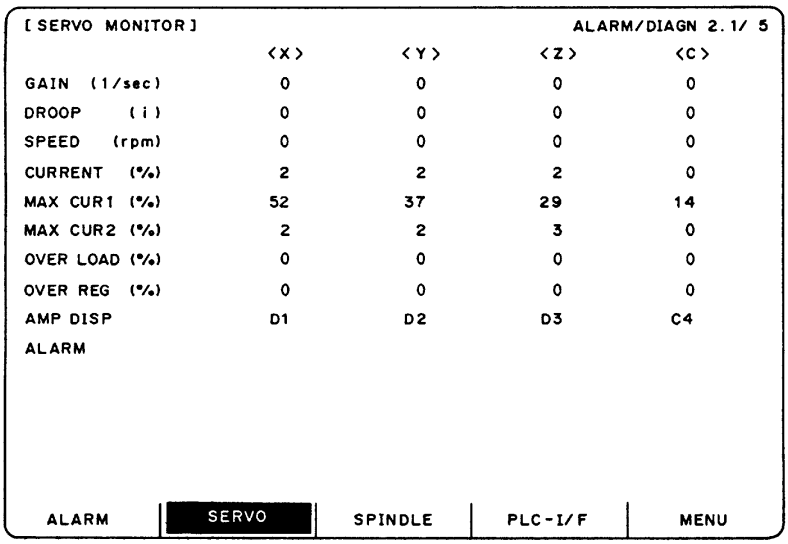
The display will return to the normal current alarm display screen. This is the same for changing over to other screens once.

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

2.7.2 Servo Monitor

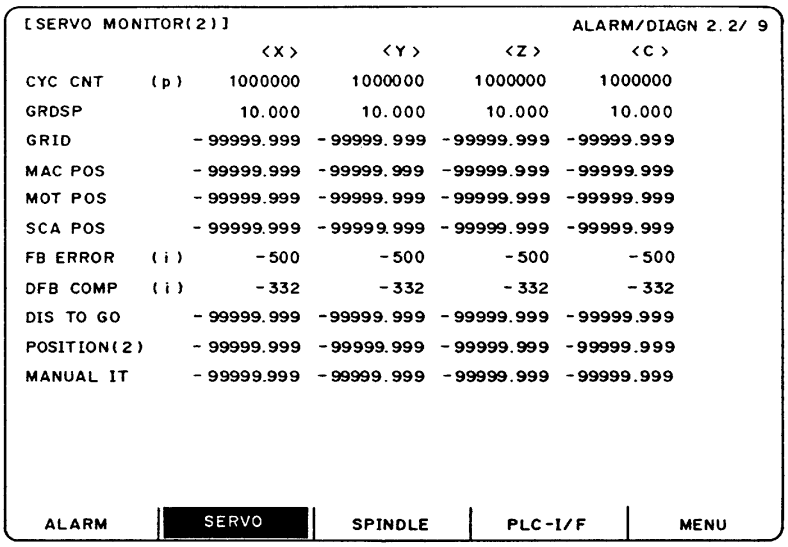
When the menu key SERVO is pressed, the SERVO MONITOR screen is displayed indicating servo status during operation.

2.7.2.1 Servo Monitor



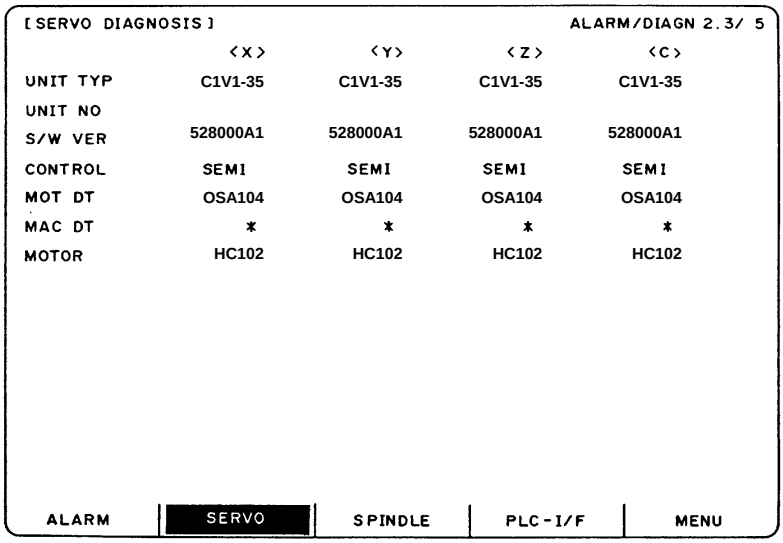
Data	Display unit	Explanation
GAIN	1/s	The position loop gain is displayed. The position loop gain is: $\frac{\text{feedrate (mm/s)}}{\text{tracking delay error (mm)}}$
DROOP	i	An error of the actual machine position to the command position is called droop. This error is proportional to the command speed value.
SPEED	r/min	Actual rotation speed of motor.
CURRENT	%	The motor current is displayed in terms of continuous current during stalling.
MAXCUR1	%	The current FB ratio to the current limit is shown with a percentage. 1) <Left> The peak value is constantly sampled, and the value is updated every second. 2) <Right> The maximum value of the current FB peak sampled after the power was turned on is constantly displayed.
MAXCUR2	%	The maximum value of the current FB peak sampled in the last two seconds is constantly displayed.
OVER LOAD	%	Data used to monitor overload.
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.
ALM	—	The alarms and warnings other than the amplifier display are displayed.

2.7.2.2 Servo Monitor (2)



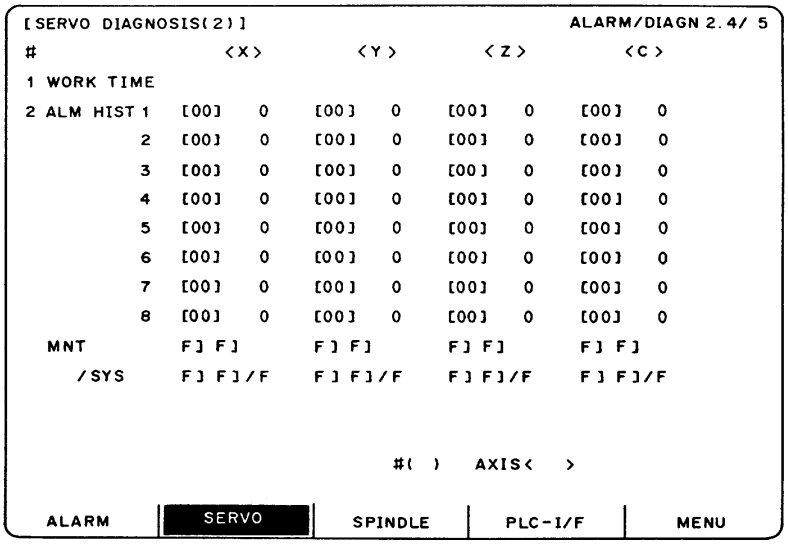
Data	Display unit	Explanation
CYC CNT	Pulse	The position within one rotation of the encoder detector is displayed. The position uses the grid point value as 0, and will display the position within one rotation within the range of 0 ~ RNG (movement unit) ∗1000.
GRDSP	Command unit	The grip interval for reference point return is displayed.
GRID	Command unit	The length from the dog-off to grip point when dog-type reference point return is executed is displayed. The grid mask amount is not included.
MAC POS	Command unit	Basic machine coordinate system position
MOT POS	Command unit	The feedback position of the speed detector is displayed.
SCA POS	Command unit	The feedback position of the position detector is displayed.
FB ERROR	i	The error of the motor end FB and machine end FB is displayed.
DFB COMP		This is not used.
DIS TO GO	Command unit	The remaining movement distance of one block is displayed.
POSITION (2)	Command unit	The value of the tool compensation amount subtracted from the current value is displayed.
MANUAL IT	Command unit	The amount of interrupt movement in the manual absolute off state is displayed.

2.7.2.3 Servo Diagnosis



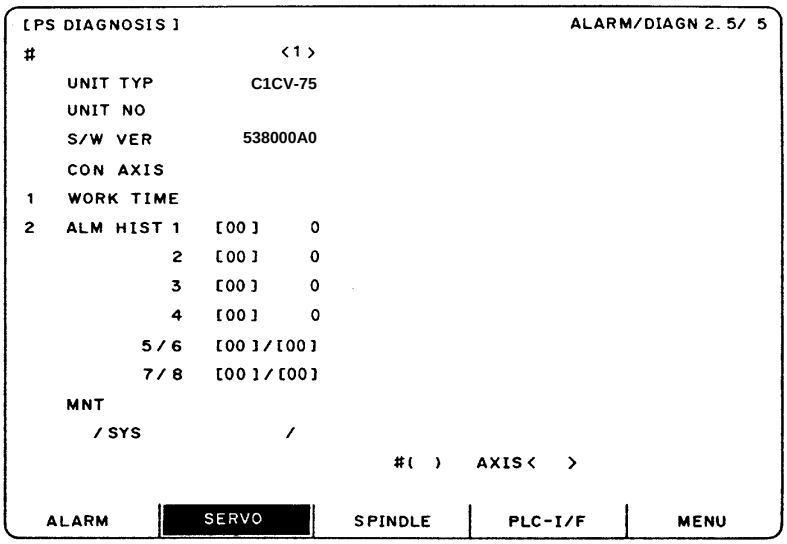
Display item	Explanation
UNIT TYP	This is the servo driver type.
UNIT NO	This is the servo driver manufacturing number.
S/W VER	This is the servo side's software version.
CONTROL	SEMI: semi-closed loop
MOT DT	This is the motor detector type.
MAC DT	This is the machine end detector type. If the SEMI control method is used, ∗ will display.
MOTOR	This is the motor type.

2.7.2.4 Servo Diagnosis (2)



Display item	Explanation
WORK TIME	This is the cumulative ready on time.
ALM HIST1	The No. of the servo alarm that has occurred is displayed.
MNT/SYS	Maintenance history NVRAM status

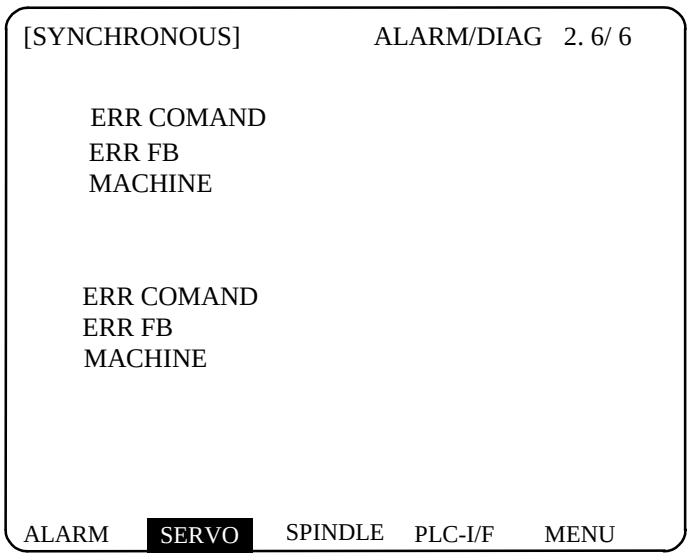
2.7.2.5 Power Supply Diagnosis



Data	Display unit	Explanation
UNIT TYP	—	This is the power supply unit type.
UNIT NO	—	This is the power supply unit manufacturing number.
S/W VER	—	This is the software version.
CON AXIS	—	This is the driver connecting each power supply unit.
WORK TIME	Time	This is the cumulative ready on time.
ALM HIST	—	This is the No. history of the servo alarms that have occurred.
	Time	This is the occurrence time history of the servo alarms that have occurred.
MNT/SYS	—	Maintenance history
	—	NVRAM status

2.7.2.6 Synchronous Error

The various data related to the synchronous error is monitored.

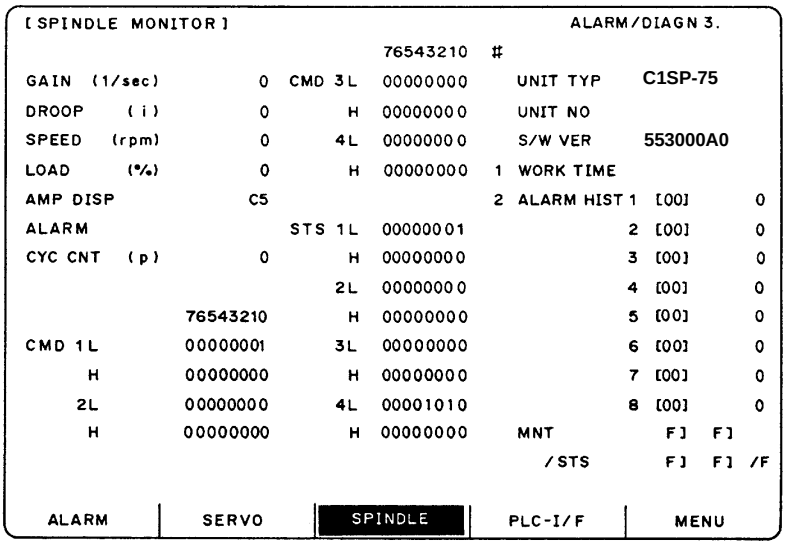


Display item	Explanation
ERR COMMAND	This is the deviation of the slave axis machine position in respect to the master axis. This displays the error of the command value to the servo control section before compensating the pitch error, incremental position, or backlash. If this error occurs, the parameters 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.

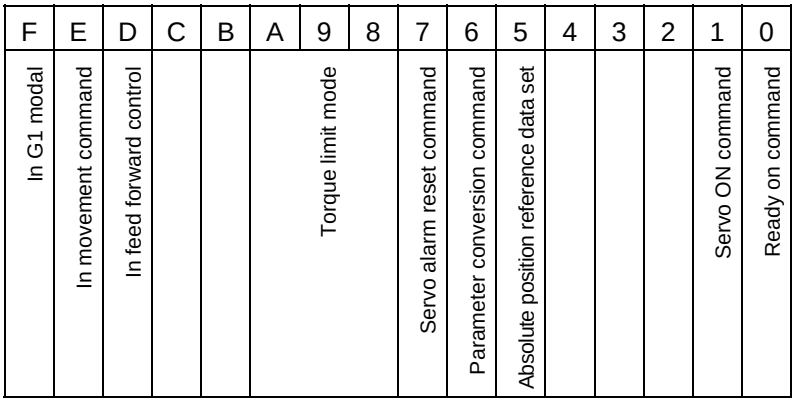
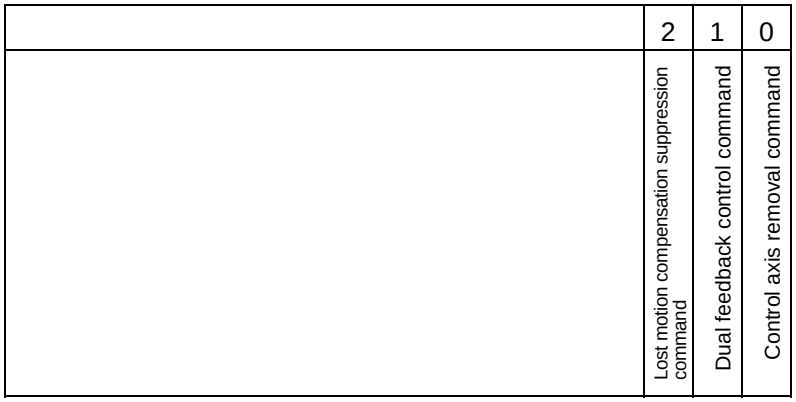
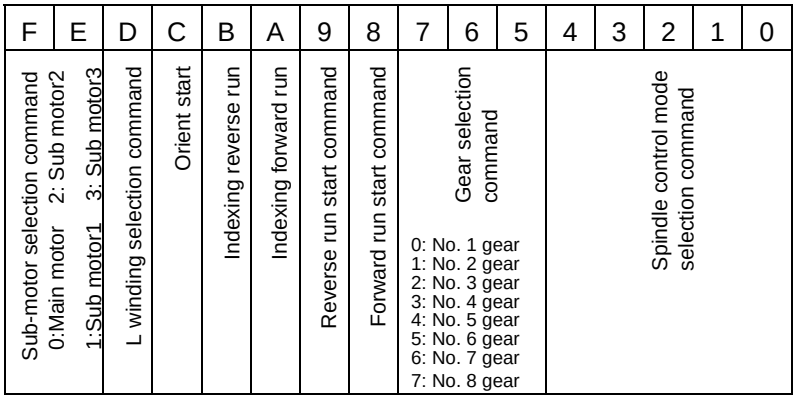
(Note 1) The error offset amount “Δ” is made when the operation method is changed because R932 R1032 register is changed.
Note that if the zero point has not established, the error offset amount “Δ” is made immediately after the zero point is established.

(Note 2) The axis names in this screen are displayed only during the synchronous or independent operation.
The value of “ERR COMMAND”, “ERR FB” and “MACHINE” are displayed only during the synchronous operation. Thus, these became “0.000” during the asynchronous or independent operation.

2.7.3 Spindle Monitor



Display item	Explanation
GAIN	Spindle position loop gain
DROOP	Position deflection amount
SPEED	Motor speed
LOAD	Motor load
AMP DISP	7-segment display of driver
ALARM	Alarms other than the amplifier display (max. 3 alarms)
CYC CNT	Angle data from detection reference position (Z phase)
CMD	Control input signal from control unit
STS	Control output signal from driver
UNIT TYP	This is the spindle type.
UNIT NO	This is the spindle manufacturing number.
S/W VER	This is the spindle side software version.
WORK TIME	The cumulative ready on time is displayed.
ALM HIST	The No. of the spindle alarm that occurred is displayed.
MNT	Maintenance history
/SYS	Status

Display item	Explanation
Control input 1	Control input signal from control unit 
Control input 2	Control input signal from control unit 
Control input 3	Control input signal from control unit 

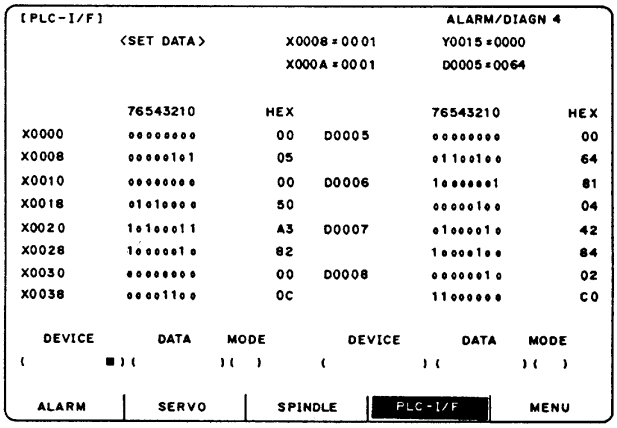
<table>
<tr><th>Display item</th><th>Explanation</th></tr>
<tr><td>Control input 4</td><td>Control input signal from control unit
<table><tr><td></td></tr><tr><td></td></tr></table></td></tr>
<tr><td>Control output 1</td><td>Driver control output signal
<table>
<tr><td>F</td><td>E</td><td>D</td><td>C</td><td>B</td><td>A</td><td>9</td><td>8</td><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td></tr>
<tr><td>In current limit</td><td>In in-position</td><td>Z-phase passed</td><td></td><td></td><td colspan="3">Torque limit mode</td><td>In servo alarm</td><td>In parameter conversion</td><td>Absolute position reference data set complete</td><td>In driver warning</td><td></td><td></td><td>In servo ON</td><td>In ready on</td></tr>
</table></td></tr>
<tr><td>Control output 2</td><td>Driver control output signal
<table>
<tr><td></td><td>2</td><td>1</td><td>0</td></tr>
<tr><td></td><td>In lost motion compensation suppression</td><td>In dual feedback control</td><td>In control axis removal</td></tr>
</table></td></tr>
</table>

<table>
<tr><th>Display item</th><th>Explanation</th></tr>
<tr><td>Control output 3</td><td>Driver control output signal
<table>
<tr><th>F</th><th>E</th><th>D</th><th>C</th><th>B</th><th>A</th><th>9</th><th>8</th><th>7</th><th>6</th><th>5</th><th>4</th><th>3</th><th>2</th><th>1</th><th>0</th></tr>
<tr><td colspan="2">In sub-motor selection
0:Main motor 2: Sub motor2
1:Sub motor1 3: Sub motor3</td><td>In L winding selection</td><td>In orient start</td><td>In index reverse run</td><td>In index forward run</td><td>In reverse run start</td><td>In forward run start</td><td colspan="3">In gear selection
0: No. 1 gear
1: No. 2 gear
2: No. 3 gear
3: No. 4 gear
4: No. 5 gear
5: No. 6 gear
6: No. 7 gear
7: No. 8 gear</td><td colspan="4">In spindle control mode selection</td></tr>
</table></td></tr>
<tr><td>Control output 4</td><td>Driver control output signal
<table>
<tr><th>F</th><th>E</th><th>D</th><th>C</th><th>B</th><th>A</th><th>9</th><th>8</th><th>7</th><th>6</th><th>5</th><th>4</th><th>3</th><th>2</th><th>1</th><th>0</th></tr>
<tr><td>No.2 in-position</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td>Orient completed</td><td>Zero speed detection</td><td>Speed reached</td><td>Speed detection</td><td>Current detection</td></tr>
</table></td></tr>
</table>

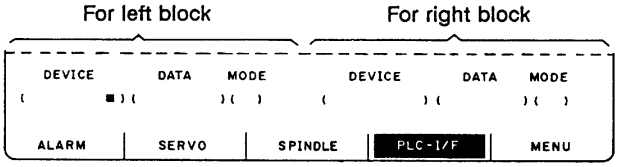
2.7.4 PLC Interface Diagnosis

When the menu key [PCL-I/F] is pressed, the 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 ( )

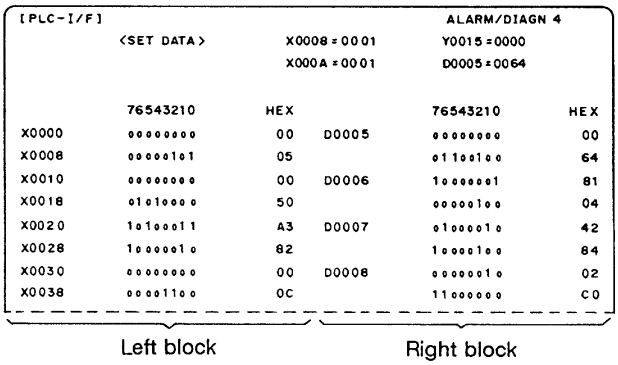
To forcibly define PLC data, set data corresponding to the setup device number. Set 1 or 0 for bit data. Set hexadecimal (HEX) data for byte data.

MODE ( )

Specify the type of forcible definition. 1: Single-shot type
2: Modal type

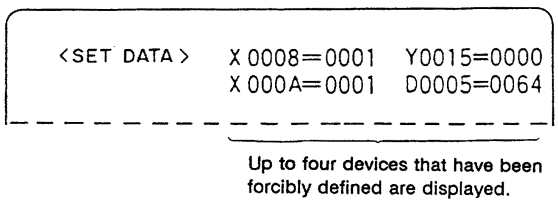
(2) Device data display area

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.

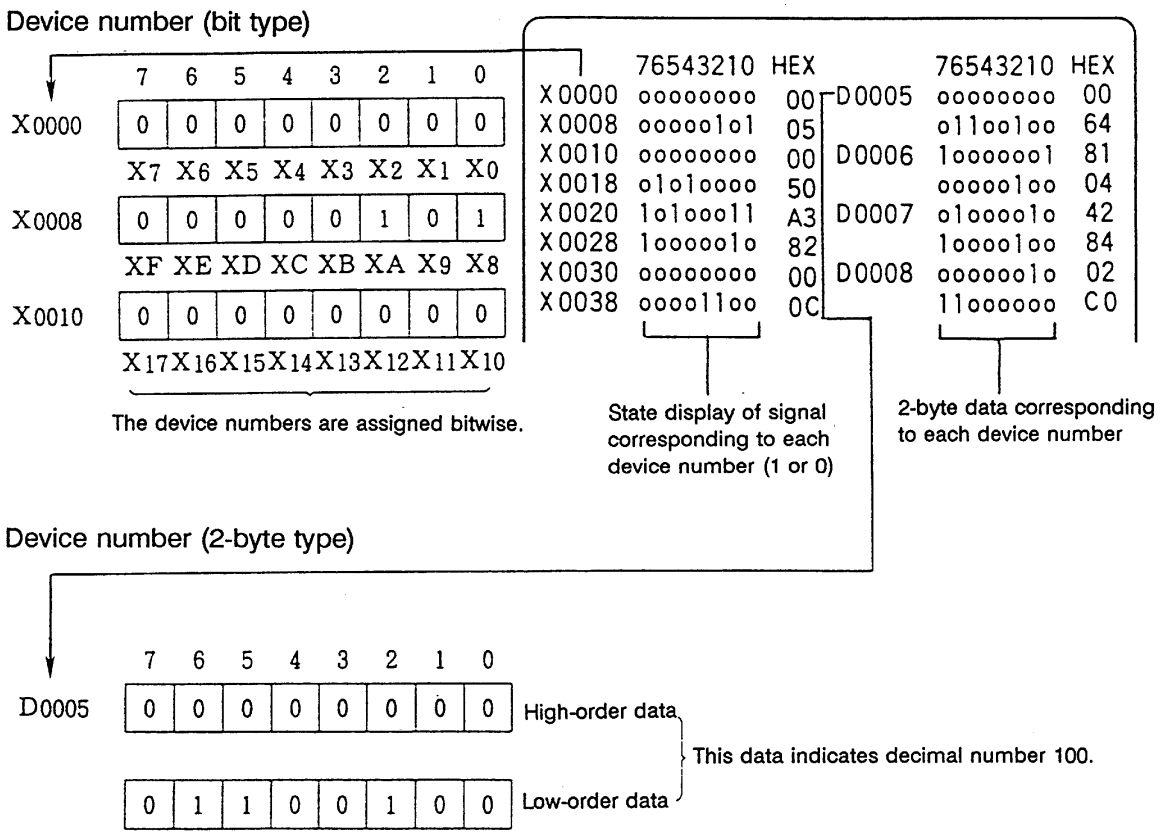
Device	Device range		Units	Details
X*	X0 to XAFF	2816 points	1-bit	Input signals to the PLC. Machine input, etc.
Y*	Y0 to YE7F	3712 points	1-bit	Output signals from the PLC. Machine output, etc.
M	M0 to M8191	8192 points	1-bit	For temporary memory
L	L0 to L255	256 points	1-bit	Latch relay (Backup memory)
F	F0 to F127	128 points	1-bit	For temporary memory. Alarm message interface
SB	SB0 to SB1FF	512 points	1-bit	Special relay for links
B	B0 to B1FFF	8192 points	1-bit	Link relay
SM*	SM0 to SM127	128 points	1-bit	Special relay
V	V0 to V255	256 points	1-bit	Edge relay
SW	SW0 to SW1FF	512 points	16-bit	Special register for links
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
	T104 to T143	40 points	1-bit/16-bit	10ms unit timer (Fixed timers)
	T144 to T239	96 points	1-bit/16-bit	100ms unit timer (Fixed timers)
	T240 to T255	16 points	1-bit/16-bit	100ms incremented timer (Fixed timers)
	T0000 to T0255	256 points	1-bit	T1: Timer coil
	T1000 to T1255	256 points	1-bit	T0: Timer contact
	T2000 to T2255	256 points	16-bit	TS: Timer setting value
	T3000 to T3255	256 points	16-bit	TA: Timer current value
C	C0 to C23	24 points	1-bit/16-bit	Counter
	C24 to C127	104 points	1-bit/16-bit	Counter (Fixed counters)
	C0000 to C0127	128 points	1-bit	C1: Counter coil
	C1000 to C1127	128 points	1-bit	C0: Counter contact
	C2000 to C2127	128 points	16-bit	CS: Counter setting value
	C3000 to C3127	128 points	16-bit	CA: Counter current value
D	D0 to D8191	8192 points	16-bit/32-bit	Data register
R*	R0 to R8191	8192 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 Z13	14 points	16-bit	Address index
N	N0 to N7			Master control's nesting level
P*	P0 to P255 P360 to P379			Conditional jump, subroutine call label
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.

(Note 3) Refer to “C6/C64/C64T PLC INTERFACE MANUAL” for the assignments of the PLC interface signals.

(5) How to read device numbers and display data



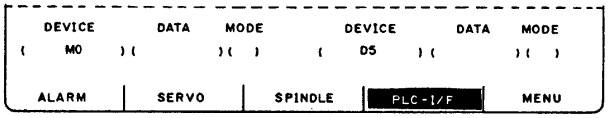
2.7.4.2 PLC Device Data Display

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.

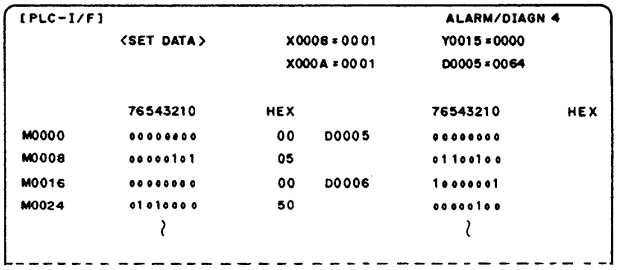
(1) Display device number setting
Set the device number in DEVICE ( ). If a different device number or device address is set in the right DEVICE ( ), the specified device numbers are displayed in the left and right halves of the screen from the display area top to bottom.

(Example 1)

Set M0 in left DEVICE ( ) and D5 in right DEVICE ( ).

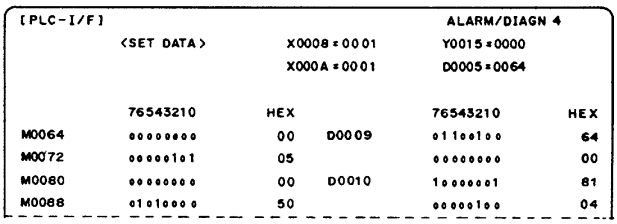


Press the INPUT key.



(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)

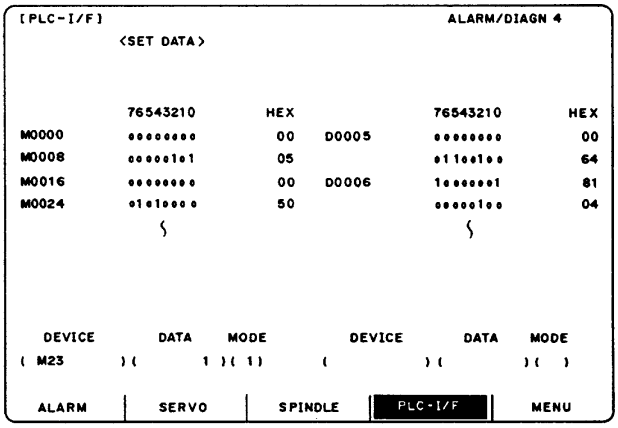
This function is used to forcibly define signals to check the PLC functions.

(1) Mode setting

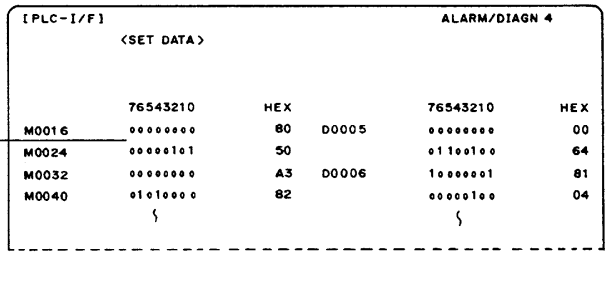
Set the device number to be forcibly set in DEVICE ( ), definition data in DATA ( ), and 1 in MODE ( ), then press the INPUT key. The setup data is processed and forcibly defined at the top of one cycle of user PLC.

(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)

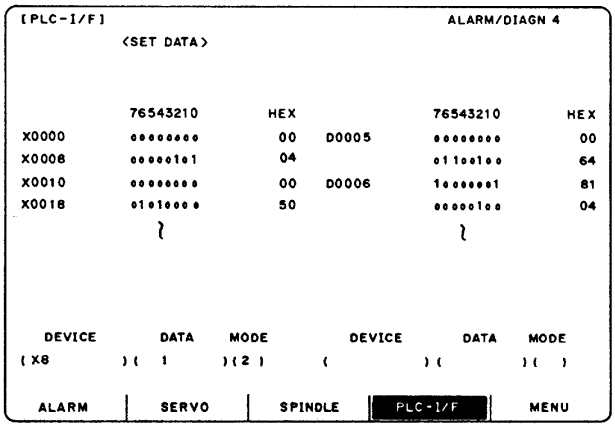
This is PLC interface signal forcible definition of modal type. Once it is set, it is held until canceled.

(1) Mode setting

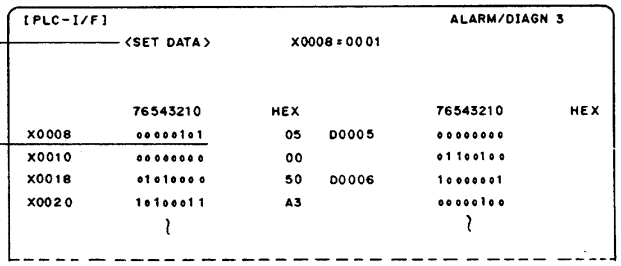
Set the device number to be forcibly defined in DEVICE ( ), definition data in DATA ( ), and 2 in MODE ( ), then press the INPUT key. The data is processed and forcibly defined.

(Example 3)

Set X8 in DEVICE ( ),
1 in DATA ( ), and
2 in MODE ( ).



Press the INPUT key.



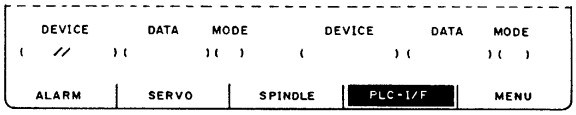
A maximum of four sets of forcibly defined device numbers and numeric data are displayed.

X0008 is displayed at the beginning and bit 0 changes to 1.

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.)

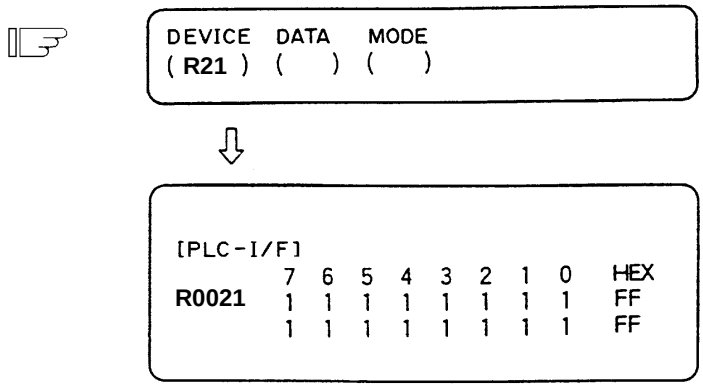
Forcibly defined data is also canceled by turning on/off power.

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.

(1) Check using the DIAGN screen

Set R21 in the DEVICE ( ) and then press [INPUT] key.



R0021 (The cause of the emergency stop)

Bit	F	E	D	C	B	A	9	8	7	6	5	4	3	2	1	0
State	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1

(0: EMG stop 1: Normal)

- Bit 0: Internal PLC software stopped
- Bit 1: Internal PLC no hardware
- Bit 2: Internal PLC preparation incomplete
- Bit 3: Internal PLC communication error
- Bit 4: Internal PLC emergency stop output
- Bit 6: Internal PLC software emergency stop output Y427 is "1"
- Bit B: User PLC illegal code found
- Bit C: PLC high-speed process error
- Bit D: Door interlock, dog/OT arbitrary assignment device illegal
- Bit E: Spindle drive unit emergency stop output
- Bit F: Servo drive unit emergency stop output

(2) Error display

EMG EMERGENCY ****
Error items

Error items	Details	Cause
PLC	The user PLC has entered the emergency stop state during the sequence process.	• Investigate and remove the cause of the user PLC emergency stop.
EXIN	The emergency stop input signal is significant (open).	• Cancel the emergency stop input signal. • Check the wiring to see if any wiring is missing.
SRV	An alarm occurred in the servo system causing an emergency stop.	• Investigate and remove the cause of the servo alarm.
STOP	The user PLC (ladder sequence) is not running.	• Check if the rotary switch CS1 (left switch) on the top of the controller front panel is set to 1. • Check if the PLC edit file save screen (onboard function) [4RUN/SP] (run/stop) switch is turned ON.
SPIN	Spindle drive unit not mounted. The spindle drive unit is not mounted.	• Cancel the causes of the other emergency stop. • Check emergency stop signal and ready signal input in the spindle amplifier.
PC_H	High-speed PC processing abnormal	• Check the sequence program. (To stop monitoring the high-speed PC processing temporarily, set "1" in #1219 aux03 bit1. Disable the monitoring function only as a temporary measure.)
PARA	Setting of the door open II fixed device is illegal. The dog signal random assignment parameter setting is illegal.	• Specify the #1155 DOOR_m and #1156 DOOR_s parameters correctly. (When the door open II fixed device is not used, set #1155 DOOR_m and #1156 DOOR_s to "100".) • Correctly set the #2073 zrn_dog, #2074 h/w_ot+, #2075 h/w_ot- and #1226 aux10 bit 5 parameters.
XTEN	External PLC H/W is not found. The external PLC card is not mounted.	• Check that the external PLC card is mounted. • Change the external PLC card.
WAIT	External PLC is not ready. The External PLC Ready signal is not turned ON.	• Reset and start the external PLC. • Turn the NC power OFF and then ON. • Check that the external PLC card is mounted. • Change the external PLC card. • Change the NC card.
LINK	External PLC communication error occurred. A communication error occurred between the external PLC and NC.	
LAD	The user PLC (ladder sequence) has an illegal code.	• Check the user PLC (ladder sequence) to see if it uses illegal device numbers or constants.

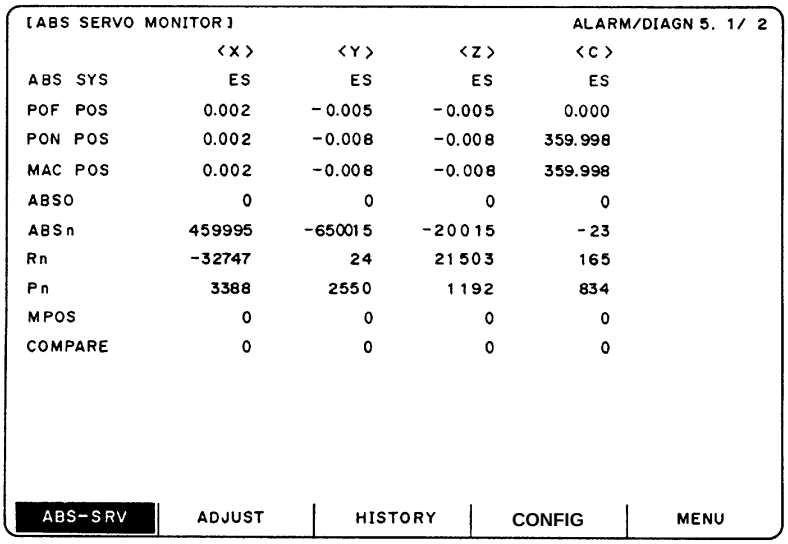
When the “Error item” is “LINK”, the cause of the error will be saved into the R register as follows.

R register	Details	
R80	The contents of the latest errors. (Cleared when the error is resolved.) Bit 0 : Emergency B Bit 1 : Emergency C Bit 2 : Emergency D Bit 3 : Emergency E Bit 4 : Emergency F	• The external PLC is not ready. • Data is not input from the external PLC. • Data is not read out by the external PLC. • Data is not written into the NC correctly. • 2-port RAM is not cleared when the PC_OK signal is turned ON.
R81	The contents of the latest errors. (Retained until the NC power is turned ON next.)	• This is cleared when the NC power is turned ON again.
R82	The control signal; External PLC -> NC	• Writing from PC to NC completion flag.
R83	The control signal; NC -> External PLC	• Writing from NC to PC completion flag.
R84	The contents of the latest errors.	• This is not cleared even if the NC power is turned ON again.

2.7.5 Absolute Position Monitor

2.7.5.1 ABS Servo Monitor

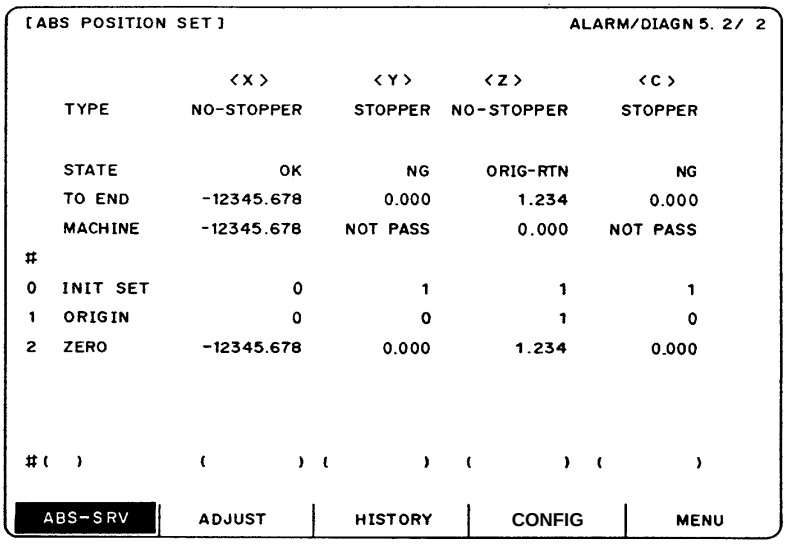
The servo state in the absolute position detection system is displayed. This can be used to confirm each detector data for the current machine value.

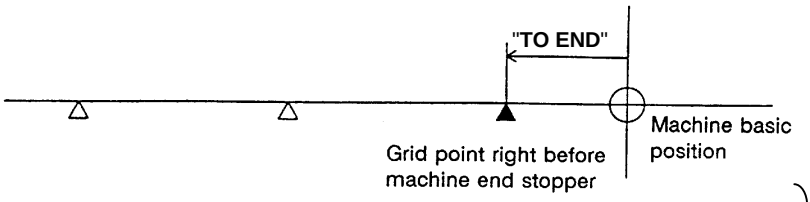
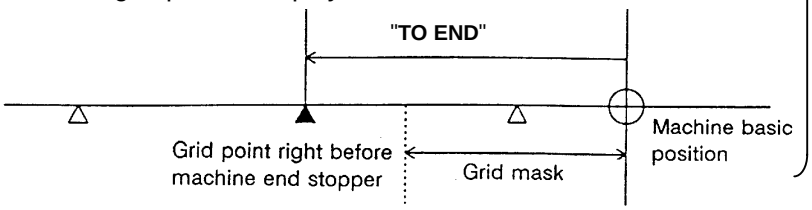


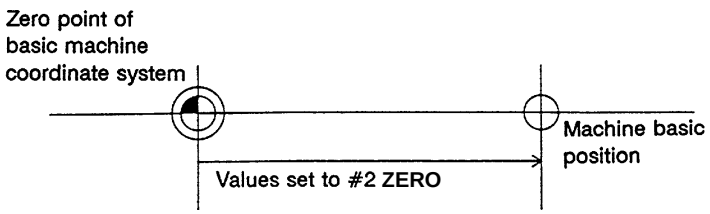
Data	Display unit	Explanation
ABS SYS		The status of the absolute position detection system on the servo side is displayed. ES : Semi-closed encoder ESS : Semi-closed high-speed serial encoder INC : Incremental
POF POS	Command unit	The absolute position when the power is turned off is displayed.
PON 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.
COMPARE	0.5µm	When the absolute position compare command is executed, the difference of the detector's absolute position and the current position for control is displayed.

2.7.5.2 Absolute Position Initialization

Pressing the menu key [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.



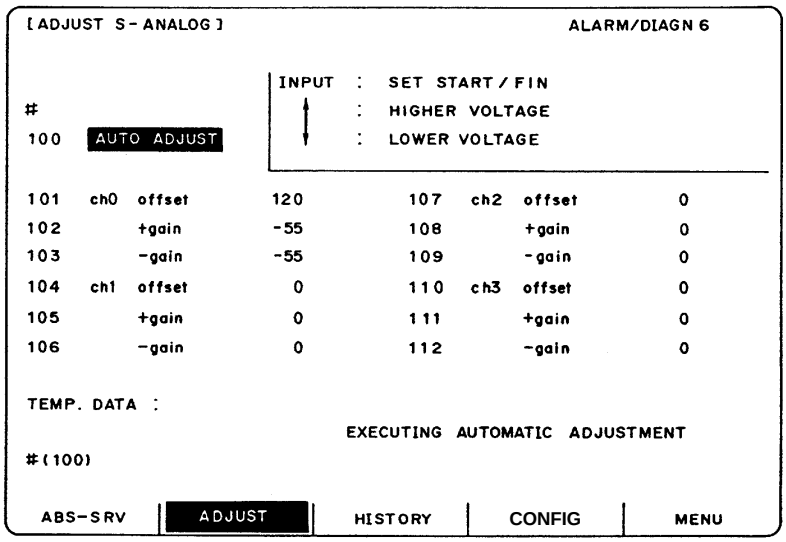
Display item	Explanation
TYPE	The position detection system selected by the absolute position parameter is displayed. INC. : Incremental system DOG : Dog type absolute position detection STOPPER : Dogless absolute position detection machine end stopper method NO STOPPER : Dogless absolute position detection 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 Chapter 3, Section 4 "Absolute Position Detection System".

2.7.6 Adjustment

Pressing the menu key ADJUST displays the ADJUST S-ANALOG screen.



2.7.6.1 Adjustment Preparation

(1) The PLC must be stopped before adjusting the S-analog output.
Stopping with rotary switch
Set the rotary switch: CS2 (on right side looking from front) on the top of the unit to 1.

(2) Select the ADJUST S-ANALOG screen.
Press the DIAGN IN/OUT, MENU and ADJUST keys.

2.7.6.2 Automatic Analog Output Adjustment

Select #100 to execute automatic adjustment.
→ "AUTO ADJUST" will be highlighted, and the message "ADJUST EXECUTION" will display.
Automatic adjustment will execute offset adjustment and +gain adjustment for the remote I/O analog outputs (max. 4ch) connected to the main unit. The –gain will be handled as the +gain.
After adjustment is completed, the message "ADJUST COMPLETE" will display, and the "AUTO ADJUST" item will return to the normal display.

2.7.6.3 Adjustment Procedure

1) Offset adjustment (Ex. ch0)
   (a) Select #101.
   → "ch0 offset" will be highlighted, indicating that it is being adjusted.
   The current setting value will display in the "TEMP.DATA" column.
   (b) Monitor the analog output voltage.
   (c) Adjust the monitor voltage to 0V ± 3.5mV. (Setting range: –999~999)
   For a positive voltage, press the [↓] key. The adjustment value will decrement in units of 100 when [SHIFT] [↓] are pressed.
   For a negative voltage, press the [↑] key. The adjustment value will increment in units of 100 when [SHIFT] [↑] are pressed.
   Press [INPUT] after completing the adjustment.
   → The TEMP DATA will be saved as the setting value and the adjustment will be validated.
   The "ch0 offset" display will return to the normal display.

2) +gain adjustment (Ex. ch0)
   (a) Select #102.
   → "+gain" will be highlighted, indicating that it is being adjusted.
   (b) The current setting value will display in the "TEMP.DATA" column.
   (c) Monitor the analog output voltage.
   Adjust the monitor voltage to 10V ± 3.5mV. (Setting range: –999~999)
   For 10V or more, press the [↓] key. The adjustment value will decrement in units of 100 when [SHIFT] [↓] are pressed.
   For 10V or less, press the [↑] key. The adjustment value will increment in units of 100 when [SHIFT] [↑] are pressed.
   (d) Press [INPUT] after completing the adjustment.
   → The TEMP DATA will be saved as the setting value and the adjustment will be validated.
   The "+gain" display will return to the normal display.

3) –gain adjustment (Ex. ch0)
   (a) Select #103.
   → "–gain" will be highlighted, indicating that it is being adjusted.
   The current setting value will display in the "TEMP.DATA" column.
   (b) Monitor the analog output voltage.
   (c) Adjust the monitor voltage to –10V ± 3.5mV. (Setting range: –999~999)
   For –10V or more, press the [↓] key. The adjustment value will decrement in units of 100 when [SHIFT] [↓] are pressed.
   For –10V or less, press the [↑] key. The adjustment value will increment in units of 100 when [SHIFT] [↑] are pressed.
   (d) Press [INPUT] after completing the adjustment.
   → The TEMP DATA will be saved as the setting value and the adjustment will be validated.
   The "–gain" display will return to the normal display.

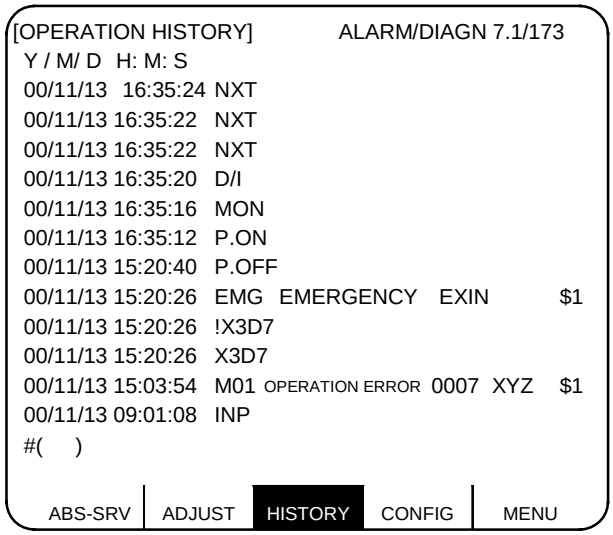
2.7.6.4 Parameter Input/Output

The settings #101 to #112 of this function can be input and output as parameters.

# number	Name	Parameter No.	Data range (decimal)
#101	ch0 offset	N10101	–999~999
#102	+gain	N10102	–999~999
#103	–gain	N10103	–999~999
#104	ch1 offset	N10104	–999~999
#105	+gain	N10105	–999~999
#106	–gain	N10106	–999~999
#107	ch2 offset	N10107	–999~999
#108	+gain	N10108	–999~999
#109	–gain	N10109	–999~999
#110	ch3 offset	N10110	–999~999
#111	+gain	N10111	–999~999
#112	–gain	N10112	–999~999

2.7.7 Operation History

Pressing the menu key [HISTORY] displays the OPERATION HISTORY screen. This function keeps tracing operation information, which is useful for failure diagnosis.
The information that can be saved is the date of occurrence, key history, alarm history and the input signal change data. This data is also backed up when the power is turned OFF.
The latest data is shown on the top line of the screen, and the oldest data is shown on the bottom of the screen.
Up to twelve history data items can be displayed on one page. The number of screen pages depends on the volume of history data. There can be up to 173 pages.



The history page is changed with the [NEXT] and [BACK] keys. Each time the [NEXT] key is pressed, the page will appear in order of older data. Each time the [BACK] key is pressed, the page will appear in order of newer data.

(1) Details of operation history

The operation history includes the key history, alarm history and history of input/output signal changes.

<Correspondence of each key and key history>

Key classify	Operation key	Key history
Function select key	MONITOR	MON
	TOOL PARAM	T/P
	EDIT MDI	E/M
	DIAGN IN/OUT	D/I
	SFG	SFG
	F0	F0
Menu keys	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 that are not registered		???
Data setting key	Delete	DEL
	Add	INS
	Clear block	C.B
	Shift + Clear block	CAN
	CALCULATE	CAL

Key name	Operation key	Key history
Cursor movement keys	↑	CRU
	↓	CRD
	←	CRL
	→	CRR
	Upper tab	TBU
	Lower tab	TBD
	Left tab (<<)	TBL
	Right tab (>>)	TBR
Alphabet key	Alphabet (A to Z)	A to Z
Number key	Number (0 to 9)	0 to 9
Symbol key	Symbol	
	(+, -, *, /, .)	+, -, *, /, .
	(,, (,), [,])	,, (, , [,]
	(EOB, =, #, SP, $)	;, =, #, SP, $

<Correspondence of state of input/output signal changes and input/output signal change history>

State of input/output signal changes	Input/output signal change history
Input signal change (ON → OFF)	Y410 to YE7F
Input signal change (OFF → ON)	!Y410 to !YE7F
Output signal change (ON → OFF)	X410 to XAFF
Output signal change (OFF → ON)	!X410 to !XAFF
Simultaneous input	Y410+ to YE7F+
Simultaneous output	X410+ to XAFF+
Power ON	P.ON
Power OFF	P.OFF

If two or more input/output signals change at the same timing, the change will be displayed with a "+", excluding the latest change.

(Example)
X002
X001+
X000+

This indicates that three signals simultaneously changed from ON to OFF.

(2) Operation history date, time display and setting

In the operation history, the year starts from 1970 and is valid for 100 years. The year is indicated with the last two digits. The date and time are set with the TIME screen (page 3 of the [MONITOR] - COMMAND screen. "00" is set for the year 2000, and "10" is set for the year 2010.

(Note) After setting the date and time, always clear the operation history and history occurrence time. If these are not cleared, the history data in the NC will not be displayed in the correct order.

(3) Suppressing the operation history function

Data is kept updated according to changes in NC alarms signals and key inputs. If it is desired to suspend the operation history function and retain the current data, enter "I" in # ( ) in the setting area and press the INPUT key.

This will suspend the updating of the operation history data.
To resume the operation history function, enter "P" in # ( ) in the setting area and press the INPUT key.

(4) Clearing operation history data

When the SHIFT, CAN C.B. and INPUT keys are pressed, all of the operation history data and occurrence time data will be cleared. The entire display will be cleared, and the screen will have only one page.

(5) 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 if the OPERATION HISTORY screen is refreshed by pressing the same menu key again, or by opening another screen and then returning to the OPERATION HISTORY screen.
- If the time that the operation histories occur is the same, the histories will be displayed in the order of alarm history, key history and history of input/output signal changes.
  The operation history occurrence time is updated in one second units.
- Up to 168 alarm histories, 400 key histories and 1500 histories of input/output signal changes can be stored.
- The 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

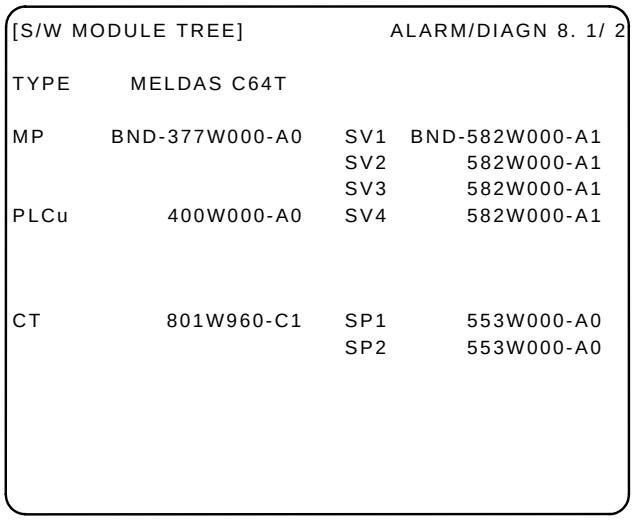
- The operation history data can be input and output without canceling the operation history mode.
- If the data is output with RS-232C while the operation history mode is suspended, the operation history mode will change from the suspended state to the execution state after the data output is completed.
- The operation history data in the NC memory cannot be deleted.

2.7.8 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.
If all of the servo axes and spindle axes cannot be displayed on one page, the subsequent details can be displayed by pressing the NEXT key.

2.7.8.1 S/W Module Tree

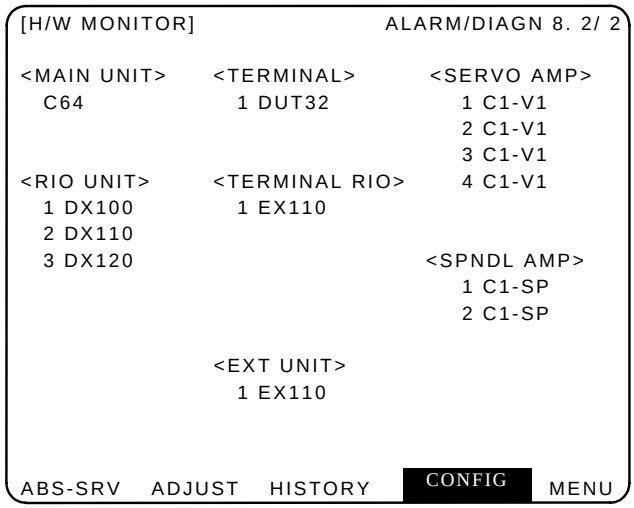
Pressing the menu key CONFIG displays the S/W MODULE TREE screen.



(Note) "SERIAL" may not be displayed depending on the model.

2.7.8.2 H/W Monitor

Pressing the menu key CONFIG and then NEXT displays the H/W MONITOR screen.



2.7.9 Auxiliary Axis Parameter

Pressing the menu key [AUX-PRM] displays the AUX-PRM screen.

```
[AUX-PARA]                                          ALARM/DIAGN  9. 1/11
  #              <1> AUX1      <2> AUX2      <3> AUX3      <4> AUX4
  1 MSR             FFFF          FFFF          FFFF          FFFF
  2 RTY             FFFF          FFFF          FFFF          FFFF
  3 PC1            32767         32767         32767         32767
  4 PC2            32767         32767         32767         32767
  5 PIT            32767         32767         32767         32767
  6 INP            32767         32767         32767         32767
  7 ATU             FFFF          FFFF          FFFF          FFFF
  8 PG1             1000          1000          1000          1000
  9                    0             0             0             0
 10 EMG            32767         32767         32767         32767
 11                    0             0             0             0
 12 FFC              100           100           100           100

#(   ) DATA(         ) (         ) (         ) (         )

    AUX-PRM       AUX-MON                                   MENU
```

Refer to “MR-J2-CT Series Specifications and Instruction Manual” (BNP-B3944) for details of the parameters.

2.7.9.1 Backup

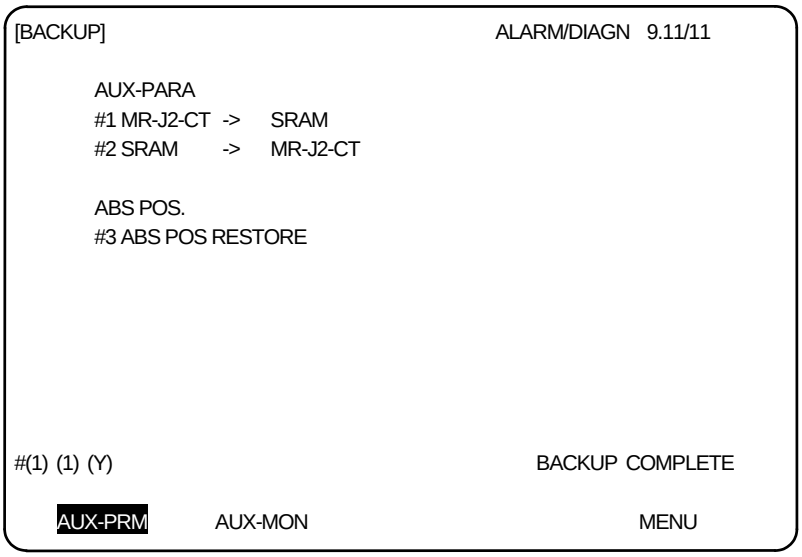
SRAM backup of the auxiliary axis parameters and absolute position restoration during amplifier replacement are executed on the BACKUP screen.
Only execute the operations during amplifier replacement.

(1) Procedure during MR-J2-CT amplifier replacement

The following shows the procedure during MR-J2-CT amplifier replacement. The operation is carried out on the BACKUP screen.
The absolute position of the current MR-J2-CT must be established first before carrying out this operation.

- SRAM backup of the auxiliary axis parameters

The current amplifier data is backed up in the SRAM of the CNC when the amplifier is replaced.
The following shows the procedure for SRAM backup of the auxiliary axis parameters.



Input "(1) (a)" from the BACKUP screen. (a: axis No. (1 to number of connected axes))
After inputting, the message "CONTINUE Y/N" will appear for confirmation of the data backup.
The data is backed up in the SRAM only if "(Y)" is input.
The message "BACKUP COMPLETE" will appear when the backup operation is finished.

If MR-J2-CT is not connected, the message "E01 SETTING ERROR" will occur, and the data will not be backed up.

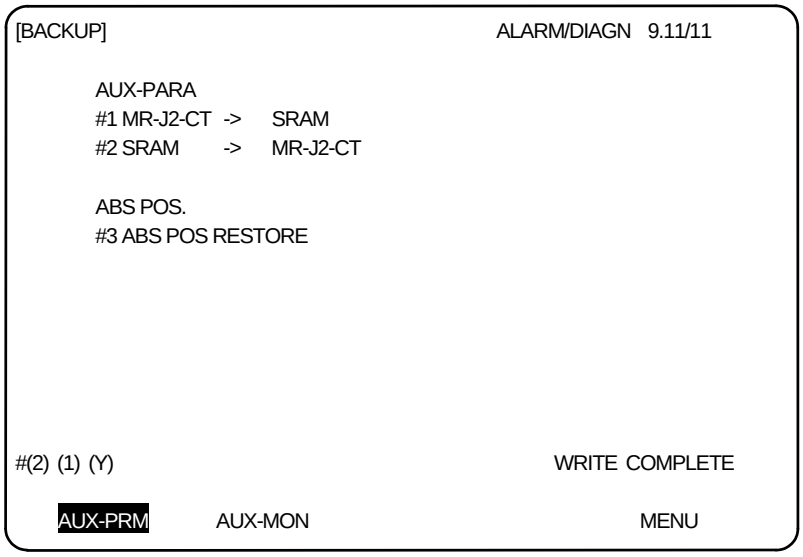
After the procedure is finished, turn OFF the CNC and MR-J2-CT power supply, and replace MR-J2-CT amplifier.
After replacing the MR-J2-CT amplifier, set the MR-J2-CT rotary switch to "7". Turn the CNC and MR-J2-CT power supply ON, wait for the 7-segment display to change from dot display, and then turn the power OFF. The auxiliary axis parameter data is written and the absolute position restored after the axis is set with the rotary switch and the power is turned ON again.

(Note) Turning ON the power supply with the MR-J2-CT rotary switch set to "7" initializes the parameters. If an amplifier with the absolute position established is replaced without carrying out this operation, the machine position will be created with the old amplifier's data.

- Writing auxiliary axis parameters to MR-J2-CT

After backing up the auxiliary axis parameters in the SRAM, turn the CNC and MR-J2-CT power OFF, and replace the MR-J2-CT amplifier.
Initialize the replacement amplifier beforehand.
After replacing the MR-J2-CT amplifier, turn the CNC and MR-J2-CT power ON. Write in the auxiliary axis parameter data and restore the absolute position.

The data in the CNC SRAM is written to the new amplifier.
The following describes the procedure for writing the auxiliary axis parameters to MR-J2-CT.



Input "(2) (a)" from the BACKUP screen. (a: axis No. (1 to number of connected axes))
After inputting, the message "CONTINUE Y/N" will appear for confirmation of the data backup.
The auxiliary axis parameters are written to MR-J2-CT only if "(Y)" is input.
The message "WRITE COPLETE" will appear when the writing operation is finished.

If MR-J2-CT is not connected, the message "E01 SETTING ERROR" will occur, and the data will not be written.

Auto-tuning parameters are written as shown below, using the auto-tuning selection (parameter #7 ATU) of the SRAM data and the MR-J2-CT side.

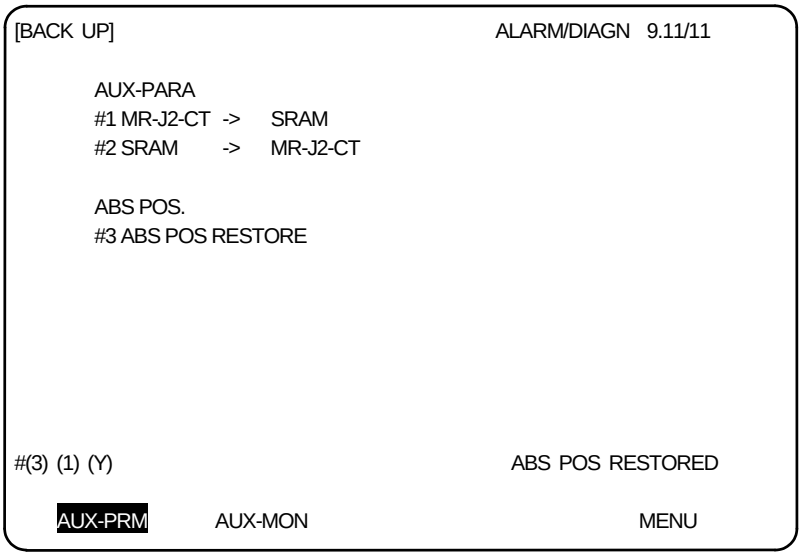
#7 ATU in SRAM	#7 ATU set in MR-J2-CT	Auto-tuning parameter written
Auto-tuned (0 or 1)	Auto-tuned (0 or 1)	Not written
Not auto-tuned (2)	Auto-tuned (0 or 1)	Written (Note 1)
Auto-tuned (0 or 1)	Not auto-tuned (2)	Written (Note 1)
Not auto-tuned (2)	Not auto-tuned (2)	Written (Note 1)

(Note 1) Which parameters can be input depends on the setting of “#7 ATU” parameter.
Refer to "2.6.2.7 MR-J2-CT parameters and N No. correspondence table" for details on auto-tuning target parameters.

(Note 2) The CNC does not hold the MR-J2-CT parameters.
The parameters are held on the MR-J2-CT side.
When the MR-J2-CT unit is replaced, the parameters must be set again in the new unit.

- Restoring the absolute position

After the procedure in “Writing auxiliary axis parameters to MR-J2-CT” is finished, restore the absolute position in the new amplifier.
The following describes the procedure for restoring the absolute position.



Input "(3) (a)" from the BACKUP screen. (a: axis No. (1 to number of connected axes))
After inputting, the message "CONTINUE Y/N" will appear for confirmation of the data backup. The absolute position is restored only if "(Y)" is input.
The message "ABS POS RESTORED" will appear when the absolute position restoration operation is finished.

If MR-J2-CT is not connected, the message "E01 SETTING ERROR" will occur, and the absolute position will not be restored.

(2) Messages displayed on the BACKUP screen

The following table shows the messages displayed on the BACKUP screen and their details.

Message	Details
CONTINUE Y/N	Determine whether to execute the operation by selecting either "Y" or "N".
BACKUP EXECUTION	The auxiliary axis parameters are being backed up in the SRAM.
BACKUP COMPLETE	The SRAM backup of the auxiliary axis parameters is finished.
AUX. WRITING EXEC.	The auxiliary axis parameters in the SRAM are being written into the MR-J2-CT.
WRITE COMPLETE	The writing of the auxiliary axis parameters in the SRAM to the MR-J2-CT is finished.
ABS POS RESTORED	The absolute position in the SRAM has been restored to the MR-J2-CT.

(3) Supplementary Items

The following shows the process when the No. of axes set in the set up parameters differs from the No. of MR-J2-CT axes connected.

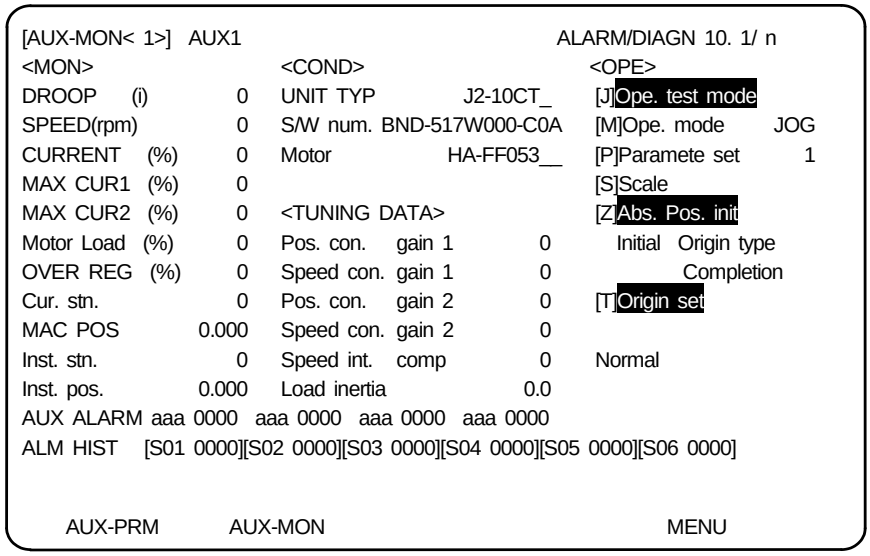
	No. of axes set No. of axes connected, or the No. of axes set = 0.
MR-J2-CT parameter input	MR-J2-CT parameter input not carried out.
MR-J2-CT parameter output	MR-J2-CT parameter output not carried out.

The following shows the process when there are more input data axis Nos. than there are axes set in the setup parameters.

	Input data axis No. > No. of axes set
MR-J2-CT parameter input	Input data exceeding the No. of set axes is not input.

2.7.10 Auxiliary Axis Monitor

Pressing the menu key [AUX-MON] displays the AUX-MON screen.



<Auxiliary axis monitor items>

Item	Data range	MR-J2-CT data name	Remarks
MON			
DROOP	–999 to 999	Position droop	(i)
SPEED		Motor speed 1	(r/min)
CURRENT	–999 to 999	Effective load rate	(%)
MAX CUR1	–999 to 999	Command torque	(%)
MAX CUR2	–999 to 999	Command torque peak hold	(%)
Motor Load	–999 to 999	Motor load rate	(%)
OVER REG	–999 to 999	Regeneration resistor heat generation load rate	(%)
Cur. stn.	1 to 360	J2CT status, station position	
MAC POS	–99999.999 to 99999.999	Feedback machine position	◦
Inst. stn.	1 to 360	Target station No.	
Inst. pos.	–99999.999 to 99999.999	Random command position	◦
AUX ALARM	Alarm No. Alarm information	System alarm	
	Alarm No. Alarm information	Servo alarm	
	Alarm No. Alarm information	System warning	
	Alarm No. Alarm information	Servo warning	
	Alarm No. Alarm information	Operation error	
ALM HIST	[Type Error No.]	Alarm history (type and error No.)	6 error max
COND			
UNIT TYP			
S/W num.			
Motor			
TUNING DATA			
Pos. con. gain 1	0 to 999	Position control gain 1	rad/s
Speed con. gain 1	0 to 999	Speed control gain 1	rad/s
Pos. con. gain 2	0 to 999	Position control gain 2	rad/s
Speed con. gain 2	0 to 999	Speed control gain 2	rad/s
Speed int. comp	0 to 999	Speed integral compensation	ms
Load inertia	0 to 999.9	Load inertia ratio	-fold

Refer to "2.7.10.2 Auxiliary axis adjustment function" for details on the <OPE> area on the screen.

2.7.10.1 Alarm History Display

Up to 6 items of alarm history (alarm No. and alarm information) for the auxiliary axis are displayed. The items are displayed in order from left to right, with the newest item on the left.

History	Newest ←					→ Oldest
Display order	1	2	3	4	5	6
Alarm history display	[S01 0000]	[S02 0000]	[S03 0000]	[S04 0000]	[S05 0000]	[S06 0000]

∗[Alarm No. Alarm information]

2.7.10.2 Auxiliary Axis Adjustment Function

The auxiliary axis adjustment function enables the auxiliary axes that were controlled using the PLC to be controlled using key operation from the screen.
The adjustment function of the auxiliary axes is carried out in the <OPE> area on the AUXILIARY AXIS MONITOR screen.
The following explains the various display items in the <OPE> area.

Display items of the <OPE> area

<table>
<tr><th>Item</th><th>Display details</th></tr>
<tr><td>[J] Ope. test mode</td><td>This displays the VALID/CANCELED status of the operation adjustment mode.
This item is highlighted when the mode is valid.</td></tr>
<tr><td>[M] Ope. mode</td><td>The operation mode of the current auxiliary axis is displayed in the column to the right of the item.
(Display details of the operation mode)
AUTOMATIC : In automatic operation mode
JOG : In JOG operation mode
STEP : In incremental mode
MANUAL : In manual operation mode
HANDLE : In handle mode
Zero-P.Ret : (dog-type return only) In refetence point return mode</td></tr>
<tr><td>[P] Paramete set</td><td>"Operation parameter group" means a series of MR-J2-CT parameters as shown below.
<table>
<tr><th>Parameter name</th><th>Parameter No.</th></tr>
<tr><td>Aspeed1-4</td><td>#150,158,166,174</td></tr>
<tr><td>Mspeed1-4</td><td>#151,159,167,175</td></tr>
<tr><td>time1.1-4.1</td><td>#152,160,168,176</td></tr>
<tr><td>time1.2-4.2</td><td>#153,161,169,177</td></tr>
<tr><td>TL1-4</td><td>#154,162,170,178</td></tr>
<tr><td>OD1-4</td><td>#155,163,171,179</td></tr>
<tr><td>just1-4</td><td>#156,164,172,180</td></tr>
<tr><td>near1-4</td><td>#157,165,173,181</td></tr>
</table>
There are 4 sets of these parameters, and the operation keys are used to changeover between them.

The current operation parameter group No. is displayed in the column to the right of the item.
<table>
<tr><th>Operation parameter group item</th><th>1</th><th>2</th><th>3</th><th>4</th></tr>
<tr><td>Display details</td><td>1</td><td>2</td><td>4</td><td>4</td></tr>
</table></td></tr>
<tr><td>[S] Scale</td><td>When the operation mode is the incremental or handle mode, the setting details are displayed in the column to the right of the item.
This column is empty when the operation mode is another mode.
(Display details during the incremental mode)
The feed amount per startup is displayed.
<table>
<tr><th>Feed amount</th><th>1°</th><th>1/10°</th><th>1/100°</th><th>1/1000°</th></tr>
<tr><td>Display details</td><td>1/1</td><td>1/10</td><td>1/100</td><td>1/1000</td></tr>
</table>
(Display details during the handle mode)
The handle rotation magnification is displayed.
<table>
<tr><th>Rotation magnification</th><th>1</th><th>10</th><th>100</th><th>1000</th></tr>
<tr><td>Display details</td><td>1</td><td>10</td><td>100</td><td>1000</td></tr>
</table></td></tr>
<tr><td>[Z] Abs. Pos. init.</td><td>This shows the initialization selected/not selected status of the absolute position.
This item is highlighted when initialization is selected.</td></tr>
</table>

<table>
<tr><th>Item</th><th>Display details</th></tr>
<tr><td>Initial</td><td>
The initialization method designated by the auxiliary axis parameter #120 ABS Type (ABSOLUTE POSITION DETECTION PARAMETER) is displayed in the column to the right of the item.
(Display details of the initialization method)
Dog type : Dog-type method
Initial Stopper type : Stopper method
Initial Origin type : Origin point alignment method

The status at initialization is displayed in the column below the item.
(Display details of the initialization status)
<table>
<tr><th rowspan="2">MR-J2-CT status</th><th colspan="3">ABSOLUTE VALUE DETECTION PARAMETER</th></tr>
<tr><th>Stopper method</th><th>Origin point alignment method</th><th>Dog-type method</th></tr>
<tr><td>ABSOLUTE POSITION LOSS</td><td>Illegality</td><td>Illegality</td><td>Illegality</td></tr>
<tr><td>EXECUTING ABSOLUTE POSITION INITIALIZATION ON</td><td>Pressing</td><td>Origin type</td><td>—</td></tr>
<tr><td>STOPPER</td><td>Press Rel.</td><td>—</td><td>—</td></tr>
<tr><td>EXECUTING REFERENCE POINT SETTING ON</td><td>—</td><td>Origin return</td><td>—</td></tr>
<tr><td>ZERO POINT RETURN</td><td>—</td><td>—</td><td>Zero-P.Ret</td></tr>
<tr><td>ABSOLUTE POSITION ESTABLISHMENT</td><td>Ret.Ref.P</td><td>Completion</td><td>Completion</td></tr>
</table>
Explanation of display details
Completion : This is displayed when the absolute position of the stopper, origin point alignment, or dog-type method is established.
For the stopper and origin point alignment methods, this shows that the grid has been reached, and the initialization has been completed.

Illegality : This is displayed when the absolute position has been lost in the stopper, origin point alignment, or dog-type method.

Pressing : This is displayed during absolute position initialization when the zero point is being initialized by the stopper method.
This status continues until the stopper is reached.

Press Rel. : This status is entered once the stopper has been reached, or when a fixed time has elapsed and the current limit has been reached.

Ret.Ref.P. : This shows the status after the stopper, from the time the stopper is released to the time immediately before the axis reaches the grid after moving in the opposite direction.
For the origin point alignment method, this indicates the status when the tool is moving in the origin point direction set in parameter #120 ABS Type to a time just before the grid is reached.

Origin type : When the zero point is initialized by the origin point alignment method, this indicates the status when the tool is moving to the machine basic point from the time during absolute position initialization, until the origin point is designated.

Zero-P.Ret : This shows that the tool has returned to the refernce point by a dog-type method initialization.
</td></tr>
</table>

Item	Display details
[T] Origin set	This shows the ON/OFF status of the origin point setting. This item is highlighted when selected.
Operation status	The operation status is displayed in the column to the right of the item. Normal Rot. : Rotation is in the forward direction. Reverse Rot. : Rotation is in the reverse direction. Stop : The auxiliary axis is stopped.

2.7.10.3 Operation Method for the Auxiliary Axis Adjustment Function

The following shows the operation method for the auxiliary axis adjustment function.

(1) Operating conditions for the operation adjustment mode

(a) The operation adjustment function is a function of the AUXILIARY AXIS MONITOR screen. Change to the AUXILIARY AXIS MONITOR screen when using this function.

(b) Confirm the following items before entering the auxiliary axis adjustment mode. The auxiliary axis adjustment mode cannot be used if the following conditions are not fulfilled.

1) There must not be the "Y03 AUX AMP UNEQU." error. (The MR-J2-CT must be connected, and in a usable state.)

2) The OPERATION ADJUSTMENT MODE VALID signal (R3684 bit0) must be ON.

3) The SERVO OFF signal and INTERLOCK CANCELED signal (R3603/R3609/R3615/R3621/ R3627/R3633/R3639 bits 0, 4, 5) must be OFF.

4) The OPERATION START signal (R3602/R3608/R3614/R3620/R3626/R3632/R3638 bit 0) must be OFF.

(c) Do not turn ON the OPERATION START signal command during the auxiliary axis operation adjustment mode. The MR-J2-CT may make unanticipated movements when the operation adjustment mode is canceled.

(2) Validating/canceling the operation adjustment mode

Carry out the following operation to validate the auxiliary axis adjustment function.

Operation key: [J]

Conversely, carry out the following operation to cancel the operation adjustment mode.

Operation key: [J] carries out a screen changeover operation.

(3) Functions of the various keys in the operation adjustment mode

(a) Setting the operation adjustment mode

The initial display of the operation adjustment mode reflects the current PLC settings.
The following operations are validated when the operation adjustment mode is turned ON.

Function	Operation key	Details
Ope. mode	Operation key: M	This changes the operation mode. The operation mode changes over as follows every time the key is pressed. JOG → INCREMENTAL → MANUAL OPERATION → HANDLE → (dog-type method only) ZERO RTN → JOG The initial display becomes "JOG" when the operation adjustment mode is validated from automatic adjustment.
Paramete set	Operation key: P	Set the operation parameter group No. The operation parameter group changes as follows every time the key is pressed. 1 → 2 → 3 → 4 → 1
Scale	Operation key: S	The settings for this function are only validated when the operation mode is the "Incremental" mode or the "Handle" mode. Incremental mode: Set the feed amount per startup. The feed amount changes as follows every time the key is pressed. 1/1° → 1/10° → 1/100° → 1/1000° → 1/1° Handle mode Set the handle rotation magnification. The handle rotation magnification changes as follows every time the key is pressed. 1 → 10 → 100 → 1000
Abs. Pos. init	Operation key: Z	The settings for this function are only validated when the zero point is initialized by the "Stopper" method or the "Reference point alignment" method. This function operates when initializing the absolute position. The function is turned ON/OFF every time the key is pressed.
Origin set	Operation key: T	The settings for this function are only validated when the zero point is initialized by the "Reference point alignment" method. Press this operation key to set the reference point. The function is turned ON/OFF every time the key is pressed.

(b) Starting and stopping the operation

The following operation start and stop operations apply to the "JOG", "INCREMENTAL", and "MANUAL OPERATION" modes.
After setting the mode with the operations described in item (a), the operation is started and stopped using the following operation keys.

Function	Operation key	Details
Normal rotation	Operation key: [↑]	This starts the rotation in the forward run direction. The rotation will stop if this key is pressed while starting.
Reverse rotation	Operation key: [↓]	This starts the rotation in the reverse run direction. The rotation will stop if this key is pressed while starting.
Stop	Operation key: Press any key other than the SHIFT key. **(Example)** [A] key	This stops the rotation during forward or reverse run.

The operation is stopped and the operation adjustment mode is canceled if after starting, the screen is changed from the AUXILIARY AXIS MONITOR screen to another screen, or the monitor axis is changed.
The following table shows the relation between the various key operations and the operation start/stop.

Key operation	Operation status (status before key operation)			Screen change
	Stop status	Forward run status	Reverse run status	
FORWARD RUN START ↑ key	Normal	Stop	Stop	No change
REVERSE RUN START ↓ key	Abnormal	Stop	Stop	No change
ALPHANUMERIC CHARACTER key	Remain in stop status	Stop	Stop	No change
CNC FUNCTION, MENU KEY, NEXT/PREVIOUS PAGE key	Remain in stop status	Stop	Stop	Change
→←, TAB key, SHIFT, CB, DEL, INPUT key, MENU key, CYCLE START, RESET key, etc.	Remain in stop status	Remain in forward run status	Remain in reverse run status	No change

(Note 1) The emergency stop of the hotline with the CNC unit is always valid, so set bit 2 (bus emergency stop invalid) of the MR-J2-CT parameter #103 Emgcont to "1".

(Note 2) CNC axis handle movement is invalid in the MR-J2-CT handle mode.
The No. 1 handle is fixed for the MR-J2-CT handle mode.

2.8 Graphics

2.8.1 Outline of Functions

The GRAPHIC screen will appear when the function selection key SFG is pressed.

Machine tool operations can be monitored, and the machining program path can be illustrated on the GRAPHIC screen. This is useful for inspecting the program.

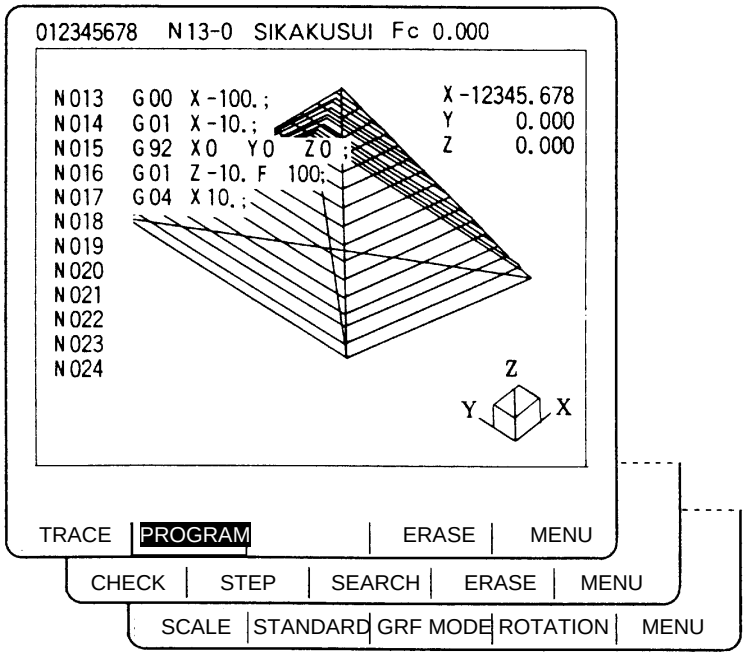
(1) Trace function
The trace function illustrates the actual machine movement path and draws the machine motion itself. The machine operation can be monitored during machining.

(2) Program check function
The program check function illustrates the machining program movement path and draws the operation results in the NC without carrying out automatic operation.
Thus, it can draw accurate figures at high speed, allowing the machining program to be checked.

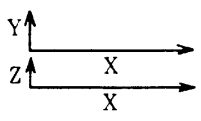
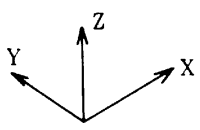
(Note 1) The graphic function is an optional function that is available only when the NC-dedicated display unit CT100 is used.
The GRAPHIC screen will not be displayed if this specification is not added.

2.8.2 Menu Function

The menu configuration in the GRAPHIC screen is shown below. These are operation menus for selecting operations on the GRAPHIC screen. Select the menus by pressing the corresponding menu key. If the required operation menu does not appear, press the MENU key. The next menu will appear.



List of functions

Menu	Function
TRACE	This function is used when carrying out trace display.
PROGRAM	The machining program can be displayed on the screen during drawing with this function.
ERASE	This function erases the graphics shown on the GRAPHIC screen.
CHECK	This function is used to continuously check the machining program.
STEP	This function is used to check machining programs one block at a time. Commands G27 to G30, G60, fixed cycle and corner R/C are divided into several blocks.
SEARCH	The program to be checked can be set (called) with this function.
SCALE	This function changes the figure scale and display position. Portions of the figure can be enlarged and drawn.
STANDARD	This function automatically changes the display center and figure scale in the machine stroke length range.
GRF MODE	1-plane display  2-plane display (M system only)  3-dimensional display (M system only)  Any of the above can be set for a random axis.
ROTATION	In the 3-dimensional display mode, a drawing of a solid object can be made looking at the object from any direction, not just the front. (M system only)

2.8.3 Use of the Trace Mode (TRACE)

To select the trace mode, press the menu key TRACE.

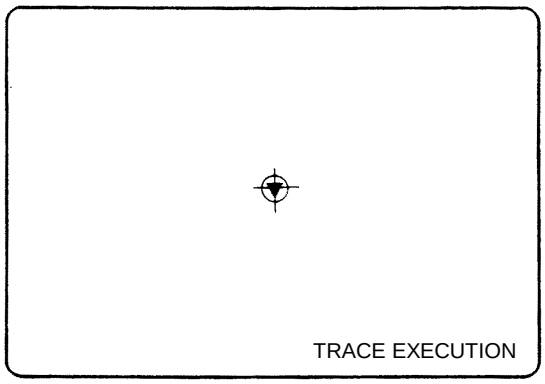
When the trace mode is selected, the machine position is indicated by a triangle mark. If this tool mark does not appear, it is because the machine position is outside the screen display range. In this case, change the display range (explained below) and move the coordinates, or widen the display range by greatly increasing the scale value.

The machine position is always drawn in the trace mode. The actual movement by automatic or manual operation is drawn. The machine zero point appears as a ⊕ mark.

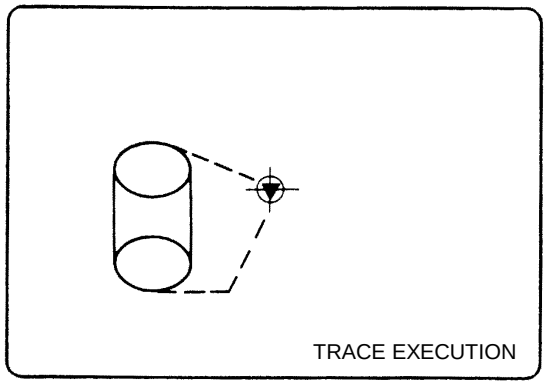
(1) Trace start

Press the menu key TRACE.

1) The message TRACE EXECUTION appears, indicating the trace mode.
2) A tool mark appears at the machine position.



3) After this, the machine movement is drawn in the trace mode.



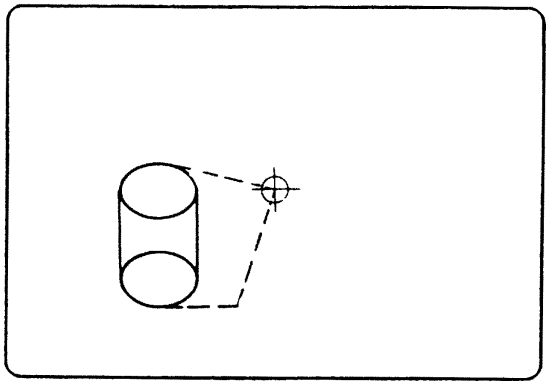
(2) Trace release

Press the menu key TRACE.

1) The TRACE EXECUTION message disappears, indicating that the trace mode is released.
2) The tool mark disappears.

(Note 1) The trace mode can also be released by the following operation:

1. Press the function selection key F0.



(3) Line types in the trace mode

1. Movement by rapid traverse and manual feed: Broken line
2. Movement by cutting feed: Solid line

(Note) When high-speed drawing is carried out in the trace mode, the drawing shape becomes more deformed as the command speed increases. Correct shapes especially cannot be drawn during high-speed machine lock. Set the cutting feed to 2000mm/min. or less.

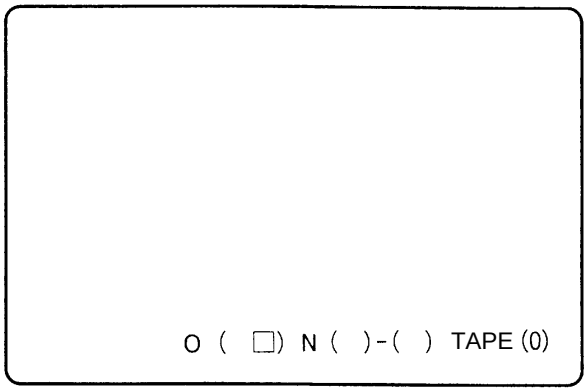
2.8.4 Use of the Check Modes

To select a check mode, press menu key [CHECK] or [STEP].
The check mode appears immediately when the check mode is selected.

(1) Preparation for checking

Press the menu key [SEARCH] to call the program.

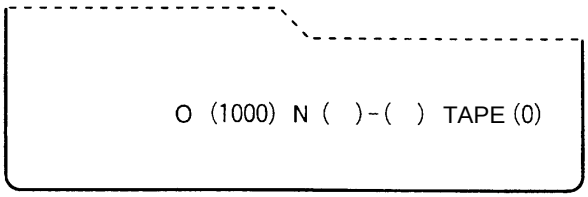
Press the menu key [SEARCH].



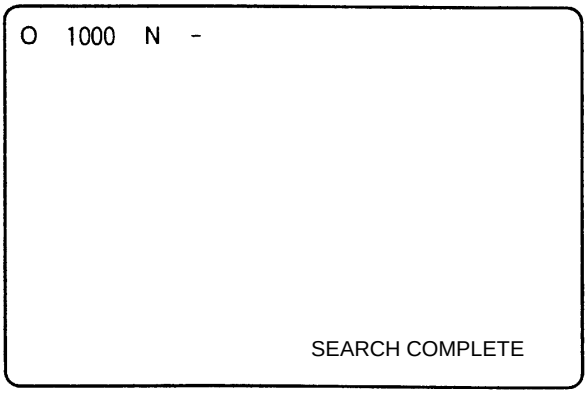
The data setting area appears.

(Example) To check O1000, proceed as follows:

O (1 0 0 0)
N ( ) – ( )
Tape (0)



Press [INPUT].



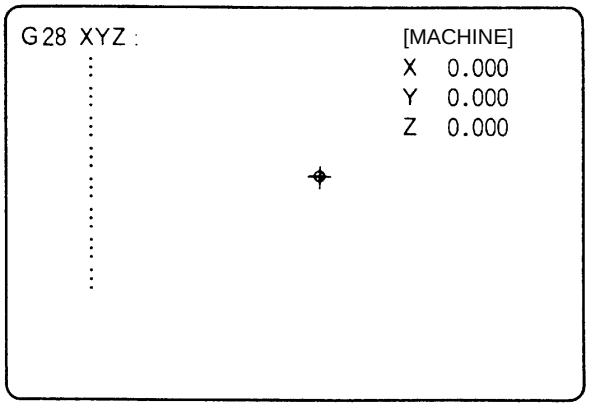
The setting area disappears.
A message appears to notify that the search is completed.

(Note 1) This operation search is exactly the same as that of the MONITOR function. Pressing the automatic start button after the search is completed starts the automatic operation.
The operation mode is designated by the operation mode selection switch on the machine operation panel.

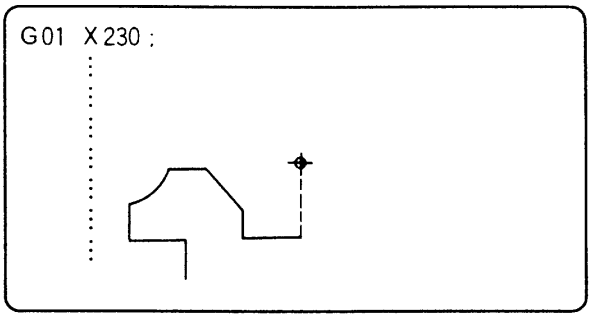
(2) Checking start

1) To check the program continuously in the same manner as in automatic continuous operation:

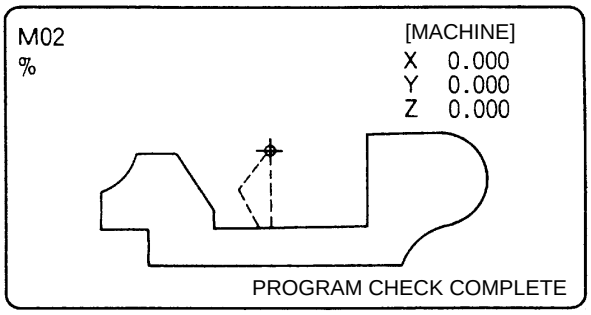
Press the menu key [CHECK] on the GRAPHIC screen.



The program check is executed.



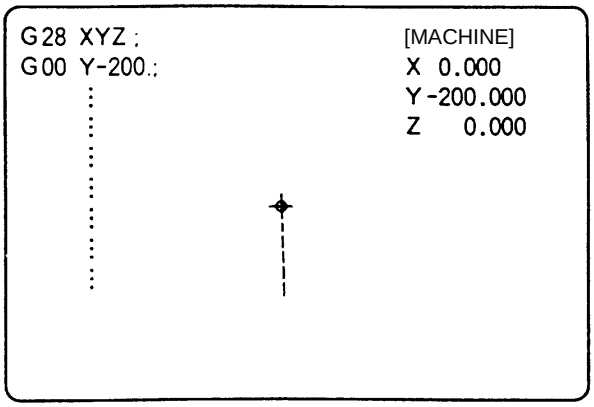
The program check stops at the program end.



2) To check the program block by block in the same manner as in single block operation:

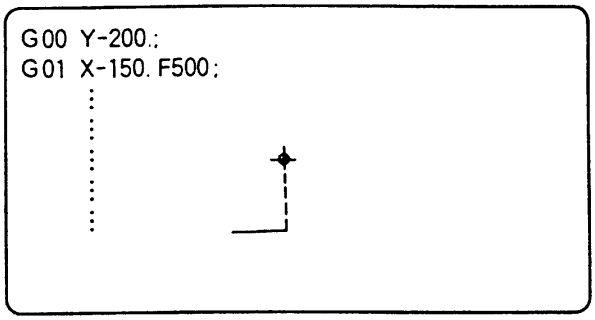
Press the menu key STEP on the GRAPHIC screen.

The system executes one block and stops.



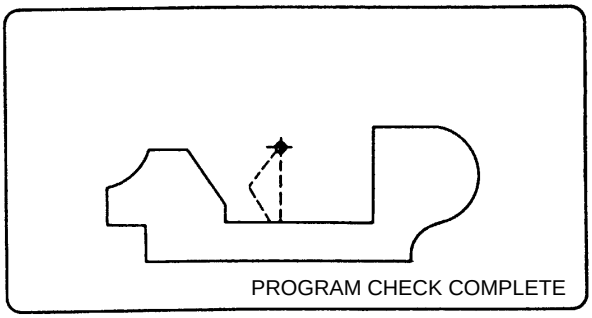
Press the menu key STEP on the GRAPHIC screen.

The system executes the next block and stops.



Press the menu key STEP successively.

The system executes one block each time the menu key STEP is pressed.

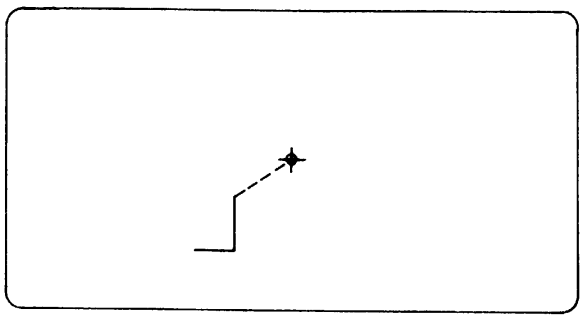


The program check stops at the program end.

Pressing the menu key STEP after the check is completed will start the step check from the program head.

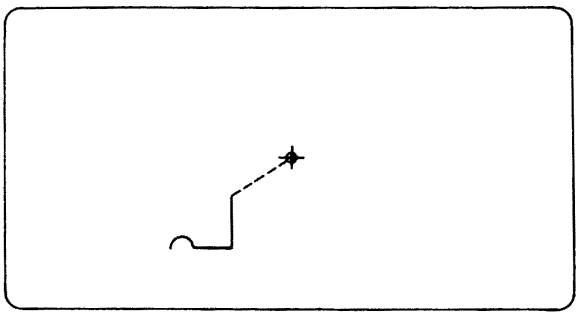
3) Changeover between continuous check and step check modes
Pressing the menu key [STEP] during continuous check execution changes the mode to the step-check mode, in which the system checks subsequent blocks of the program one at a time.
Similarly, the mode can be also changed from the step-check mode to the continuous check mode using the same key.

Execute the continuous check.



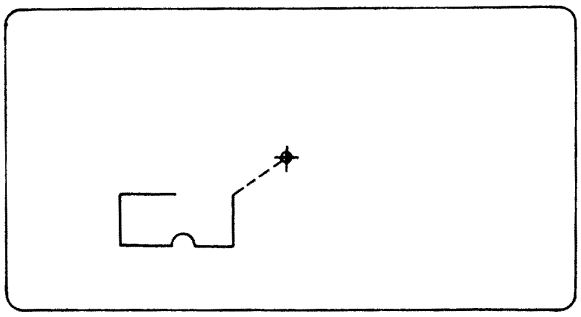
Press the menu key [STEP].

The system executes one block and stops.



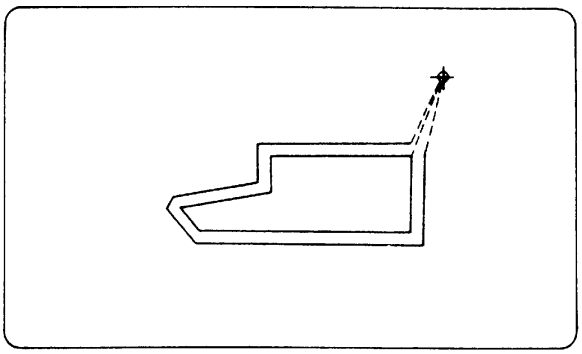
Press the menu key [STEP].

The system starts a continuous check again.



4) Drawing during tool nose radius compensation
If the program being checked involves tool nose radius compensation, both the program path and tool center path are traced.

Execute the continuous check.



(3) Line types during checking

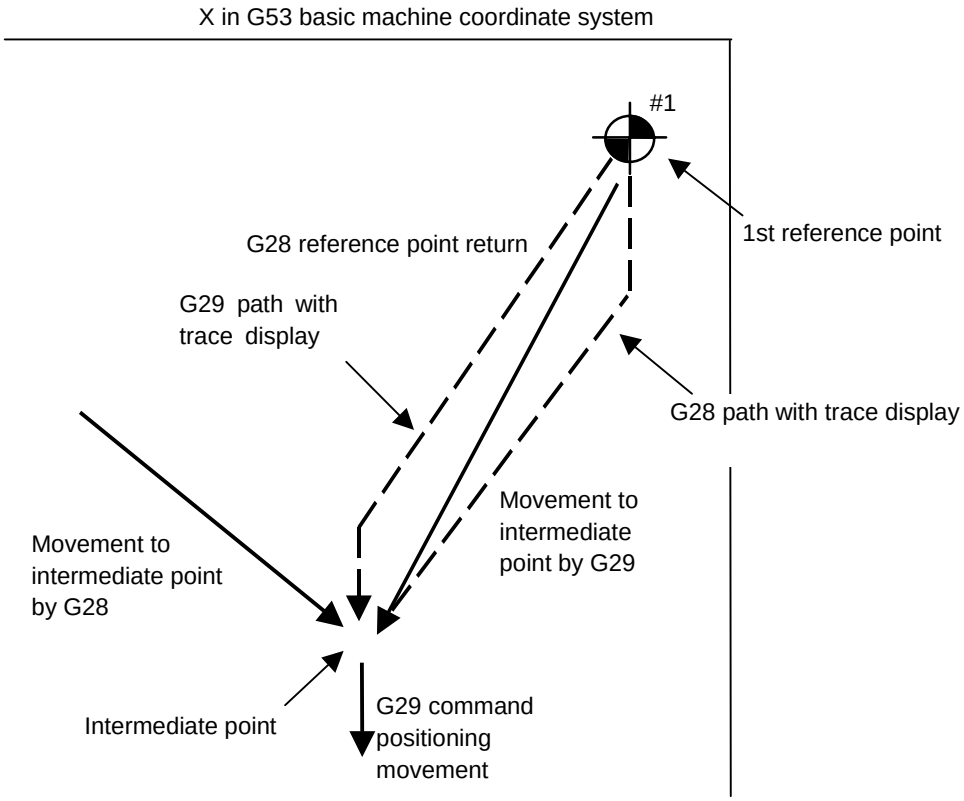
		During tool nose radius compensation	Not during tool nose radius compensation
Manual feed		—	—
Rapid traverse	Program path	Broken line	—
	Tool center path	Broken line	Broken line
Cutting feed	Program path	Solid line	—
	Tool center path	Solid line	Solid line

(4) Relation with other functions

Function name	Graphic check	Remarks
Coordinate system rotation	○	
Figure rotation function	○	
Decimal point input command	○	Either Type I or Type II is possible.
Mirror image function	○	Refer to (6) Precautions, item 4.
Z axis cancel function	×	The path is drawn exactly as the commanded movement.
Interlock	×	
External deceleration	×	
Override	×	
Feed hold	×	
Cycle start	×	If CYCLE START is pressed during checking, a "CHECK EXECUTION" alarm will appear while it is held down.
Auto-restart	×	CHECK COMPLETE occurs with M02/M30.
Workpiece coordinate system offset	○	This is set at the position determined by the offset amount actually set.
Local coordinate system offset	○	Same as above.
Compare stop function	×	The operation does not stop even if the set stop block is executed.
Helical interpolation function	×	Drawn with straight lines.
Fixed cycle/Special fixed cycle function	○	
Manual mode and handle	×	During checking, the machine can be moved by changing the mode to manual or handle feed. Checking is invalid.
F1 digit feed function	×	
Reference point return/ Start position return	×	Refer to Note 1.

Function name	Graphic check	Remarks
User macro I and II	○	Basic variable operations, and all judgment and branch functions are valid. Note that the following functions are invalid. • Macro interface input/output • NC alarm • Single block stop, miscellaneous function complete signal, WAIT suppression. • Feed hold, feedrate override, G09 valid/invalid • Position data Note that the end point coordinates of the previous block are valid. When these commands are issued, they are ignored or illegal data is input.

(Note 1) Reference point return commands and start position return commands are valid, but the actual machine movement (trace display) will differ partially from the drawing by the program check function.
When a reference point return is carried out via the intermediate point with a G28 or G30 command, positioning to the intermediate point is in a straight line. Positioning from the intermediate point to the reference point is carried out independently for each axis.
With the G29 command, positioning to the intermediate point is also carried out independently for each axis.
However, drawings using program checking are always drawn with straight lines, even when returning via the intermediate point. Thus, the path will differ partially from the trace display.



(5) Handling of variables, parameters and compensation amounts

All the various data in program checking is handled in the same manner as when all operations are executed.

	Saving data before the checking start	Explanation
Parameters	Not possible	Input commands executed in program checking are set as actual data.
Workpiece offset	Not possible	Same as above.
Common variable	Not possible	Same as above.
Local variable	Not possible	Same as above.
Tool compensation amount	Not possible	Same as above.

(6) Precautions

(1) If menu key [STANDARD] or [SCALE] is selected during drawing in trace or check mode, the drawing is interrupted. The drawing resumes when the selected function is finished.
In trace mode, the part that would be drawn during the interruption is not drawn.

(2) In trace or check mode, drawing will continue even if the screen changed to other function screens.

(3) All graphics are erased if the function key [F0] is pressed. The trace and check modes are also cleared.

(4) In check mode, machine operation switches such as external mirror image are validated only if set before starting a check.

(5) If the automatic start button is pressed during checking, operation alarm "EXECUTING PROGRAM CHECK" will occur while the button is held down, and automatic start will not be possible.
Press the automatic start button after completion of checking or after the interruption by the reset.

(6) If a check search or check start is attempted during automatic start or automatic operation pause, operation alarm "PROGRAM RUNNING" will occur, and the search or start will not be possible.

(7) Drawings in the check function are made only according to the NC internal operation results, so drawing is not possible for commands requiring machine movement.

(8) The various data set during checking and in the program are handled as follows:
1) Workpiece offset, common variables and local variables
Data before checking is not saved.
2) Tool compensation amounts
Data before checking is not saved.
3) Parameters
Data before checking is not saved.

(9) Checking finish
1) Execution of an M00/M01 command causes "PROGRAM STOP".
(Note) When M01 is executed, drawing will stop regardless of whether the optional stop switch signal is ON or OFF.
2) Execution of an M02/M03 command causes "PROGRAM CHECK END".
(Note) The tape is not rewound in the tape mode.
3) The checking is terminated by resetting, pressing [TRACE], or pressing [F0].
(Note) If a program error occurs, cancel the error by one of the operations above.

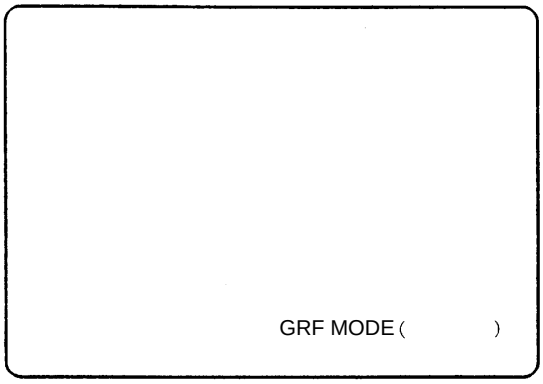
2.8.5 GRF Mode (GRF MODE)

(1) GRF mode types

There are three types of GRF modes: 1-plane, 2-plane and 3-dimensional. The axes of each plane or solid object can be randomly designated.

(2) Setting the GRF mode

Press the menu key [GRF MODE].



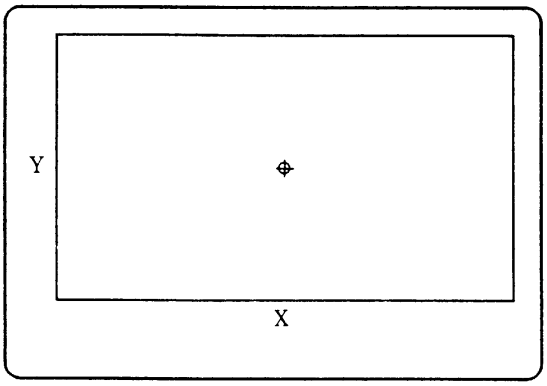
1) The GRF MODE setting area appears.
2) Directly set the axis names according to the following examples.

(Example 1) Setting the 1-plane display mode

Set the horizontal and vertical axis names in order, and press the [INPUT] key.

Set the GRF mode.
(Example) GRF mode ([X] [Y])

Press the [INPUT] key.



1) When the GRF mode changes over, all already drawn graphics are erased, and the new coordinate axes, etc., appear.

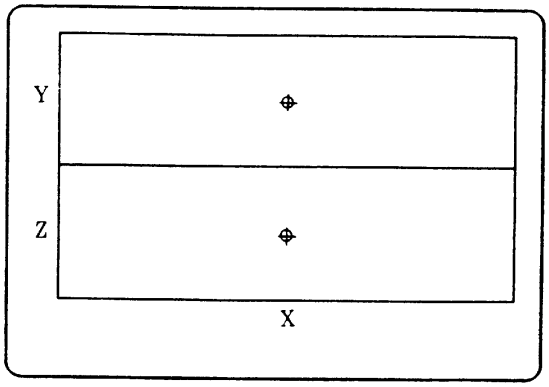
(Example 2) Setting the 2-plane display mode (M system only)

Set the horizontal and vertical axis names of the upper plane and those of the lower plane in order, and press the [INPUT] key.

Designate a common name for the horizontal axes of the upper and lower planes.

Set the GRF mode.
(Example) GRF mode
([X] [Y] [X] [Z])

Press the [INPUT] key.

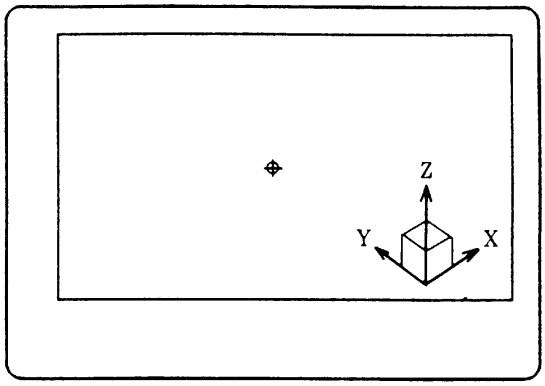


1) When the GRF mode changes over, all already drawn graphics are erased, and the new coordinate axes, etc., appear.

(Example 3) Setting the 3-dimensional display mode (M system only)

Set the 3-dimensional display mode.
(Example) GRF mode
(X Y Z)

Press the INPUT key.



1) When the GRF mode changes over, all already drawn graphics are erased, and the new coordinate axes, etc., appear.

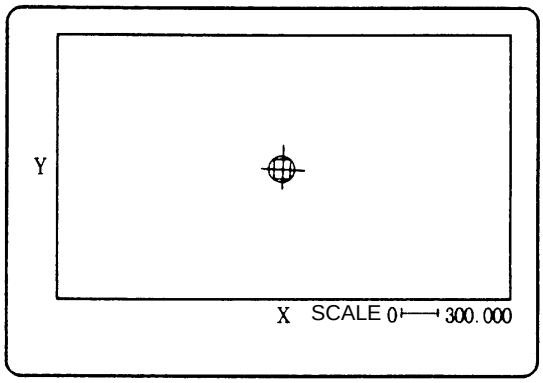
2.8.6 Scale (SCALE)

The size and position of the graphics drawn on the GRAPHIC screen can be changed.

2.8.6.1 Changing the Scale

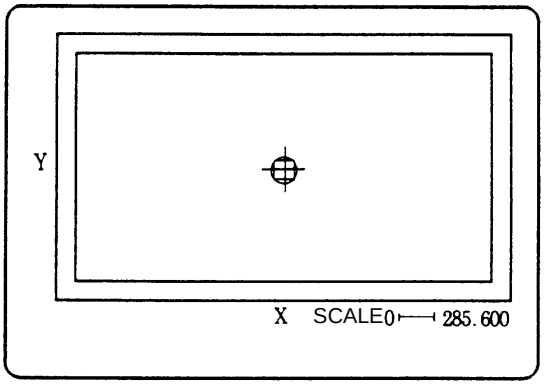
Press the menu key SCALE .

1) SCALE appears in the lower right portion of the screen. The current scale value appears.
2) The cursor indicates the center of the screen.



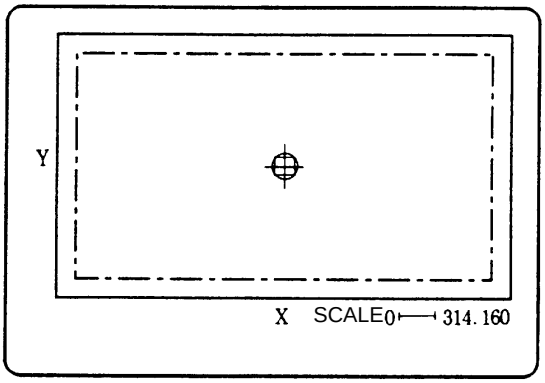
To enlarge the graphics, press the = - key to decrement the scale value.

1) The scale value is decremented.
2) The frame indicating the display range according to the set scale value is displayed with a solid line.



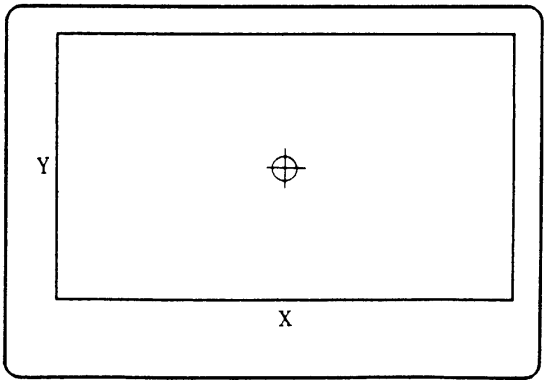
To reduce the graphics, press the + key to increment the scale value.

1) The scale value is incremented.
2) The frame indicating the reduced size of the current display range according to the set scale value is displayed with a chain line.



Press the INPUT key.

1) All already drawn graphics (if present) are erased.



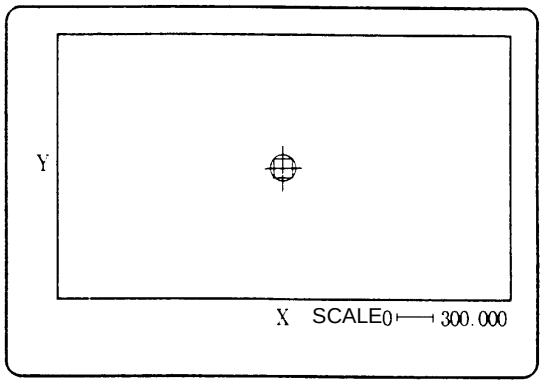
(Note 1) The scale value is incremented/decremented by approx. 5% each time the [+] or [=/-] key is pressed.
(Note 2) The scale value can be changed in the range of 0.100 to 9999.999.

2.8.6.2 Changing the Display Position

To move the drawing position, either designate the center of the display range using the cursor key, or use the current tool position as the center of the display range using the [CAN C.B] key.

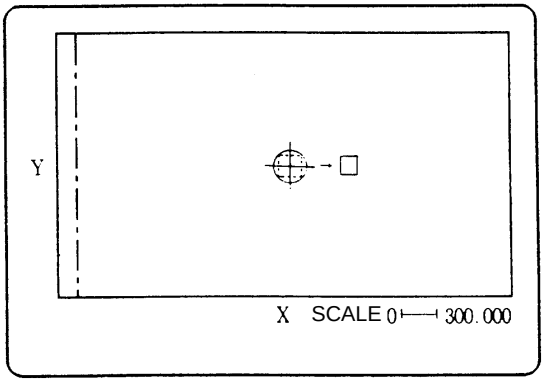
(1) Designating the display range center

Press the menu key [SCALE].

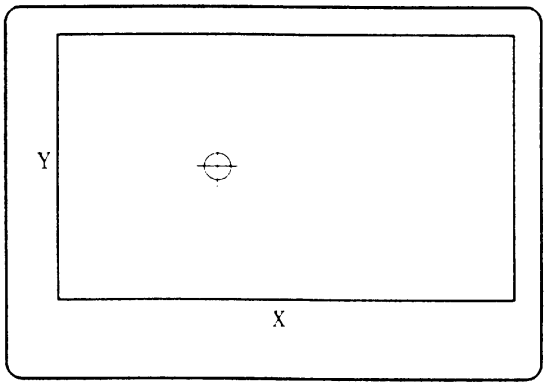


1) The cursor indicating the display range center appears.

Move the cursor to the required display center position using the cursor keys [↑], [↓], [←], [→].



Press the [INPUT] key.

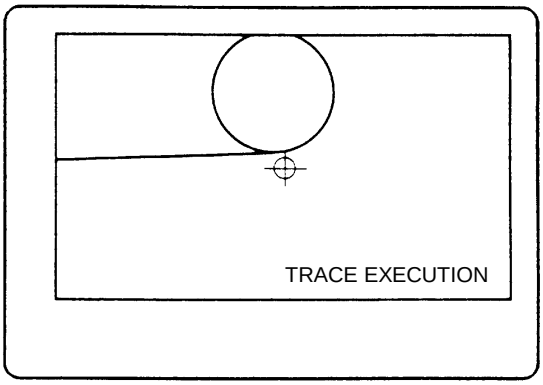


1) The coordinate system changes to the new coordinate system (having the cursor position as the screen center).
2) All already drawn graphics (if present) are erased, and the screen returns to the initial screen.

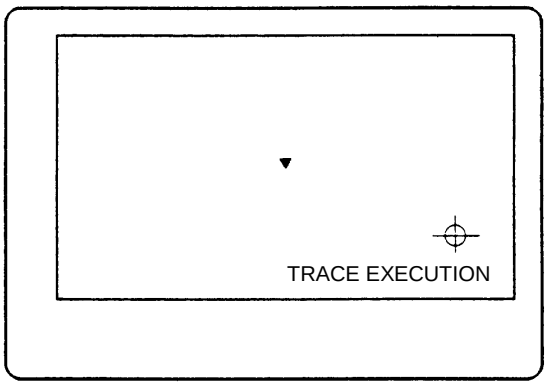
(Note 1) The amount the center moves for one cursor shift varies depending on the current scale value.

(2) Using the tool position as the display range center

For example, assume that the tool mark is not in the screen center, and a drawing is made as shown at the right.



Press the keys [SHIFT] and [CAN C.B].



1) The tool mark appears in the screen center.
2) If the message TRACE EXECUTION appears, consecutive drawing is made.

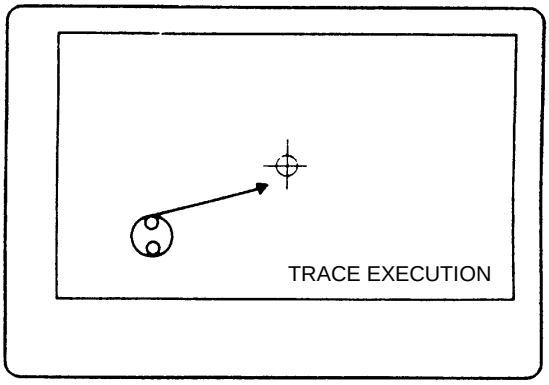
(Note) [SHIFT] + [CAN C.B] function

This function displays the current machine position in the screen center.

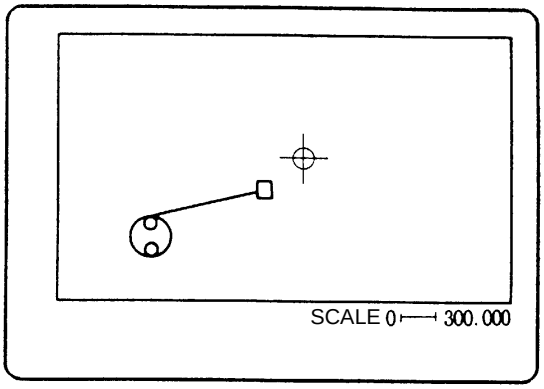
1) This operation cannot be carried out while any one of SCALE, GRF MODE, ROTATION, or CHECK SEARCH is valid.
2) The machine position is not moved in program check mode, even if drawing is carried out. Therefore, the tool center cannot be displayed in the center of the screen, even with the above key operation.
3) However, both the drawing and machine position move in the trace mode, so the tool center can be displayed in the center of the screen with the above key operation.

(Example) Changing the scale and display position simultaneously

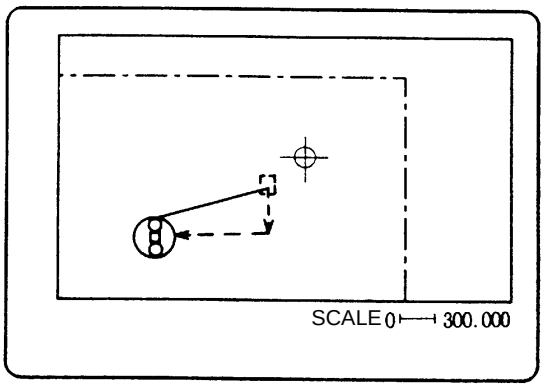
To enlarge the graphics (shown at the right) in the screen center, carry out the following operations:



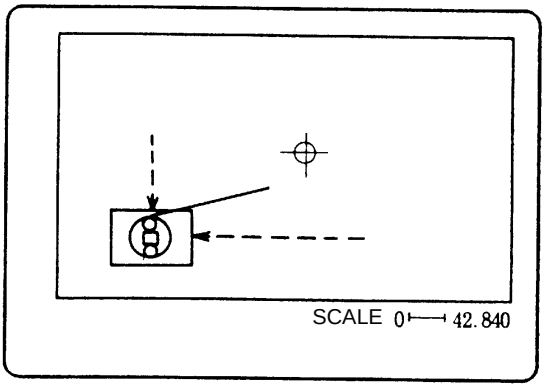
Press the menu key SCALE.



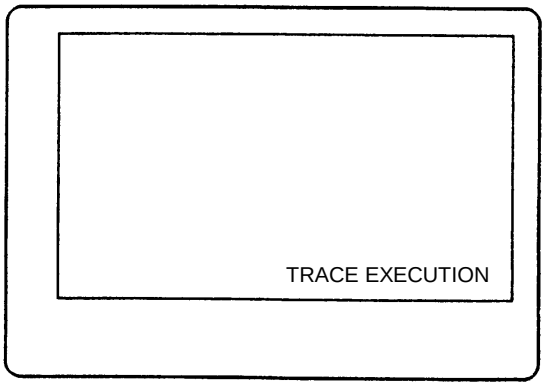
Using the cursor shift keys, move the cursor near the graphics center.



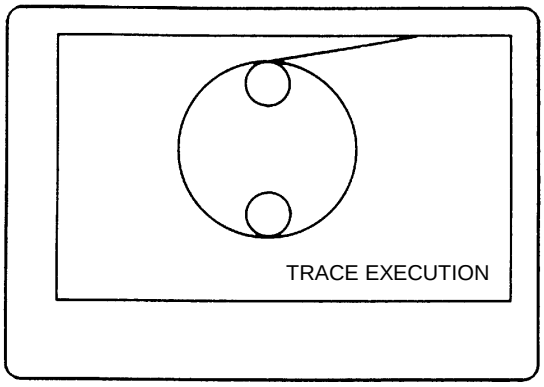
Using the = - key, fully enclose the graphics within the frame.



Using the INPUT key, fully enclose the graphics within the frame.



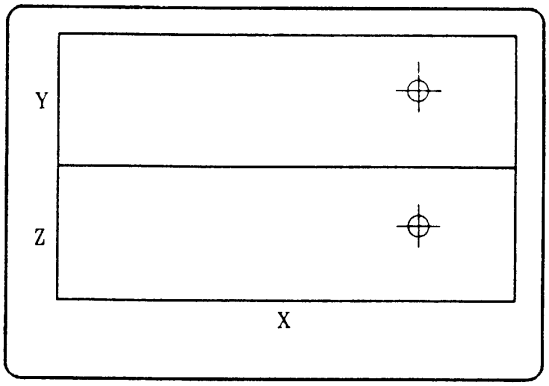
Draw the graphics again with the same program.



(3) Changing the scale and display position during the 2-plane display mode

1-plane and 3-dimensional display modes are as explained before, but in the 2-plane display mode the display positions of the upper and lower planes can be changed separately.

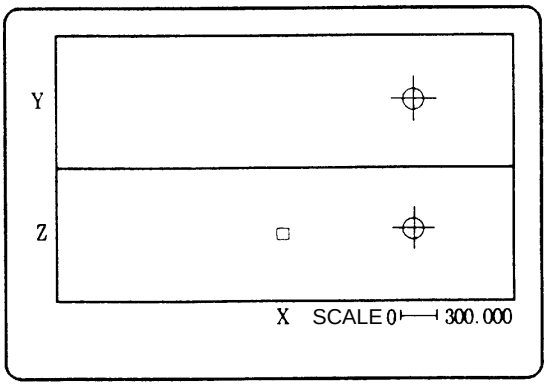
Although the scale of either plane can be changed in the 2-plane display mode, the same scale is always applied to the upper and lower planes. Similarly, the horizontal axis can also be changed on either plane.



Changeover between upper and lower planes

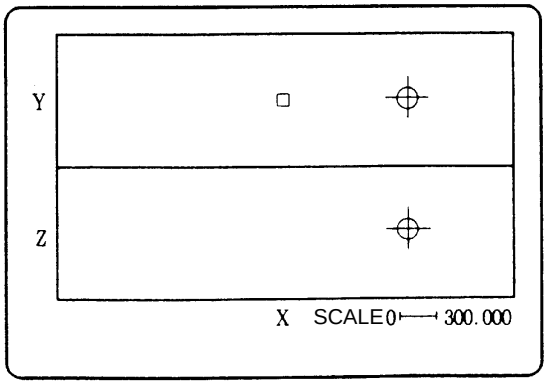
Press the menu key SCALE.

1) The cursor indicating the center appears in the center of the lower plane. The lower plane display range can now be changed.
2) The method for changing the scale and display position is the same as for 1-plane and 3-dimensional display described above.



Press the menu key SCALE again.

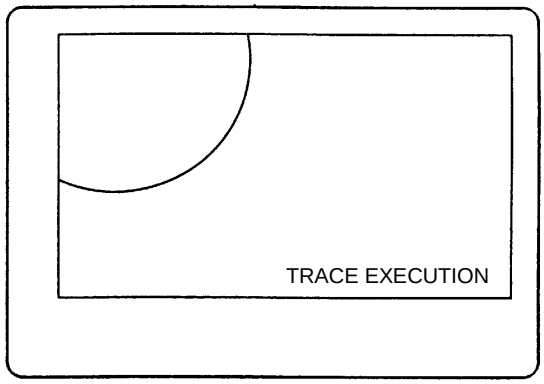
1) The cursor moves to the upper plane. The upper plane display range can now be changed.



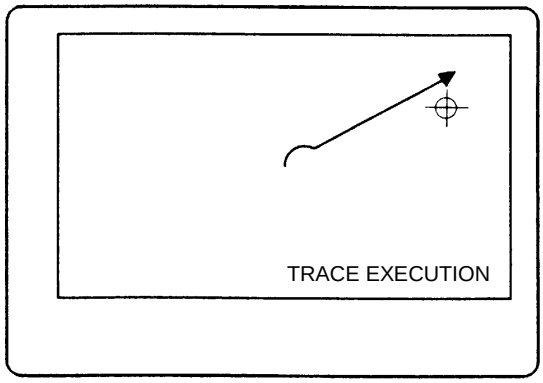
2.8.7 Standard Range (STANDARD)

When the menu key STANDARD is pressed, the machine's moveable range determined in setup parameters OT+ and OT– (stored stroke limit range) becomes the display range. The scale and display position are automatically changed. This function is useful if the drawing graphics abnormally shift out of the screen.

For example, assume the tool mark disappears from the screen during drawing.



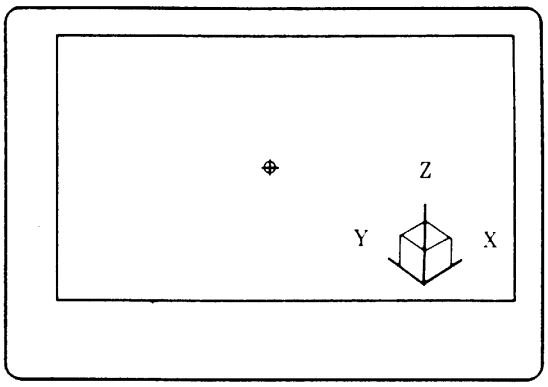
Press the menu key STANDARD.



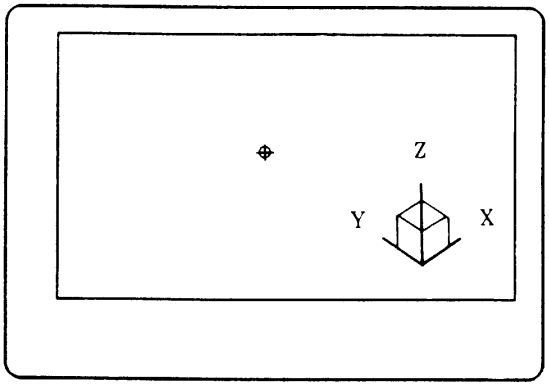
1) The display range is changed.
2) All already drawn graphics (if present) are erased, and a new drawing starts in the changed display range.

2.8.8 Rotate (ROTATION) (M system)

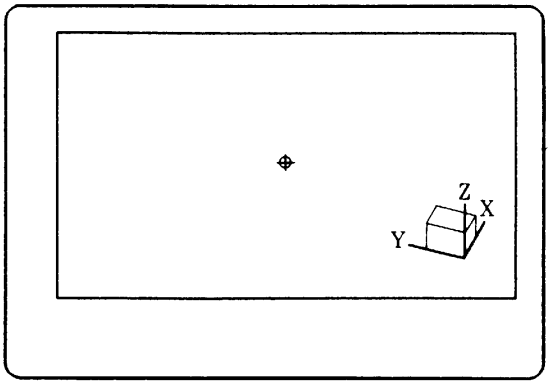
A 3-dimensional display can be rotated in any direction.



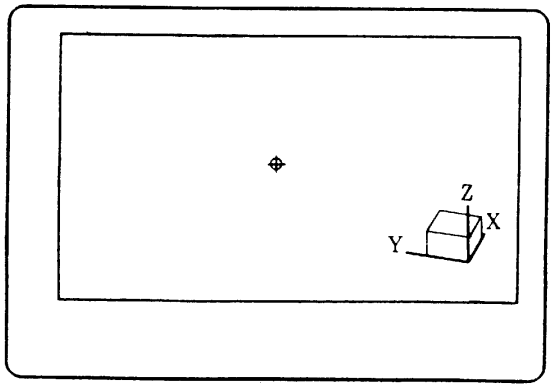
Press the menu key ROTATION.



Rotate the cube using the cursor keys ←, →, ↑, ↓. For example, press the → key to rotate the cube as shown in the figure at the right.



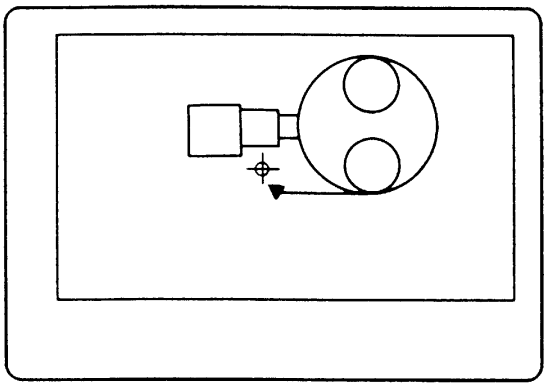
Press the INPUT key, or press the ROTATION key again.



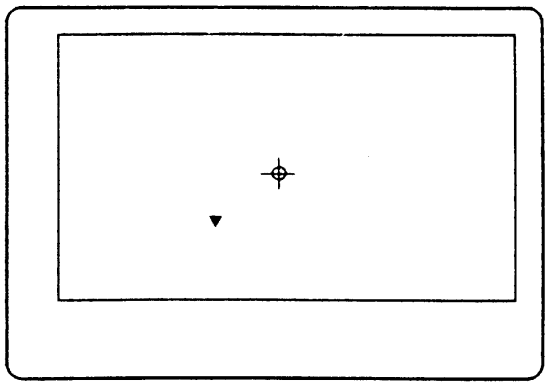
1) All already drawn graphics (if present) are erased.

2.8.9 Delete (ERASE)

When the menu key ERASE is pressed, graphics drawn with the trace mode and program check mode can be deleted.



Press the menu key ERASE. ☞

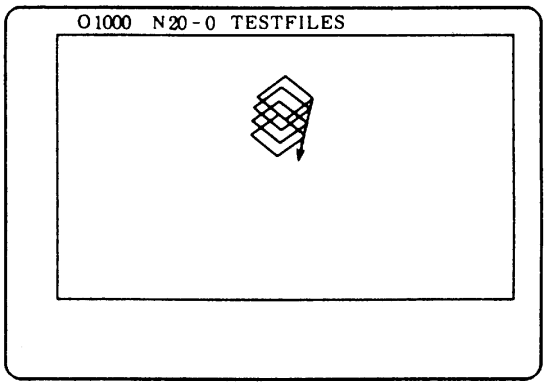


1) All already drawn graphics are deleted.

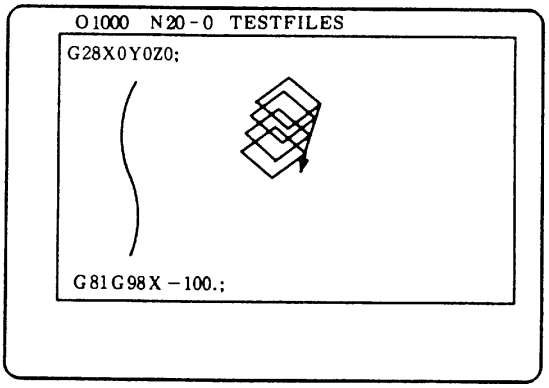
(Note 1) When graphics are deleted in the trace mode, drawing begins after completion of the deletion.

2.8.10 Program (PROGRAM)

When the menu key PROGRAM is pressed, the details of the program being drawn can be displayed. If these are not required, press the PROGRAM key again and the details will disappear.



Press the menu key PROGRAM.



1) The No., comments and machining program details of the program being executed appear.

3. Screen Operation When Using a Panel Computer

This section describes screen operations when a panel computer is used as a display.

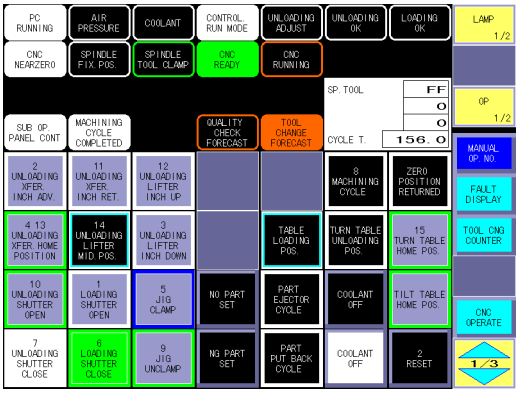
3.1 Screen Composition

3.1.1 Screen Transition

(1) Screen Transition When Power Is Turned ON

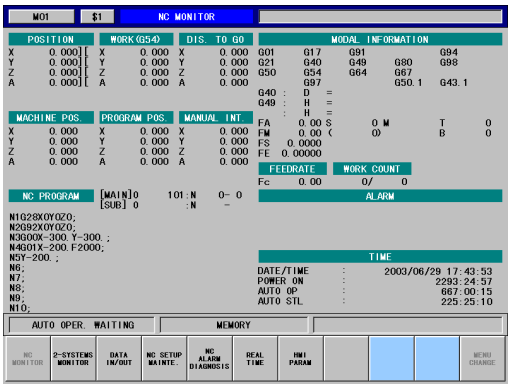
1) Turn the power ON.

➡ Operation screen appears after Windows ® starts.



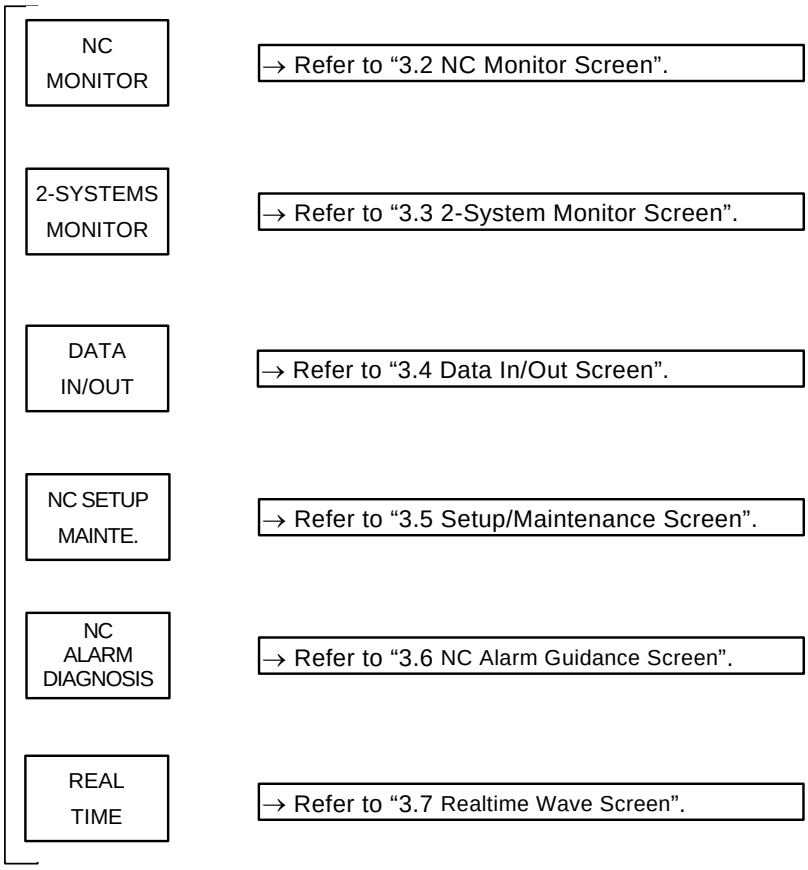
2) Press the CNC OPERATE. menu.

➡ NC MONITOR screen appears.



(2) Screen Transition Diagram

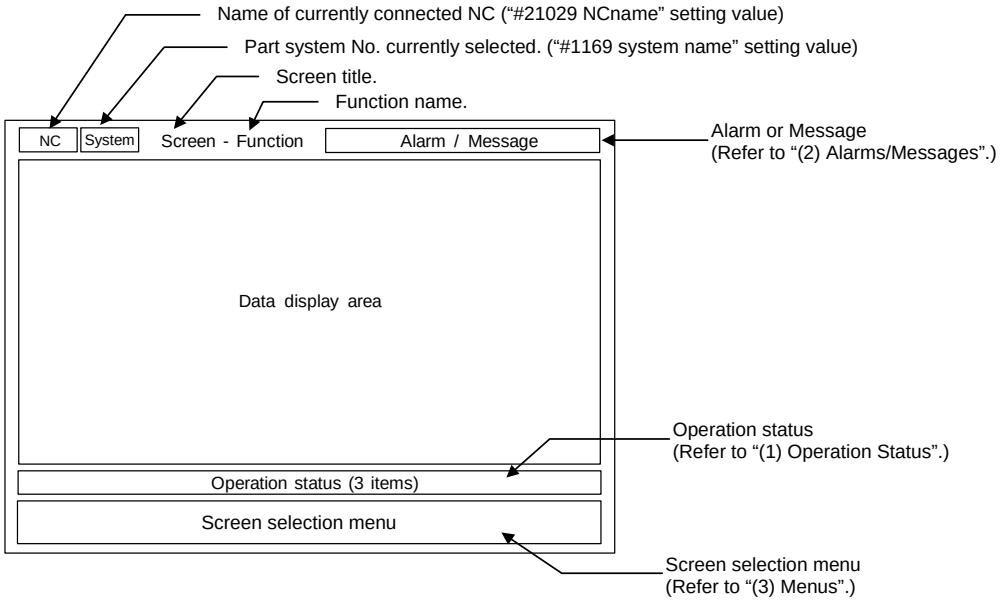
Following screens are provided.



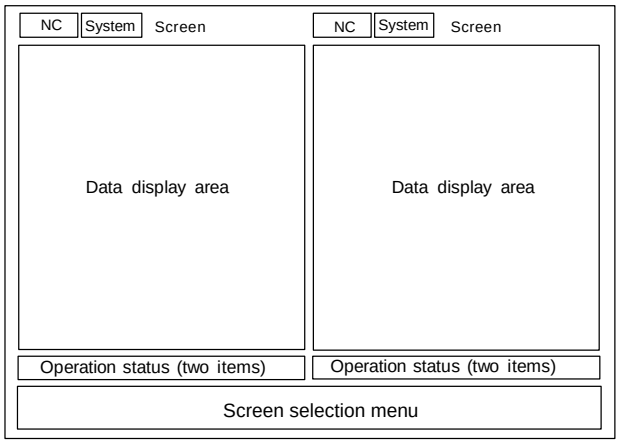
3.1.2 Display Composition

Display screen consists of the areas as shown below.

■ **NC MONITOR screen and DATA IN/OUT screen**



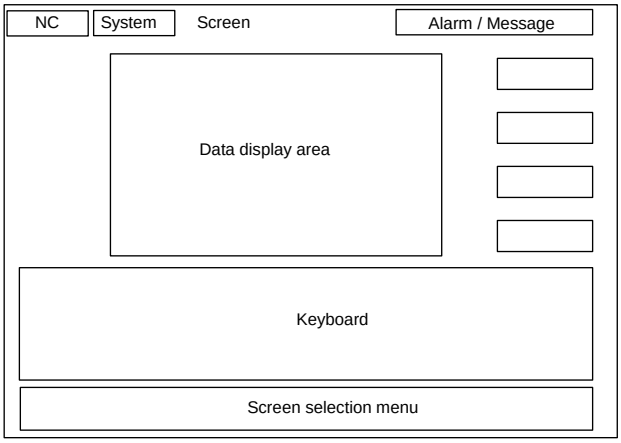
■ **2-SYSTEM MONITOR screen**



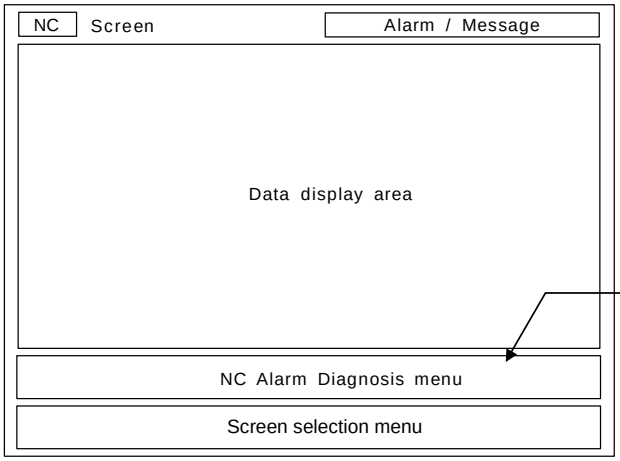
Arbitrary NC or part system information can be displayed on the left and right sides of the 2-SYSTEM MONITOR screen.

■ **SETUP/MAINTENANCE screen**

(Note) Refer to “3.5 Setup/Maintenance Screen” for details of SETUP/MAINTENANCE screens.



■ **NC ALARM GUIDANCE screen (WAVEFORM screen), REALTIME WAVE screen**



The menus used on the NC ALARM GUIDANCE screen and REALTIME WAVE screen are displayed.
(Refer to section "3.6 NC Alarm Guidance Screen" and “3.7 Wave Disp Screen” for details.)

(1) Operation Status

■ NC MONITOR screen and DATA IN/OUT screen

ST1	ST2	ST3

■ 2-SYSTEM MONITOR screen

ST1	ST2

The details of Operation status are as follows.

(a) ST1: Displays the NC status.

Operation Status	Color	
	Character	Back
EMERGENCY STOP	White	Red
RESET	Black	Gray
FEED HOLD STOP	White	Yellow
SINGLE BLOCK STOP	White	Yellow
AUTO OPER WAITING	Black	Gray
AUTO OPER RUNNING	White	Blue

(b) ST2: Displays the selected operation mode.

Operation Status	Color	
	Character	Back
MEMORY	Black	Gray
MDI	↑	↑
JOG	↑	↑
RAPID	↑	↑
HANDLE	↑	↑
ZP-RTN	↑	↑
STEP	↑	↑
MANUAL	↑	↑
INIT-SET	↑	↑
JOG+HANDLE	↑	↑

(c) ST3: Displays the other status.

Operation Status	Color		Remarks
	Character	Back	
TURN THE POWER ON AGAIN	White	Blue	Indicates that the set parameter will become valid when the NC power is turned ON again.

(Note) ST3 is displayed in NC MONITOR screen and DATA IN/OUT screen.

(2) Alarms/Messages

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.

The background color of the alarm/messages is gray.

The background changes to red when an error occurs in the communication with the NC.

Message	Details of message	Remedy
NC COMM. IMPOSSIBLE	An error has occurred in the communication with the NC.	• Check the connection between the NC and panel computer. (Cable connection, noise, etc.) • Check the NC and panel computer 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.

(3) Menus

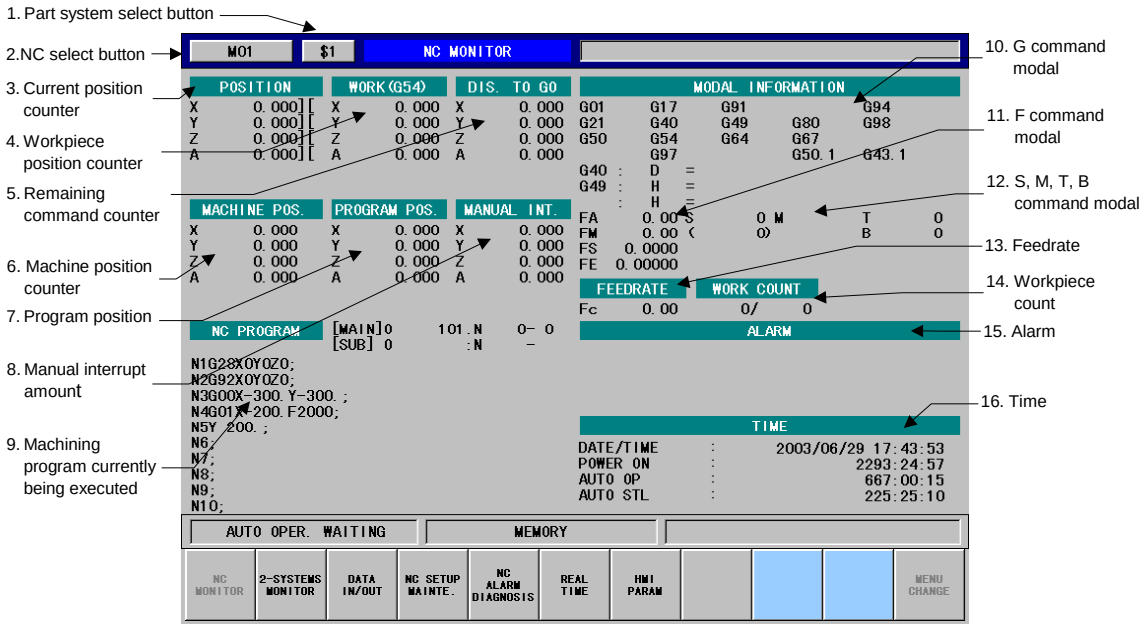
Following menus are available.

Menu	Details	Reference
NC MONITOR	Displays NC MONITOR screen.	3.2. NC Monitor Screen
2-SYSTEMS MONITOR	Displays 2-SYSTEM MONITOR screen.	3.3. 2-System Monitor Screen
DATA IN/OUT	Displays DATA IN/OUT screen.	3.4. Data In/Out Screen
NC SETUP MAINTE.	Displays SETUP/MAINTENANCE screen. **(Note 2)**	3.5. Setup/Maintenance Screen
NC ALARM DIAGNOSIS	Displays NC ALARM GUIDANCE screen.	3.6. NC Alarm Guidance Screen
REAL TIME	Displays REALTIME WAVE screen.	3.7 Realtime Wave Screen
OPERATION SCREEN	Displays OPERATION screen.	-
EXECUTE	Executes the selected function. Used in DATA IN/OUT screen.	3.4.5 Executing Function
MENU CHANGE	Used to change the menus. **(Note 3)**	-

(Note 1) If the color of menu character string is gray, the menu cannot be selected.
(Note 2) When the SETUP/MAINTENANCE screen is opened, this is invalidated (characters are displayed in gray), and the menu cannot be selected.
(Note 3) The MENU CHANGE display indicates that there is another menu opened.

3.2 NC Monitor Screen (For only the panel computer)

The NC information needed during NC running can be monitored in this screen.



(1) Display items

Display items	Details
1. Part system select button	This displays the name of the currently selected part system. A pull-down menu for selecting the part system will appear when this button is pressed.
2. NC select button	This displays the name of the selected NC. A pull-down menu for selecting the NC will appear when this button is pressed.
3. Current position counter	This displays the currently executed position and the status symbol when the position is at a specified position or status. #1 to #4 : 1st to 4th reference point position] [: Servo OFF state > < : Axis removal state MR : Mirror image
4. Workpiece position counter	This displays the G53 basic machine coordinate system and the G54 to G59 workpiece coordinate systems modal Nos. and the workpiece coordinate position in that workpiece coordinate system. The modal number such as P1 or P2 appears when the expanded workpiece coordinate system is used.
5. Remaining command counter	This displays the remaining distance of the movement command being executed during automatic start or automatic halt. (The remaining distance is the incremental distance from the current position to the end point of that block.)
6. Machine position counter	This displays the coordinate value of each axis in the basic machine coordinate system having a characteristic position, specified by the machine, as a zero point.

Display items		Details
7. Program position		[Current position] – Tool compensation amount = [Program position] This displays the value obtained by subtracting the tool offset amount for that axis from the position actually being executed for each axis.
8. Manual interrupt amount		This displays the amount moved with the manual mode while the manual absolute switch was OFF.
9. Machining program currently being executed		
	[MAIN]O ···	This displays the program No., sequence No. and block No. currently being executed.
	[SUB] O ···	When executing a subprogram, the subprogram program No., sequence No. and block No. are displayed.
	(PROGRAM)	The details of the machining program currently being executed are displayed. The characters of the block being executed are displayed in blue.
10. G command modal		The state of the G command modal currently being executed is displayed. The G command modal displayed differs depending on the model (lathe/ machining center).
	G01···G94···	The state of the G command modal currently being executed is displayed.
	G41: D1 =30.000: 0.040	The tool radius offset modal is displayed. Offset No. Shape offset amount for tool radius Tool radius wear amount
	G43: Z H20 =250.500: 0.240	The tool length offset modal is displayed. Offset axis name Offset No. Offset amount Tool length wear amount
11. F command modal		
	FA	The program command F modal value currently being executed is displayed.
	FM	The manual feedrate is displayed.
	FS	The synchronous feedrate is displayed.
	FE	The thread cutting feedrate is displayed.
12. S, M, T, B command modal		
	S ( )	The program command S modal value currently being executed is displayed. The value in parentheses indicates the actual spindle rotation speed. When the 2nd spindle is used, the modal value for the 2nd spindle is also displayed.
	M	The program command M modal value (max. four sets) currently being executed is displayed.
	T	The program command T modal value currently being executed is displayed.
	B	The program command 2nd miscellaneous function modal value currently being executed is displayed.

Display items		Details
13. Feedrate (FC)		During interpolation feed, the speed in the vector direction currently being moved in is displayed. During each axis independent feed, the speed of the axis with the highest speed is displayed.
14. Workpiece count		The data indicated the counted No. of workpieces is displayed on the left, and the maximum value (#8003 WRK LIMIT) is displayed on the right.
15. Alarm		The code and No. or message regarding the operation alarm, program error, MCP alarm, spindle alarm, servo alarm or system error is displayed. Up to four alarms can be displayed.
16. Time		
	DATE/TIME	The current time and date which are set to NC system are displayed. The year is displayed in the Christian era. The time is displayed in 24-hour style.
	POWER ON	The total time from when the NC power was turned ON until OFF is displayed.
	AUTO OP	The total cumulative time for each machining from when the automatic start button is pressed in the memory mode to when M02/M30 is issued or the reset button is pressed is displayed.
	AUTO STL	The total cumulative time during automatic start from when the automatic start button is pressed in the memory mode to when feed hold stop or block stop is issued or the reset button is pressed is displayed.

3.2.1 Selecting NC No. and Part System

When the multiple part systems are applied, the display of various information for one part system such as counters, modals, etc. can be changed to that for the other part system.
1st part system is selected when the power is turned ON.
When the single part system is applied, part system display cannot be changed.

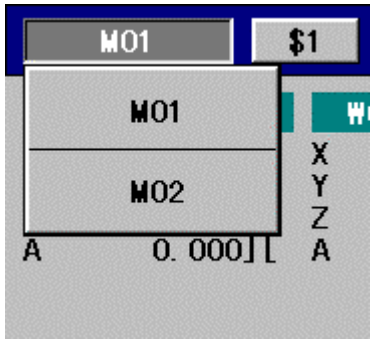
3.2.1.1 Selecting from the Pull-down Menu

(1) To select the NC

1)

Press the NC select button.
(Ex.) Press M01.

➡ A pull-down menu to select an NC will appear.

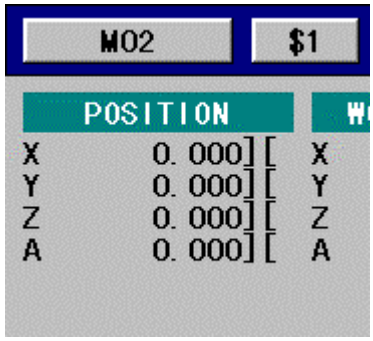


2)

Select the NC to be displayed from the pull-down menu.

(EX.) Select “M02”.

➡ The information for M02 controller are displayed in the screen.



(Note 1) After the NC is selected, the contents for the 1st part system will be displayed at first.
(Note 2) If there are eight or more NC device names that can be selected, six will appear in the pull-down menu. Press the (Cont. ▼) button at the very bottom of the pull-down menu to display the seventh and following devices.
(Note 3) This function can be used with the MELDAS C64T Version C or higher system.

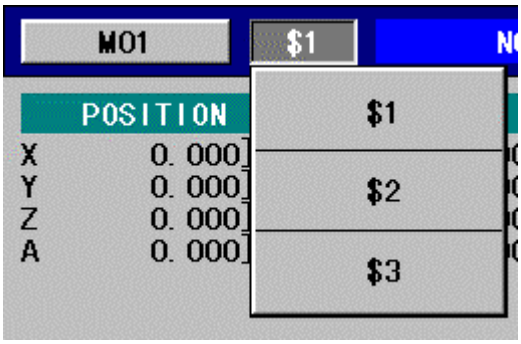
(2) Select a part system

1)

Press the part system select button.

(Ex.) Press $1.

A pull-down menu to select a part system will appear.

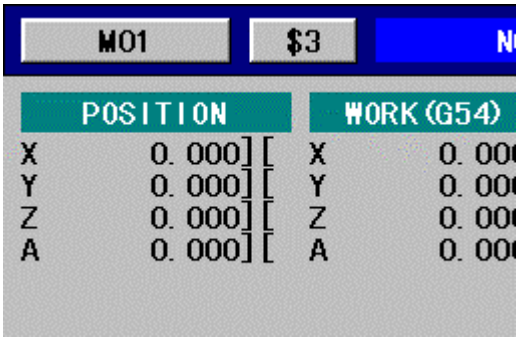


2)

Select the part system to be displayed from the pull-down menu.

(EX.) Select “$3”.

The information for the 3rd part system will appear.



(3) Closing the pull-down menu without changing settings

To close the pull-down menu to select an NC or a part system without changing current setting, perform one of the following operations:

- Press the same button which was pressed to display the pull-down menu. (NC select button or part system select button)
- Select the same name (NC or part system) as that displayed currently.
- Touch the panel wherever but outside of the pull-down menu.
- Press the other button (NC or part system).
  → The pull-down menu which has been displayed will close and a new pull-down menu will appear according to the pressed button.
- Change the screen with the screen select menu.
  → The pull-down menu will close and the selected screen will appear.

3.2.1.2 Cautions

(1) The pull-down menu includes the NC name or part system name which can be selected when pressing the button.

(2) The 1st part system will always be selected when selecting an NC.

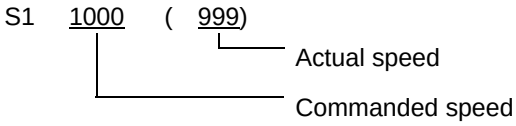
(3) Item “S” in modal information area is displayed one data for one part system.

$1 : S1

$2 : S2

$3 : S3 ....

[Display format]



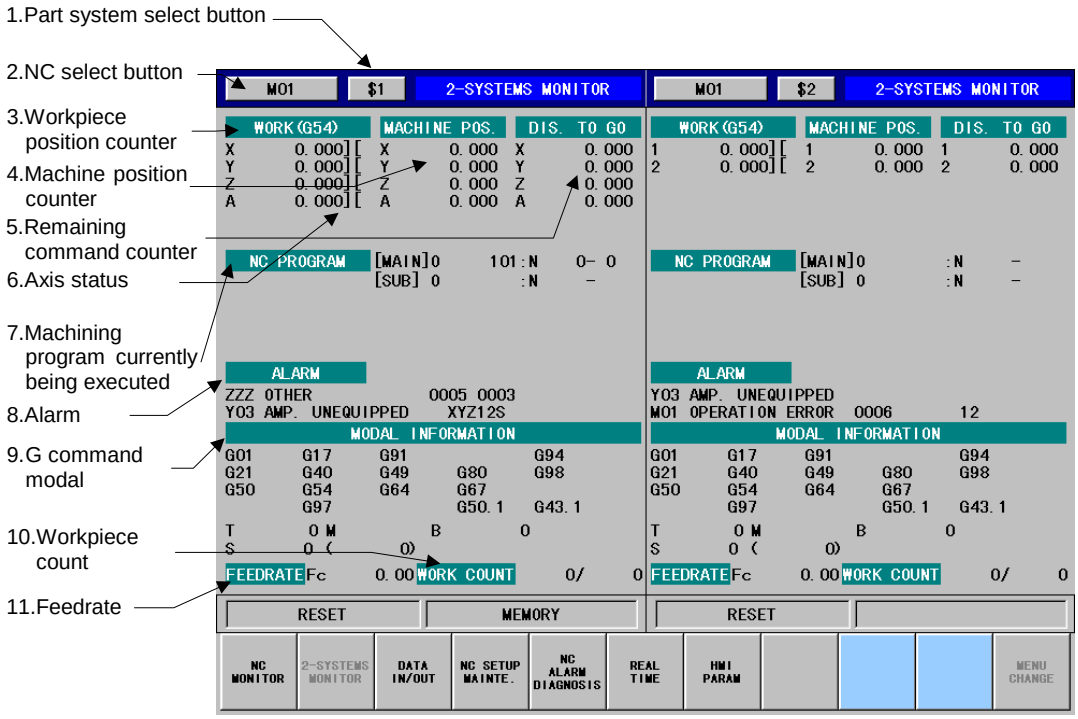
[Example]

- Single spindle : S 1000 (999)
- Multiple spindles : S1 1000 (999)

(4) Up to two alarm messages are displayed for each part system.

3.3 2-System Monitor Screen (For only the panel computer)

The information for random two NC screens can be monitored at the same time in this screen.



The display items and contents in the right area is the same as that in the left area (1. to 11.).
The information of the selected part system of the selected NC are displayed in respective area.

(1) Display items

Display items	Explanation
1. Part system select button	This displays the part system name currently selected. Press here to display a pull-down menu to select a part system. The operation methods for the left and right screens are the same as the NC MONITOR screen. Refer to section "3.2 NC Monitor Screen".
2. NC select button	This displays the NC name currently selected. Press here to display a pull-down menu to select an NC. The operation methods for the left and right screens are the same as the NC MONITOR screen. Refer to section "3.2 NC Monitor Screen".
3. Workpiece position counter	This displays the G54 to G59 workpiece coordinate systems modal Nos. and the workpiece coordinate value in that workpiece coordinate system. The modal number such as P1 or P2 appears when the expanded workpiece coordinate system is used.
4. Machine position counter	This displays the coordinate value of each axis in the basic machine coordinate system having a characteristic position, specified by the machine, as a zero point.
5. Remaining command counter	This displays the remaining distance of the movement command being executed during automatic start or automatic halt. (The remaining distance is the incremental distance from the current position to the end point of that block.)

Display items	Explanation
6. Axis status	This displays the currently executed position and the status symbol when the position is at a specified position or status. #1 to #4 : 1st to 4th reference point position] [: Servo OFF state > < : Axis removal state MR : Mirror image
7. Machining program currently being executed	
[MAIN] O...	This displays the program No., sequence No. and block No. currently being executed.
[SUB] O...	When executing a subprogram, the subprogram program No., sequence No. and block No. are displayed.
(PROGRAM)	The details of the machining program currently being executed are displayed. The characters of the block being executed are displayed in blue.
8. Alarm	The code and No. or message regarding the operation alarm, program error, MCP alarm, spindle alarm, servo alarm or system error is displayed. Up to two alarms can be displayed.
9. G command modal	The state of the G command modal currently being executed are displayed. The G command modal displayed differs depending on the modal (lathe/ machining center).
G01··· G94···	The state of the G command modal currently being executed is displayed.
G41: D1 =30.000: 0.040	The tool radius offset modal is displayed. Offset No. Shape offset amount for tool radius Tool radius wear amount
G43: Z H20 =250.500: 0.240	The tool length offset modal is displayed. Offset axis name Offset No. Offset amount Tool length wear amount
10.Workpiece count	The data indicated the counted No. of workpieces is displayed on the left, and the maximum value (#8003 WRK LIMIT) is displayed on the right.
11.Feedrate (FC)	During interpolation feed, the speed in the vector direction currently being moved in is displayed. During each axis independent feed, the speed of the axis with the highest speed is displayed.

3.3.1 Cautions

(1) The pull-down menu includes the NC name or part system name only which can be selected when pressing the button.

(2) After the NC and part system were selected once, screen displays the NC and part system selected last.

(3) The 1st part system will always be selected when selecting an NC.

(4) The NC and part system are selected on NC MONITOR screen and 2-SYSTEM MONITOR screen respectively. The setting in NC MONITOR screen has no influence on 2-SYSTEM MONITOR screen, and vice versa.

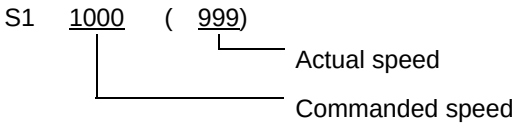
(5) Item “S” in modal information area is displayed one data for one part system.

$1 : S1

$2 : S2

$3 : S3 ....

[Display format]



[Example]

- Single spindle : S 1000 (999)
- Multiple spindles : S1 1000 (999)

(6) Up to two alarm messages are displayed for each part system.

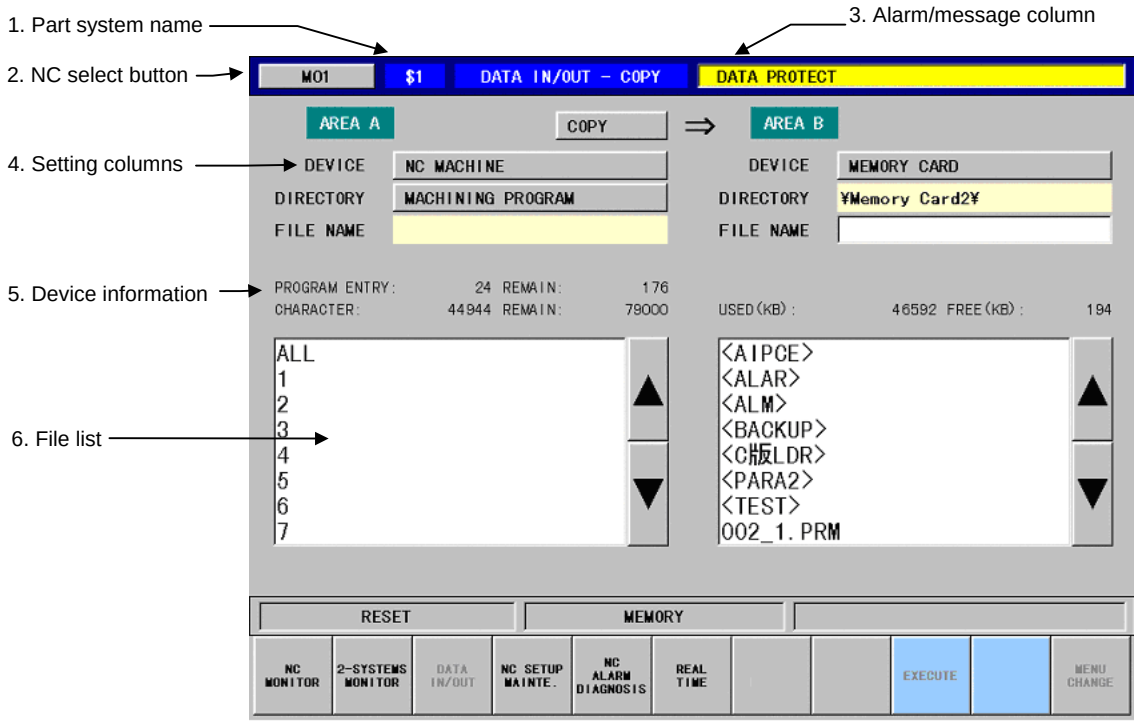
(7) The display items are the same as NC MONITOR screen and POSITION screen of SETUP/ MAINTENANCE screen except that listed above.

3.4 Data In/Out Screen (For only the panel computer)

In this screen, various NC data can be input or output between NC memory and user's compact flash card mounted on the panel computer (Windows® CE). The following data items are the target.

1. Machining programs (including fixed cycle programs)
2. Tool offset data
3. Parameters
4. Common variables
5. Workpiece offset data
6. Maintenance data
7. Operation history
8. MR-J2-CT parameters

The maintenance data is output as binary codes, and the other data is output as ASCII codes or Shift JIS codes.



(1) Display items

Display items	Details
1. Part system name	This displays the name of the selected part system.
2. NC select button	This displays the name of the selected NC. Press here to display a pull-down menu to select an NC.
3. Alarm/message column	This displays the first alarm that has occurred in the NC. The operation state is displayed while data input/output is executed.
4. Setting columns	Area A: The information of the source file are displayed and selected. To delete a file, designate the file to be deleted in area A. Area B: The information of the destination file are displayed and selected.
Function	The function to be executed is displayed and selected with pull-down menu.
Device	The target device name is displayed and selected with pull-down menu.
Directory	The target directory name is selected and displayed. When NC unit is selected : Pull-down menu When memory card is selected : Current directory
File name	The target file name or O No. is displayed. When machining programs are copied to the NC unit, or if the copy destination is the memory card, the copy destination file name can be designated from the keyboard.
5. Device information	This displays the information according to the selected device. PROGRAM ENTRY and REMAIN: Displays the number of user machining programs registered in the memory and remaining number of the programs that can be registered. CHARACTER and REMAIN: Displays the number of characters registered in the user machining programs and remaining number of characters that can be registered. A value in 250-character units is displayed in REMAIN column. USED and FREE: The used capacity and open capacity in the storage device, such as the memory card, etc. is displayed.
6. File list	The directories and files in the current directory are listed. Selecting ".." moves to upper directory. The list page can be scrolled up or down using the scroll button.

3.4.1 Selecting a Function

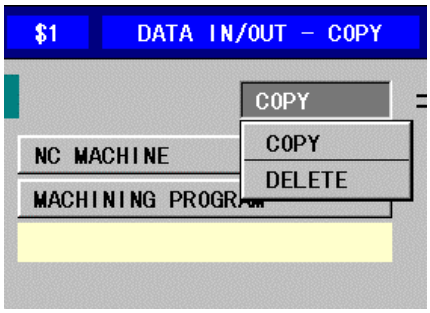
There are following functions in DATA IN/OUT screen.

Function	Details
COPY	Copies the file selected in Area A and pastes it in Area B.
DELETE	Deletes the file selected in Area A.

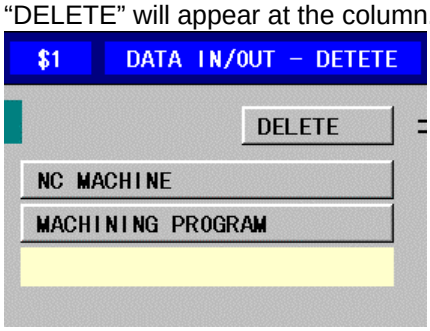
(1) Operation method (Changing from “COPY” to “DELETE”)

A pull-down menu appears when touching “FUNCTION” column.
Select a function in the pull-down menu.

1) Touch the “FUNCTION” column. → A pull-down menu will appear.



2) Select “DELETE” in the pull-down menu. → “DELETE” will appear at the column.



(2) Displayed contents

The displayed contents depend on the selected function.

Items	Details
Function name in title bar	Displays the selected function.
Device column (Area A)	"NC MACHINE"
Device column (Area B)	"MEMORY CARD". Note that "DRIVE C" will appear if there is a hard disk drive.
Directory column	When device is NC, "MACHINING PROGRAM" is displayed. When device is external peripheral device, the root directory of the device is displayed.
File column	(Blank)
File list	No file is selected. Files are listed from the top.

When “DELETE” is selected, columns such as the device, directory, file name, and file list in Area B cannot be selected.

3.4.2 Selecting a Device

Select a device to copy or delete files. The following device can be selected.

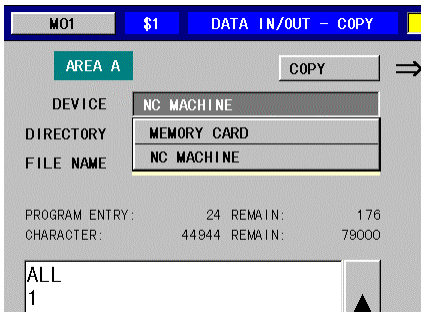
Device	Meaning
NC MACHINE	The NC unit displayed in the title bar is selected. Use the NC select button on the title bar to switch to another NC unit. (Refer to the section "3.2 NC Monitor Screen".)
MEMORY CARD	2nd card of mounted IC card slot. The 1st card is used by system and cannot be selected.

Select a device in Area A and B respectively.

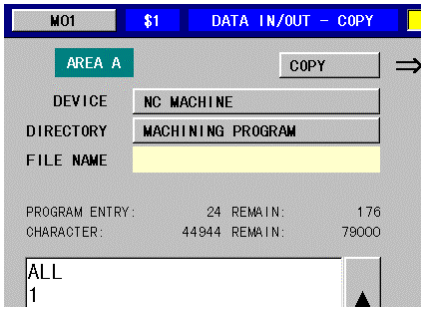
(1) Operation method

A pull-down menu appears when touching “DEVICE” column.
Select a device in the pull-down menu.

1) Touch the “DEVICE” column. → A pull-down menu will appear.



2) Select a device. → Selected device name will appear.



(2) Displayed contents

The displayed contents depend on the selected device.

Items	Details
Directory column	When device is NC, “MACHINING PROGRAM” is displayed. When device is external peripheral device, the root directory of the device is displayed.
File name column	(Blank)
File list	No file is selected. Files are listed from the top.

(3) Notes

- When no IC card is inserted in a slot, “MEMORY CARD” is not displayed.
- If the "NC MACHINE" is selected for area A with the COPY function, the area B device will be changed to "MEMORY CARD". Conversely, if "MEMORY CARD" is selected for area A, area B will be changed to "NC MACHINE".
  Note that if the system does not have a memory card, but has a hard disk drive, the contents of the hard disk will be displayed in the area B. At this time, "DRIVE C" will be displayed instead of the "MEMORY CARD" as the device name.

3.4.3 Selecting a Directory

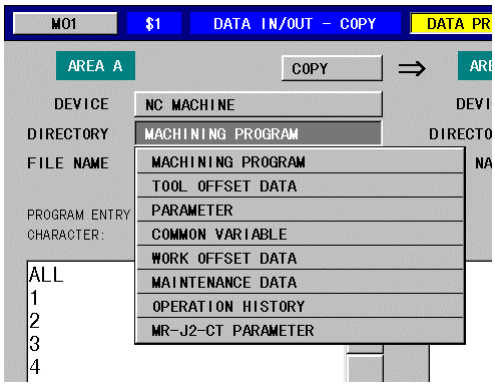
Select a directory including the target file.
Select a directory in Area A and Area B respectively.

The selecting method depends on the selected device (NC and external peripheral device).

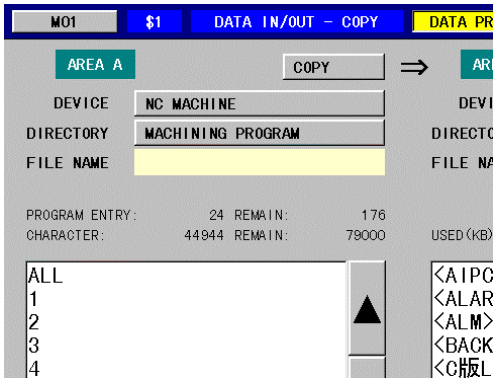
(1) When an NC is selected

A pull-down menu appears when touching “DIRECTORY” column.
Select a directory in the pull-down menu.

1) Touch the “DIRECTORY” column. ➡ A pull-down menu will appear.



2) Select a directory. ➡ Selected directory name will appear.
The files of selected directory are displayed in the "File List" column.



All the directories and their contents in the NC are as follows.

Directory names	Contents	File names to be displayed
MACHINING PROGRAM	Machining program	Files names are displayed with numbers following O. Files names are displayed in order. **(Note 1)**
	Machining program (batch)	ALL This appears when the following conditions are satisfied. • When the "Copy" function is selected • When the area A device is the "NC MACHINE" • When there are one or more machining programs
TOOL OFFSET DATA	Tool offset data	TOOL.OFS
PARAMETER	Parameters	ALL.PRM
COMMON VARIABLE	Common variables	COMMON.VAR
WORK OFFSET DATA	Workpiece offset data	WORK.OFS
MAINTENANCE DATA	Exceptional process history data	ILLEGAL.ERR
	PLC ladder	USERPLC.LAD
	R register data	RREG.REG
	C register data	CREG.REG
	T register data	TREG.REG
	Tool life management data (binary type)	TOOLLIFE.TLF
	Parameters (binary type)	PARAMET.BIN
	File system (binary type)	FILESYS.BIN
	Workpiece coordinate offset (binary type)	OFFSET.WRK
	Tool offset data (binary type)	OFFSET.TOL
OPERATION HISTORY	History data	TRACE.TRC
MR-J2-CT PARAMETER	MR-J2-CT parameter	MRJ2CT.PRA This appears when the following conditions are satisfied. • When the "Copy" function is selected • All the auxiliary axes are mounted on the NC unit.

(Note 1) If the Base specifications parameter "#1166 fixpro" is set to "1", the fixed cycle programs will be the target.

(2) When device is a storage device such as a memory card

Change directories following to the procedures as below.
“DIRECTORY” column displays the current directory.

Operation method : Touch the displayed item on the panel.

Items	Operation	Remarks
<..>	Moves to the upper directory	If "MEMORY CARD" is selected as the device, the directory "Storage Card2" is the highest and cannot be changed further.
<*Directory name*>	Moves to the directory of <*Directory*>	
Other file name	Selects a file	The current directory is selected.

Note that the directories are displayed preceding file names in the list if "MEMORY CARD" is selected as the device.
Directories and file names are displayed in ASCII character order.
(Example) A → AB → ... → 1 → 10 → 2 → 21 → 220 → 3 ... , etc.

3.4.4 Selecting a File

Select a file from the list.
When the "Copy" function is selected, if the copy destination (area B) is "MEMORY CARD" or if the copy destination is "NC MACHINE" and the directory is "MACHINING PROGRAM", the copy destination file name can be input from the keyboard.
Refer to section "3.4.6 Copying Files" for details on the copy source and copy destination file names.

3.4.5 Executing a Function

Press [EXECUTE] menu to execute the selected function.

[EXECUTE] menu is valid when the following conditions are satisfied.

Function	Area A			Area B		
	Device	Directory	File name	Device	Directory	File name
COPY	Select	Select	Select	Select	Select	**(Note 1)**
DELETE	Select	Select	Select			

(Note 1) [EXECUTE] menu is valid whether a file in Area B is selected or not.

(Note 2) If the data is protected with the data protection function, the [EXECUTE] menu will be invalid.

(1) Operation method

1)

Select [EXECUTE] menu. ➡ A dialogue box will appear.
When the "DELETE" function is selected, the file name will also appear.

2)

Select [YES]. ➡ The selected function will be executed.
When input/output is started:
A message indicating execution will appear.
When input/output is completed:
A message indicating completion will appear.

(2) Notes

- When COPY function is used, if a file name in Area A exists in the directory selected in Area B, a dialogue box will appear to confirm the rewriting.
  Selecting [YES] starts copying.
- When data input/output is completed, File name column will be blank and list selection will be canceled.
- Changing to the other screens cannot be performed during data input/output.

3.4.6 Copying Files

This function is used to copy various data from the NC unit to a storage device such as a memory card, or vice versa.
The following data in the NC unit can be copied.

1. Machining programs (fixed cycle programs)
2. Tool offset data
3. Parameters
4. Common variables
5. Workpiece offset data
6. Maintenance data
7. Operation history
8. MR-J2-CT parameter

<Precautions>

(1) When overwriting a file in a storage device such as a memory card, the data cannot be copied if the file attributes are read-only.
(2) When copying data to the NC unit directory and designating a directory other than a machining program, designate the file name of both the copy source and copy destination.
(3) The copy source and copy destination file names are changed according to the following rules.
FILE1 to FILE19 indicate random file names.

Directory names	File names in NC unit	Direc-tion	Copy destination selection state	Name of file in memory card
Machining programs (→ Refer to (4).)	ALL	→	Not selected (not input)	ALL.PRG
	ALL	→	Selected (input)	FILE1
	10	→	Not selected (not input)	10.PRG
	10	→	Selected (input)	FILE1
	20	←	Selected	FILE2 (Designate "20" for the copy destination when 010; is set in the file.)
	10	←	Not selected (not input)	FILE3 (010; designated at head of file)
Tool offset data	TOOL.OFS	→	Not selected (not input)	TOOL.OFS
	TOOL.OFS	→	Selected (input)	FILE4
	TOOL.OFS	←	Selected	FILE4
Parameters	ALL.PRM	→	Not selected (not input)	ALL.PRM
	ALL.PRM	→	Selected (input)	FILE5
	ALL.PRM	←	Selected	FILE5
Common variables	COMMON.VAR	→	Not selected (not input)	COMMON.VAR
	COMMON.VAR	→	Selected (input)	FILE6
	COMMON.VAR	←	Selected	FILE6
Workpiece offset data	WORK.OFS	→	Not selected (not input)	WORK.OFS
	WORK.OFS	→	Selected (input)	FILE7
	WORK.OFS	←	Selected	FILE7

Directory names	File names in NC unit	Direc-tion	Copy destination selection state	Name of file in memory card
Maintenance data	ILLEGAL.ERR	→	Not selected (not input)	ILLEGAL.ERR
	ILLEGAL.ERR	→	Selected (input)	FILE8
	ILLEGAL.ERR	←	Selected	FILE8
	USERPLC.LAD	→	Not selected (not input)	USERPLC.LAD
	USERPLC.LAD	→	Selected (input)	FILE9
	USERPLC.LAD	←	Selected	FILE9
	RREG.REG	→	Not selected (not input)	RREG.REG
	RREG.REG	→	Selected (input)	FILE10
	RREG.REG	←	Selected	FILE10
	CREG.REG	→	Not selected (not input)	CREG.REG
	CREG.REG	→	Selected (input)	FILE11
	CREG.REG	←	Selected	FILE11
	TREG.REG	→	Not selected (not input)	TREG.REG
	TREG.REG	→	Selected (input)	FILE12
	TREG.REG	←	Selected	FILE12
	TOOLLIFE.TLF	→	Not selected (not input)	TOOLLIFE.TLF
	TOOLLIFE.TLF	→	Selected (input)	FILE13
	TOOLLIFE.TLF	←	Selected	FILE13
	PARAMET.BIN	→	Not selected (not input)	PARAMET.BIN
	PARAMET.BIN	→	Selected (input)	FILE14
	PARAMET.BIN	←	Selected	FILE14
	FILESYS.BIN	→	Not selected (not input)	FILESYS.BIN
	FILESYS.BIN	→	Selected (input)	FILE15
	FILESYS.BIN	←	Selected	FILE15
	OFFSET.WRK	→	Not selected (not input)	OFFSET.WRK
	OFFSET.WRK	→	Selected (input)	FILE16
	OFFSET.WRK	←	Selected	FILE16
	OFFSET.TOL	→	Not selected (not input)	OFFSET.TOL
	OFFSET.TOL	→	Selected (input)	FILE17
	OFFSET.TOL	←	Selected	FILE17
Operation history	TRACE.TRC	→	Not selected (not input)	TRACE.TRC
	TRACE.TRC	→	Selected (input)	FILE18
MR-J2-CT parameter (→ Refer to (5).)	MRJ2CT.PRM	→	Not selected (not input)	MRJ2CT.PRM
	MRJ2CT.PRM	→	Selected (input)	FILE19
	MRJ2CT.PRM	←	Selected	FILE19

(4) When copying machining programs, the O No. (NC unit side) and file name (memory card side) are determined according to the following rules.

NC unit directory	Copy source input state	Copy destination input state	File name or O No. after copying
Copy source (NC to card)	Input (selected)	Not input (not selected)	An extension (.prg) is added to the input O No.
	Input (selected)	Input (selected)	The name designated as the copy destination file name is used.
	Not input (not selected)		Copying is not possible. The EXECUTE menu is invalid.
Copy destination (Card to NC)	Input (selected)	Not input (not selected)	The O No. used in the copy source file is used.
	Input (selected)	Input (selected)	The machining program with the first O No. in the copy source file is used as the O No. designated as the copy destination and is stored in the NC unit. Note that if there are multiple machining programs in one file, the machining programs with the second and following O Nos. used in the file are used as the machining program No. even after copying.
	Not selected		Copying is not possible. The EXECUTE menu is invalid.

(5) MR-J2-CT parameter COPY function

Inputting the MR-J2-CT automatic tuning parameters
Whether to input or ignore the automatic tuning parameters depends on the combination of the automatic tuning selection parameter (auxiliary axis parameter #7 ATU) input data and NC unit value.

Table of automatic tuning selection parameter combinations

Selection of input data automatic tuning	Selection of NC side automatic tuning	Input of automatic turning parameters
Selected (0 or 1)	Selected (0 or 1)	Ignore
Not selected (2)	Selected (0 or 1)	Input
Selected (0 or 1)	Not selected (2)	Input
Not selected (2)	Not selected (2)	Input

(Refer to the correspondence of the MR-J2-CT parameters and N No. explained in the "MELDAS64 Series MR-J2-CT Link Specifications" (BNP-B3941) for details on the automatic tuning parameters.)

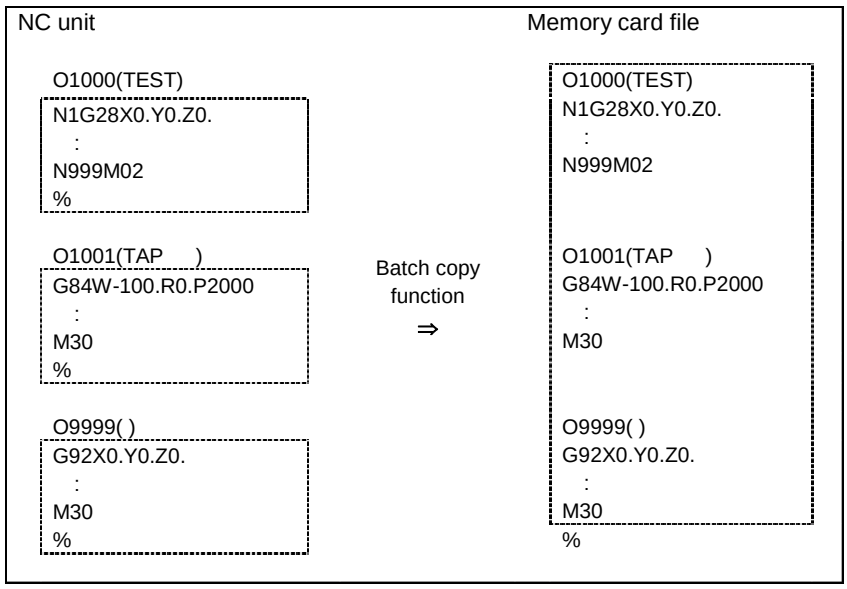
(Note 1) Some MR-J2-CT parameters are validated only when the power is turned OFF and ON after inputting the parameter.

(Note 2) When the MR-J2-CT parameters are returned, the auxiliary axis' absolute position data can also be returned. Note that the data cannot be returned for the rotary axis, so after turning the power OFF and ON, initialize the auxiliary axis.
If MR-J2-CT parameters backed up with another machine are input, the absolute position zero points of all auxiliary axes will deviate. Initialize all auxiliary axes after turning the power OFF and ON.

3.4.6.1 Examples of Operation]

(1) Copying multiple machining programs in a batch (NC unit to memory card)

When "ALL" is designated for the copy source, all machining programs can be copied to one file.

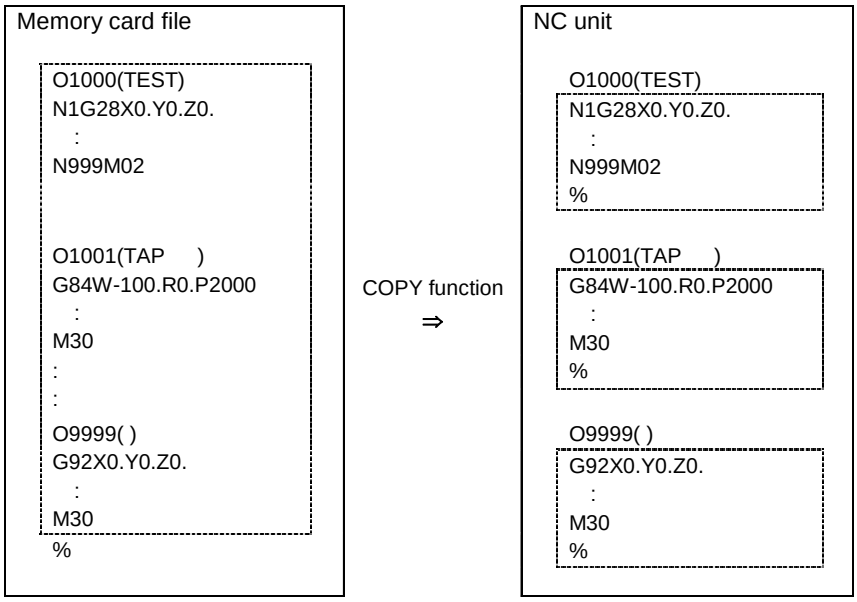


<Precautions>

- A message confirming the operation will first appear before the files are copied. Copying will start only when execution is instructed in respect to the confirmation message.
- When the files that already exist in the copy destination are to be overwritten, a confirmation message will first appear. The files will be overwritten when YES is selected.
- When the files to be overwritten are in a storage device such as a memory card, the data cannot be copied if the file attributes are read-only.
- While the machining programs are copied, if there is a file that cannot be copied because of the data protection function, the O No. and a confirmation message will appear. The file will be created with the data that was copied successfully before the protected file. If the first file could not be copied, a file containing only "%" will be created.

(2) Copying several machining programs in a file (Memory card to NC unit)

A file containing several machining programs in one file, such as a file created with batch backup, can be copied to the NC unit with the COPY function.



When a file created with batch backup is copied, the operation will differ as shown below according to the selection state of the copy destination file.

Copy destination file	Machining program in file	Operation
O No. selected	Head machining program	The head O No. in the file is ignored, and the selected machining program is overwritten.
	Second and following machining programs	The machining programs are copied according to the O numbers in the file.
Not selected	All machining programs	The machining programs are copied according to the O numbers in the file.

<Precautions>

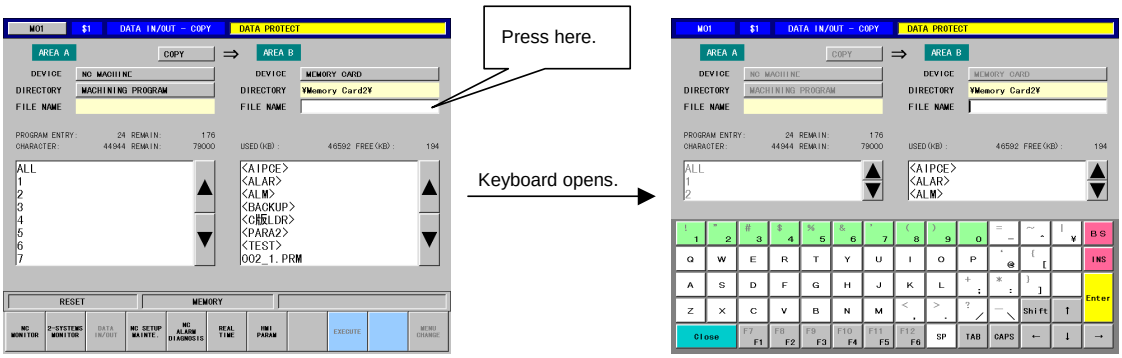
- If machining programs already exist, a confirmation message will appear before overwriting. The files will be overwritten only when the overwrite execution is instructed. If overwrite is not instructed, copying will continue from the next O No. in the file.
- The "%" line at the head of the file is ignored. ("%" is inserted for an RS-232C output file, etc.)
- If the O No. does not appear even once in the file, a message indicating that copying failed will appear.
- While the machining programs are copied, if there is a file that cannot be copied because of the data protection function, the O No. and a message will appear. If YES is selected, copying will continue from the next O No. in the file.

3.4.6.2 Keyboard Function

The keyboard can be used to designate the copy destination file name in the COPY function.
The keyboard function can be used under the following conditions.

- When the copy destination is "MEMORY CARD"
- When the copy destination is "NC MACHINE" and the directory is "MACHINING PROGRAM"

The keyboard will open when the file name input file din area B is pressed.



The data set in the file name input column will be input when the Enter key is pressed, and the keyboard will close.

(1) Explanation of each key 1 (Character keys)

Key	Operation
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 SP + - _ * /	Each character is input at the cursor position.
¥	"¥" is input at the cursor position. (This is for future expansion and cannot be used for the file name.)
.	"." is input at the cursor position. Use this as the delimiter between the memory card file name and extension.
1 2 3 4 5 6 7 8 9 0	Use these to input the machining program's O No. and to input the directory name and file name in the storage device, such as the memory card.
! # $ ' ( ) ` = , [ ]	These characters can be input when the SHIFT key is pressed.

(2) Explanation of each key 2 (Special keys)

Key	Operation
Enter	The data set in the file name input column is input and the keyboard is closed.
CLOSE	The data set with the keyboard is canceled, and the keyboard is closed.
Shift	The keyboard enters the shift state, and the characters on the upper left of the key can be input. The shift state is canceled if this key is pressed again or a character is input.
BS	The character before the cursor is deleted.
← →	The cursor is moved one character forward or backward.
|← →| ↑ ↓	These keys are invalid.
CAN C•B	All characters in the file name input column are deleted.

(3) Explanation of file name list key

Key	Operation
▲ ▼	The list of file names is scrolled and displayed.

(4) Precautions

- The function select button, device name select button and directory select button cannot be used while the keyboard is open.
- When the file is selected from the list and then the file name input column is pressed, the keyboard will appear with the cursor at the end of the file name.
- The directory can be changed by pressing the directory name.
- If the file name is pressed in the list of file names in area B, the current keyboard inputs will be canceled, and the file name will be input in the file name input column.
- If the CLOSE key is pressed, the data input from the keyboard will be deleted from the file name input column and the keyboard will be closed.
  The file name input area will return to the state before the keyboard was opened. However, the directory changed while the keyboard was opened will not return to the original directory.
- An error will occur if the file name contains "\".
- If the file name input area is touched ("dragged"), the characters will be highlighted. If a key is pressed in this state, the highlighted area will be replaced with the pressed key.

3.4.7 Deleting a File

Files in the NC unit or a storage device, such as a memory card, can be deleted.
Only machining programs can be deleted from the NC unit.
All files in the storage device, such as the memory card, can be deleted.
(Note that protected data cannot be deleted.)

(Note) Directories, read-only files and hidden files cannot be deleted. Set important files in the memory card as a read-only file or a hidden file.

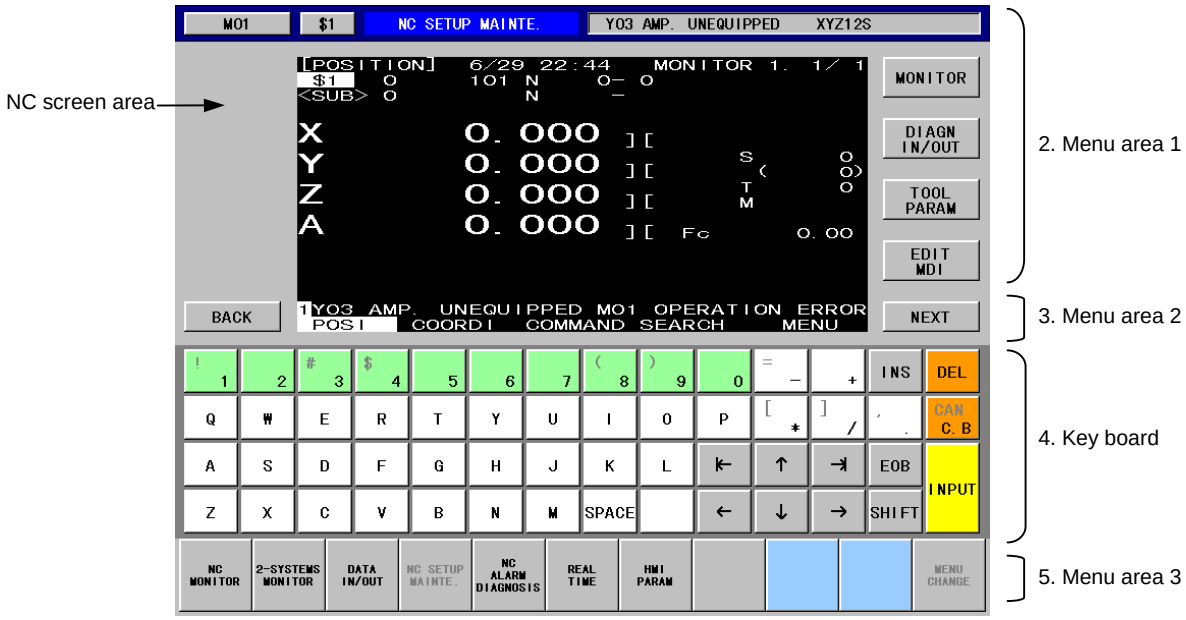
3.4.8 Data Protection Function

The data protection keys 1, 2 and 3, and the edit lock B and C data protection functions work in the same manner as the conventional input/output function.
Refer to the Appendix for details on the data protection function.

<table>
<tr><td colspan="3">Data protection key 1
(Input/output and delete prohibited when KEY1 is OFF)</td><td>Tool offset data, tool life management data, workpiece offset data</td></tr>
<tr><td colspan="3">Data protection key 2
(Input/output and delete prohibited when KEY2 is OFF)</td><td>Parameters, common variables, MR-J2-CT parameters</td></tr>
<tr><td colspan="3">Data protection key 3
(Input/output and delete prohibited when KEY3 is OFF)</td><td>Machining program A 1 to 7999
Machining program A 10000 to 99999999</td></tr>
<tr><td></td><td colspan="2">Edit lock B
(Input/output and delete prohibited when "EDIT LOCK B" is set to 1)</td><td>Machining program B 8000 to 8999
(User standard subprograms)</td></tr>
<tr><td></td><td></td><td>Edit lock C
(Input/output and delete prohibited when "edlk-c" is set to 1)</td><td>Machining program C 9000 to 9999
(Machine maker custom programs)</td></tr>
</table>

3.5 Setup/Maintenance Screen (For only the panel computer)

In this screen, the various information which are needed to setup and maintain the machine and NC system are displayed and set.
When the panel computer is used, SETUP/MAINTENANCE screen appears.
When the NC-dedicated display unit or A985GOT is used, refer to the section “2.1 Setting and Display Unit Operation” and onwards.



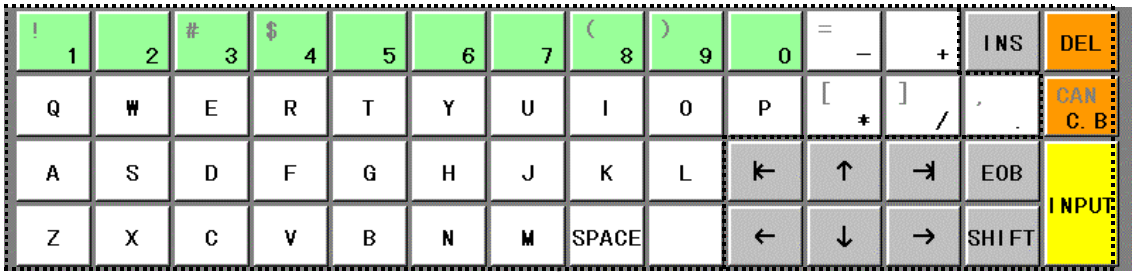
(1) Display items

Display items	Details
1. NC screen area	The functions such as position monitoring, alarm diagnosis, tool offset/parameters, and program edit. They have equivalent function as the screens of NC-dedicated display unit. Note that graphic trace function and PLC ladder edit/monitor function are not provided.
2. Menu area1	The following keys used to select the display function. These keys are equivalent to Function Select Key of NC-dedicated display unit.
MONITOR	Selects position monitoring function. Refer to “2.2 Monitor”.
DIAGN IN/OUT	Selects alarm diagnosis function. Refer to “2.7 Diagnosis”.
TOOL PARAM	Selects tool offset function and parameter setting. Refer to “2.3 Tool Offset” and “2.4 Parameters”.
EDIT MDI	Selects program edit function. Refer to “2.5 Program”.
3. Menu area2	
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.
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. Key board	This is used to set a data in NC screens. Refer to “(2) Composition of displayed keyboard“ for details.
5. Menu area3	Displays the other screen by the panel computer.

(2) Composition of displayed keyboard

Composition of the displayed keyboard is shown below.

Data setting keys



Special keys

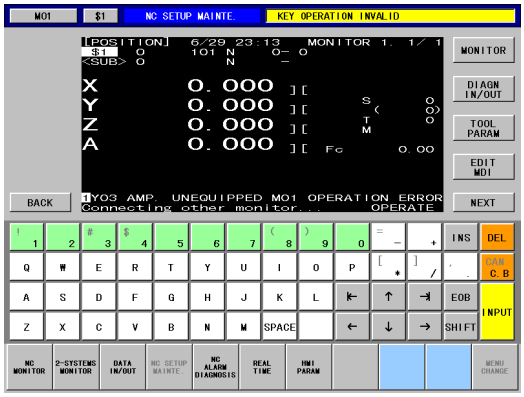
Key type	Keys	Function
Data setting keys	A ~ Z ! 1 2 # 3 $ 4 5 6 7 (8) 9 0 SPACE = - + * / , . [ ]	These keys are used to set alphabetic characters (only capital letters), numbers, space, and symbols, etc. The characters in gray can be set after touching SHIFT key.
Special keys	↑ ↓ ← →	Moves a cursor 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.

(3) Precautions at connecting with both the NC-dedicated display unit and the panel computer

A message appears on the panel computer's title bar and menu area 2 when an NC-dedicated display unit is connected. The menu keys and the keys on the panel computer's keyboard such as data setting keys and special keys became invalid. The keys pressed on an NC-dedicated display unit or the other display unit are valid. To validate the key operations on the panel computer, press the OPERATE menu.
Screen select keys on the bottom of the screen such as NC MONITOR and 2-SYSTEMS MONITOR are available.

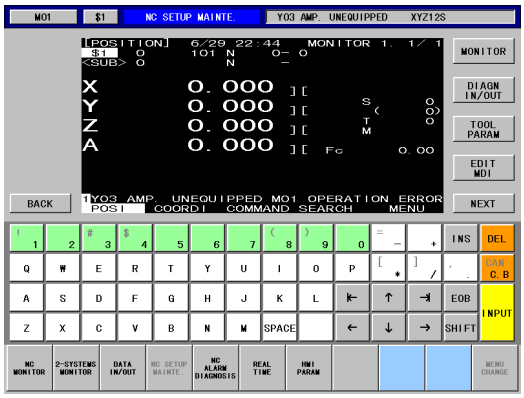
1)

Connect with NC-dedicated display unit.	➡	A message will appear on the displayed keyboard of SETUP/MAINTENANCE screen. Menu keys and keys of data setting keys and special keys are invalid. Part systems cannot be changed while the key operation is invalid. (Only for SETUP/MAINTENANCE screen.)



2)

Press the OPERATE menu.	➡	Key operations with this display unit (the panel computer) will be enabled.

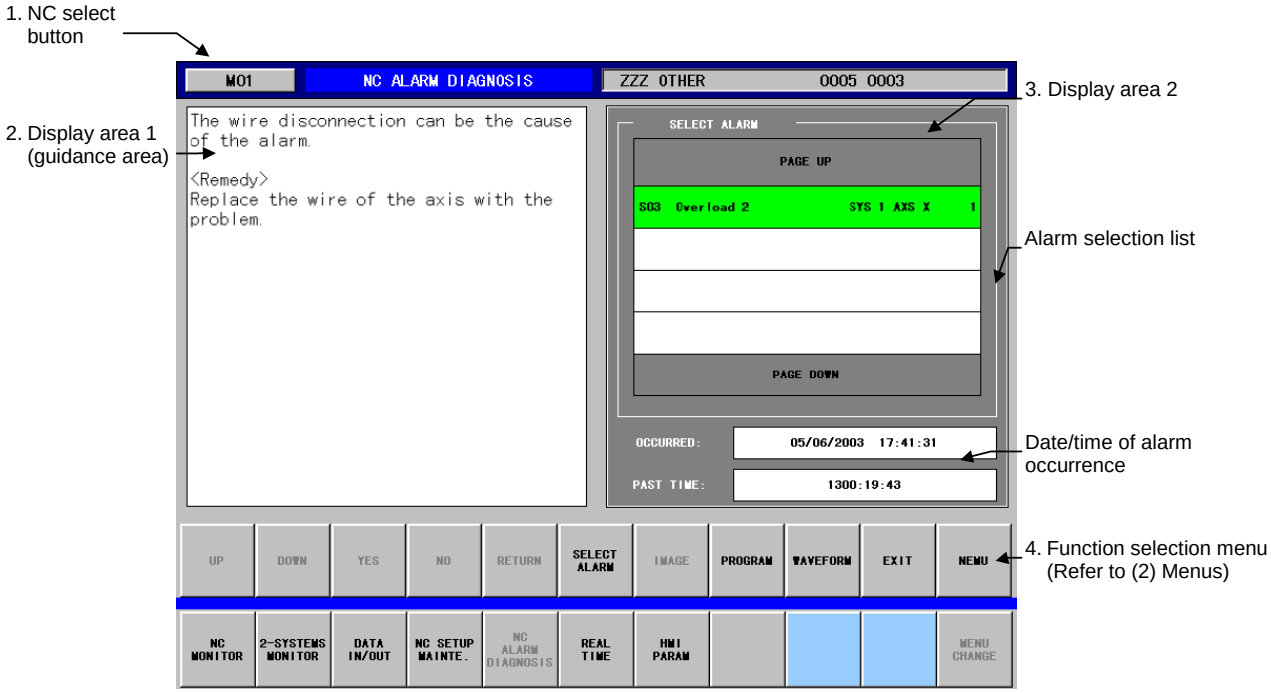


If the OPERATE menu is pressed on another display unit, that display unit will be validated.

3.6 NC Alarm Guidance Screen (For only the panel computer)

3.6.1 NC Alarm Guidance Screen

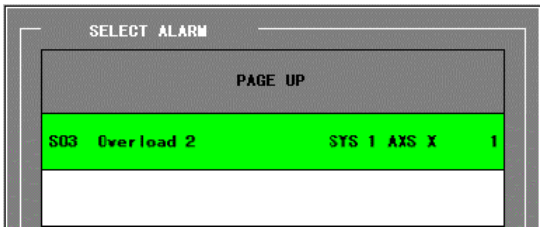
Guidance corresponding to the currently occurring alarm is displayed on this screen.
By using this guidance information, the cause of the alarm can be pinpointed from the speculated factors, and the remedy methods can be determined.
If several alarms occur simultaneously, the current guidance can be canceled, and the guidance for other alarms can be displayed.
The causes of past alarms can be listed, making it possible to find the cause based on past cases.
When this screen is opened, the details displayed last will appear if the power has not been turned off.



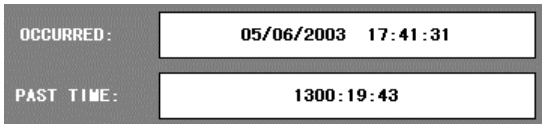
(1) Display items

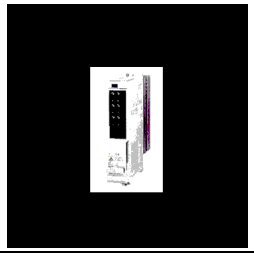
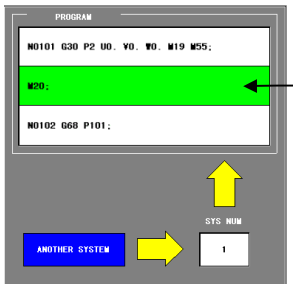
Display items	Details
1. NC select button	The name of the selected NC is displayed. Press here to display a pull-down menu to select an NC. The operation method is the same as the NC MONITOR screen.
2. Display area 1 (guidance area)	Troubleshooting guidance is displayed. The details corresponding to the alarm selected from the alarm selection list are displayed.
3. Display area 2	The display differs according to the function selection menu. When this screen is selected, the alarm selection list and date/time of alarm occurrence will appear.
4. Function selection menus	When a function is selected and the menu is pressed, the display will be highlighted and the function setting will be enabled. Refer to the next page for explanations on each menu.

(2) Menus

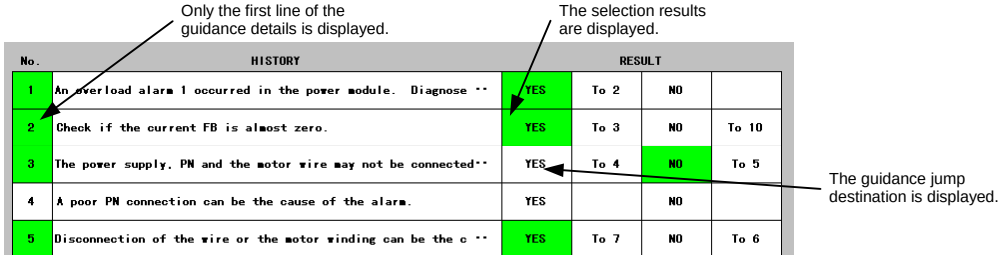
<table>
<tr><th>Menu</th><th>Details</th></tr>
<tr><td>UP
DOWN</td><td>The guidance display page is scrolled up (↑) or down (↓).</td></tr>
<tr><td>YES
NO
RETURN</td><td>YES : Press when the guidance details match.
NO : Press when the guidance details do not match.
RETURN : The previous guidance will appear.</td></tr>
<tr><td rowspan="3">SELECT ALARM</td><td> 

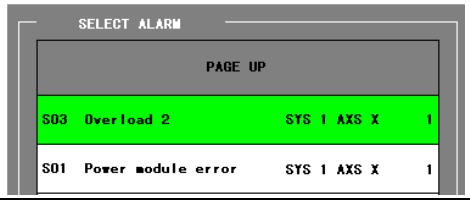
The alarms occurring simultaneously are displayed in the "SELECT ALARM" field. When the corresponding alarm is selected from the alarm list, the guidance corresponding to that alarm will appear.
Up to two alarm histories can be stored. In each history, up to four alarms occurring simultaneously with the alarm can be added if these occur within the time set in the parameter (base specifications parameter #21025).

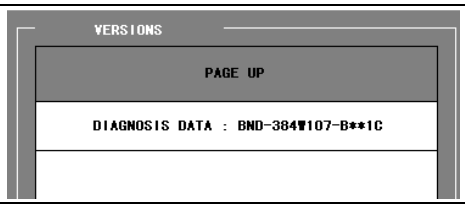
Data for alarms that occur while diagnosing the occurring alarm can also be stored. If an alarm occurs while two alarm histories are stored, the alarm cannot be stored as a history. To store the data for the new alarm, press EXIT and clear the alarm history. The first history is displayed in black, and the second history is displayed in blue.</td></tr>
<tr><td> 

The date/time that the alarm occurred, and the time elapsed are displayed. (These are displayed only for servo alarms.)</td></tr>
<tr><td>IMAGE</td><td> 
When the IMAGE menu is pressed, an image will appear over the alarm selection area.
If there is no image related to the guidance, the IMAGE menu will be invalid.</td></tr>
<tr><td>PROGRAM</td><td> 

Block in which alarm is occurring
Up to three blocks before the block in which the alarm occurred can be displayed.
The line highlighted in green displayed at the center of the three blocks in the figure is the block containing the alarm.
When the ANOTHER SYSTEM button is pressed, the program for another part system when the alarm occurred can be displayed.</td></tr>
<tr><td colspan="2">(Note 1) If the alarm occurred in the head block, nothing will appear in the head line.
(Note 2) If there is no operation taking place with program commands, such as during reference point return, nothing will appear.</td></tr>
</table>

<table>
<tr><th>Menu</th><th>Details</th></tr>
<tr><td>WAVEFORM</td><td>The WAVEFORM screen will open.
Refer to the "3.5.2 NC Alarm Screen (waveform display)".</td></tr>
<tr><td>HISTORY</td><td>When the HISTORY menu is displayed, the guidance displayed up to that point will appear.
If the history contents do not fit in the list, press the function selection menus UP and DOWN to scroll the page.
The first line of the guidance is displayed for the evaluation item.
The YES and NO selected according to the guidance details are highlighted, and number that the display was jumped to is also displayed.
 

<table>
<tr><th>No.</th><th>HISTORY</th><th colspan="4">RESULT</th></tr>
<tr><td>1</td><td>An overload alarm 1 occurred in the power module. Diagnose ··</td><td>YES</td><td>To 2</td><td>NO</td><td></td></tr>
<tr><td>2</td><td>Check if the current FB is almost zero.</td><td>YES</td><td>To 3</td><td>NO</td><td>To 10</td></tr>
<tr><td>3</td><td>The power supply, PN and the motor wire may not be connected··</td><td>YES</td><td>To 4</td><td>NO</td><td>To 5</td></tr>
<tr><td>4</td><td>A poor PN connection can be the cause of the alarm.</td><td>YES</td><td></td><td>NO</td><td></td></tr>
<tr><td>5</td><td>Disconnection of the wire or the motor winding can be the c ··</td><td>YES</td><td>To 7</td><td>NO</td><td>To 6</td></tr>
</table></td></tr>
<tr><td>CAUSE LIST</td><td>When the CAUSE LIST menu is pressed, all alarm factors are displayed.
The number of alarms that occurred, and the date of the last alarm occurred are displayed in the list. If the possible causes do not fit in the list, press the function selection menus UP and DOWN to scroll the page.
(Note) For the number of occurrences, the cause displayed when the EXIT menu is pressed is counted up.
Overload 2
<table>
<tr><th>No.</th><th>POSSIBLE CAUSE</th><th>COUNT</th><th>LAST OCCURRENCE TIME</th></tr>
<tr><td>1</td><td>A poor PN connection can be the cause of the alarm.</td><td>1</td><td>04/22/2003 10:25</td></tr>
<tr><td>2</td><td>The wire disconnection can be the cause of the alarm.</td><td>1</td><td>04/22/2003 10:11</td></tr>
<tr><td>3</td><td>The motor winding disconnection can be the cause of the ala ··</td><td>0</td><td></td></tr>
<tr><td>4</td><td>The flawed servo amplifier can be the cause of the alarm.</td><td>0</td><td></td></tr>
<tr><td>5</td><td>An overcurrent alarm occurred because of a ground fault of ··</td><td>0</td><td></td></tr>
<tr><td>6</td><td>The short-circuited wire can be the cause of the alarm.</td><td>0</td><td></td></tr>
<tr><td>7</td><td>The short-circuited motor winding can be the cause of the a ··</td><td>0</td><td></td></tr>
<tr><td>8</td><td>The flawed servo amplifier can be the cause of the alarm.</td><td>0</td><td></td></tr>
<tr><td>9</td><td>The alarm might have caused by the improper setting of the ··</td><td>0</td><td></td></tr>
<tr><td>10</td><td>A machine overload which is almost as large as a collision ··</td><td>0</td><td></td></tr>
</table></td></tr>
<tr><td>SEARCH</td><td> 
</td><td>Alarms other than those currently occurring can also be searched.
Touch the alarm in the list for which the guidance is to be displayed.</td></tr>
</table>

Menu	Details	
VER.		The guidance version is displayed.
MENU	This changes the function selection menu.	
EXIT	When EXIT is pressed, the currently saved alarm data will be erased. The alarms for which guidance has ended will be erased from the alarm selection list. Using the alarm that occurred first as a reference, the alarms that occurred within the time set in the NC base specifications parameter "SmpDelay (#21025)" are interpreted as alarms occurring with the same cause, and are erased. If the EXIT menu is pressed while the alarm occurrence causes are displayed (last guidance), the alarm will be counted in the number of occurrences displayed in "COUNT" of the cause list. The alarm will not be counted if the menu is pressed during the guidance.	

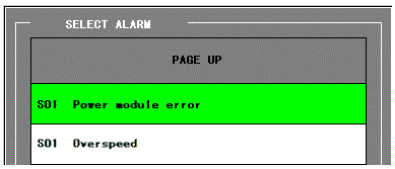
3.6.1.1 Analyzing the Cause of Alarms

An example of analyzing the alarm with this screen is explained in this section using the NC alarm "Power module error" as an example.

1)

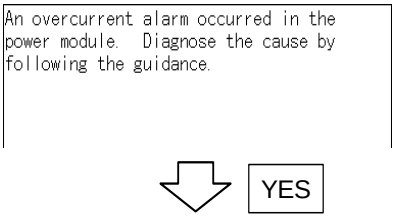
Press and select the corresponding "Power module error" from the list.

The corresponding guidance will appear in the guidance display area on the left.



2)

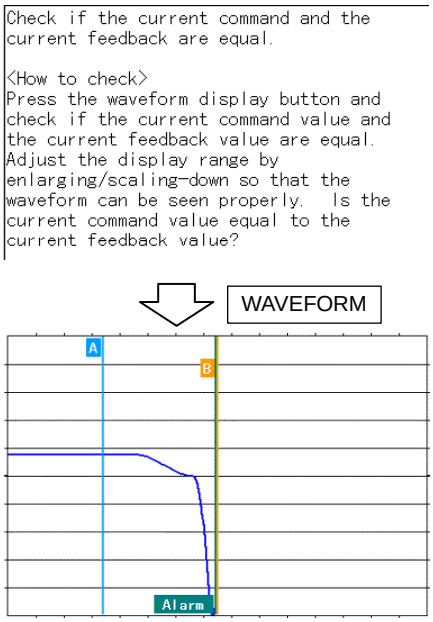
Check the guidance details, and answer the question.
Fits the statement : Press the YES button.
Does not fit : Press the NO button.
If the guidance display does not fit on one page, scroll with the UP and DOWN keys.



3)

If the servo or spindle data confirmation guidance appears in the guidance details, press the WAVEFORM menu.

The WAVEFORM screen will open.
The corresponding waveform data will appear according to the guidance details.
Vertical axis : The vertical axis is automatically adjusted according to the display area.
Time axis : Press the TIME/DIV menu to adjust the time axis.



4)

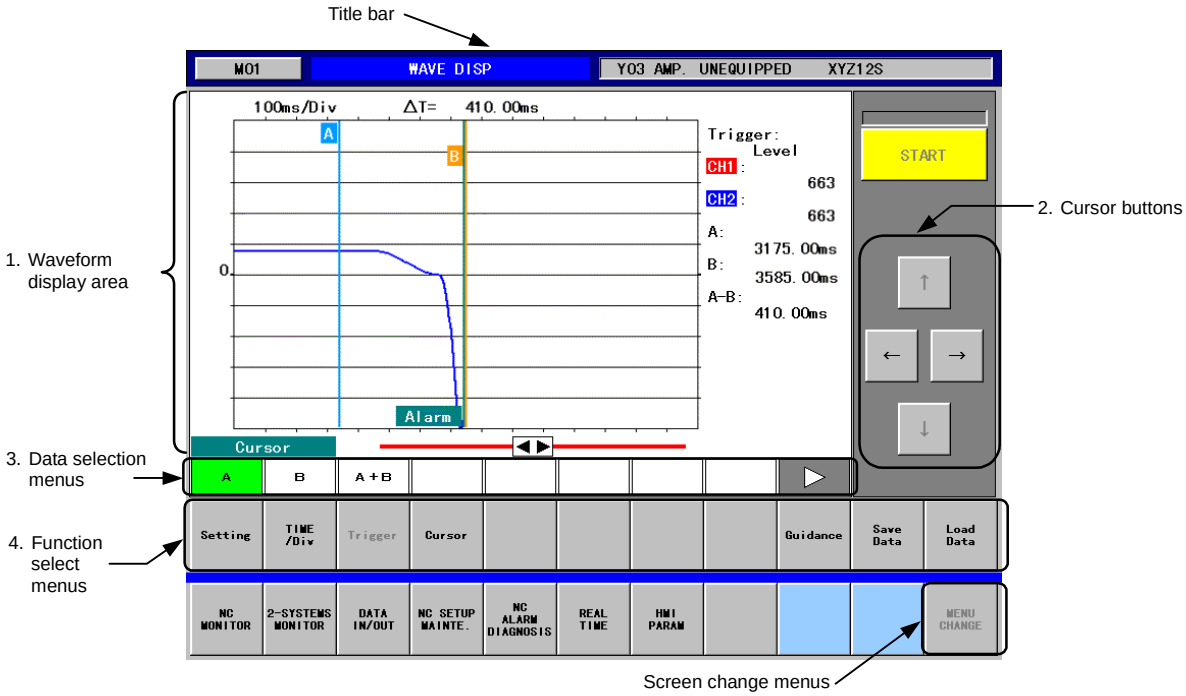
If the cause matches, press the EXIT menu, and end the alarm diagnosis.

When YES is pressed, the diagnosed alarm will be deleted from the alarm selection list together with other alarms that occurred simultaneously.

If the guidance does not match the alarm occurrence cause, press the RETURN button to retrace the steps just taken, and recheck the guidance details.

3.6.2 NC Alarm Screen (waveform display)

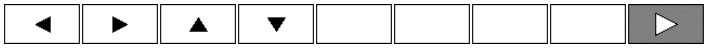
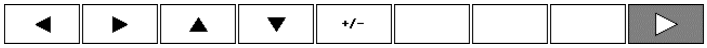
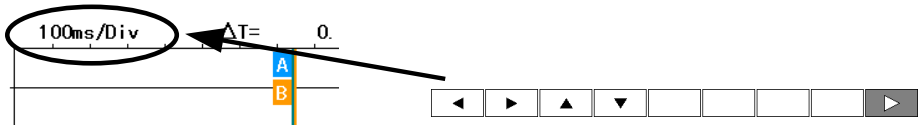
The waveform display screen will open when the WAVEFORM menu is pressed on the NC Alarm Guidance screen.
The servo and spindle data current when the servo and spindle alarm occurred are displayed on this screen.

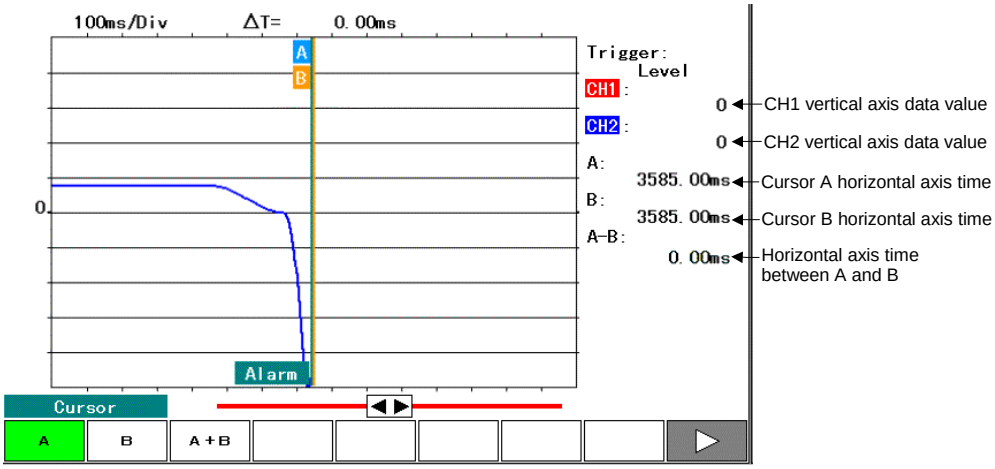


(1) Display items

Display items	Details
1. Waveform display area	The servo and spindle data current when the servo and spindle alarm occurred is displayed. This display area is also used when setting the various display conditions.
2. Cursor buttons	The cursor buttons are used to scroll the waveform display, etc. (The START/STOP buttons are not used when the data current at the alarm occurrence is displayed.) ↑ ↓ : Use these to select the data item to be set. ← → : Use these to move the cursor left and right.
3. Data selection menus	These menus differ according to the set item, and appear when display data or a movement target cursor is selected. These are also used to set and change the value data.
4. Function selection menus	When a function is selected and the menu is pressed, the display will be highlighted and the function setting will be enabled. Refer to the next page for explanations on each menu.

(2) Menus

Menu	Details
Setting	The following display data setting items will appear. Select the setting item by touching the line to be selected or by using the UP/DOWN cursor buttons. Select the display data from the data select button. Axis: Displays the axis to be displayed in waveform. If there are multiple axes, the displayed axis can be changed. Data: Select the display data from the following data select buttons.  Data/Div: Set the vertical axis scale. (0 to 99999999) The setting can be changed with ▲▼. The setting digit can be selected with ◀▶.  Offset: Set the vertical axis' basic position. (-999999999 to 999999999) The setting can be changed with ▲▼. The setting digit can be selected with ◀▶. 
TIME /Div	Set the horizontal axis (display range) scale. When this menu is pressed, the display range in the waveform display area will be highlighted. The setting can be changed with ▲▼. The setting digit can be selected with ◀▶. The scale can be set in the range of 0 to 9999. 

Menu	Details
Cursor	Two cursor signs will appear. Measure the time difference and data with these two cursor signs (A and B). The cursor signs can be moved with ← and → buttons.  **(Note)** CH1 and CH2 show the intersection coordinate value of the waveform when A is selected and the A cursor, this also applies when B is selected. The CH1 and CH2 vertical axes are not displayed when "A+B" is selected.
Guidance	The NC ALARM GUIDANCE screen will open. Refer to the "3.6.1 NC Alarm Guidance Screen".
Save Data	The current sampling data and setting values are saved in the memory card.
Load Data	The saved data can be read out. The File Select screen will open when this menu is pressed. Select the data to be displayed, and then press the FIX button.

(3) Precautions

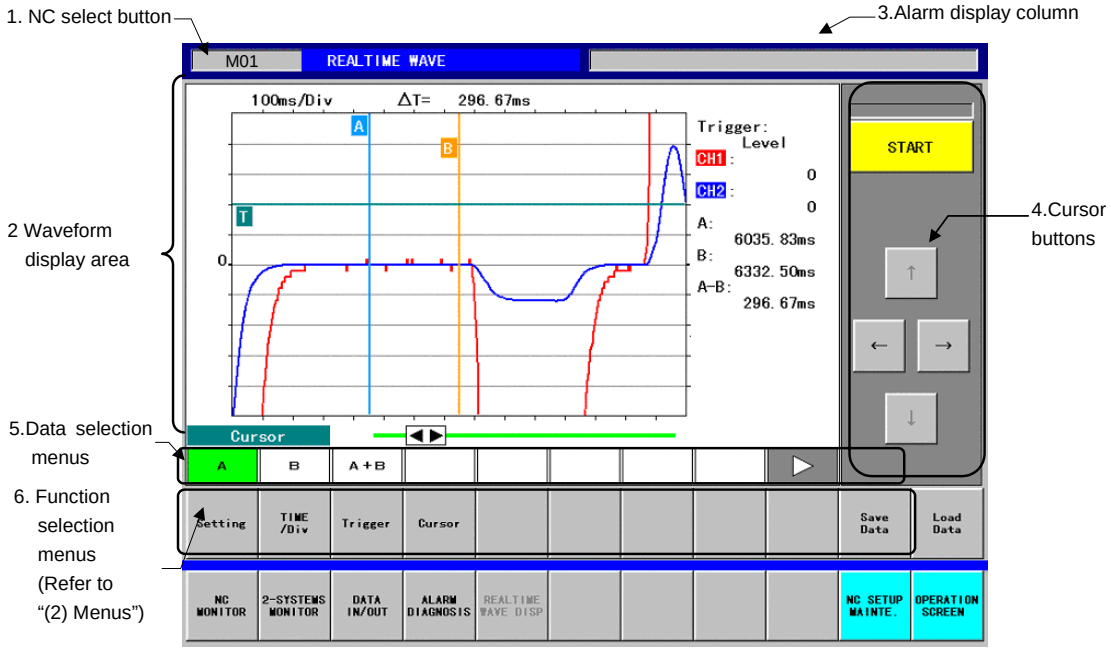
- There are two sampling buffers for saving the data current when the alarm occurs. Thus, the data can be saved even if an alarm occurs while diagnosing an occurring alarm.
  The data cannot be saved if the alarm occurs when two storage areas are already used.
  Press the EXIT menu in the NC ALARM GUIDANCE screen, and release the buffer area to enable use.
- After an alarm occurs, the sample data is automatically erased when the time set in the base specifications parameter (#21030) has elapsed.
  The sampling data will also be erased if the EXIT menu in the NC ALARM GUIDANCE screen is pressed.

3.7 Realtime Wave Screen (For only the panel computer)

Servo data/Spindle data waveform display tool of C64T has two modes;

- The mode to display the waveform at the alarm occurrence. (Available for NC alarm guidance.)
- The mode to display the real time waveform of Servo data and Spindle data.

Refer to “3.6.2 NC Alarm Screen (waveform display)” for the waveform at the alarm occurrence.
This section (3.7) describes the waveform of the real time mode.
During the real time mode, the screen displays the waveform of the synchronous error amount, that is, the position error amount between the spindle and the tapping axis during synchronous tapping.

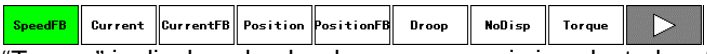
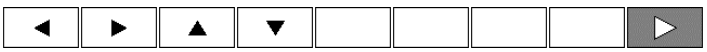


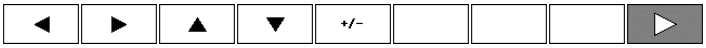
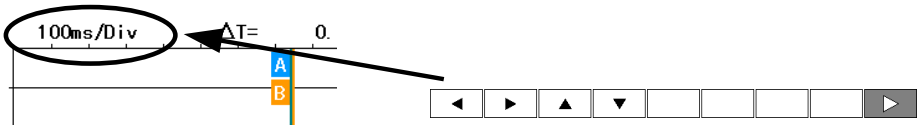
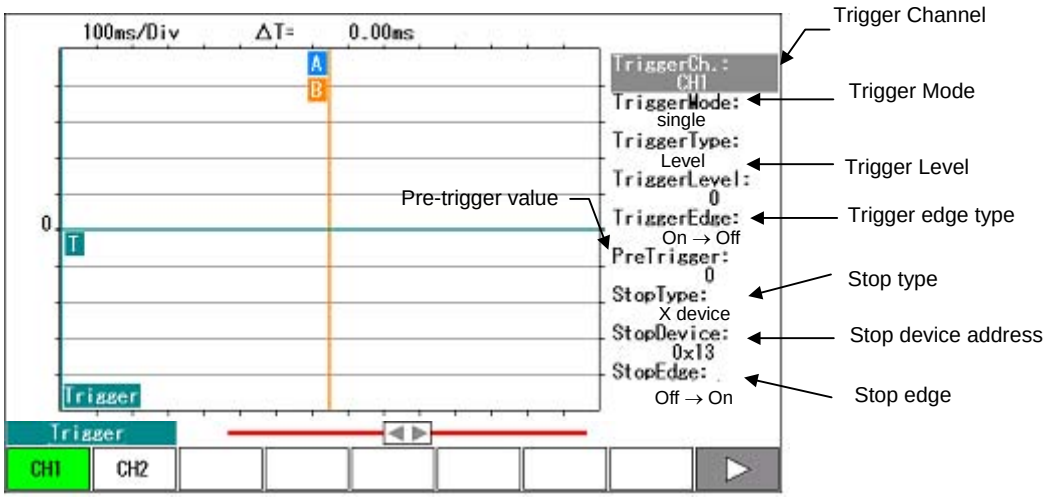
(1) Display items

Display items	Details
1. NC select button	The name of the selected NC is displayed. Press here to display a pull-down menu to select an NC. The operation method is the same as the NC MONITOR screen.
2. Waveform display area	The servo and spindle data current when the servo and spindle alarm occurred is displayed. This display area is also used when setting the various display conditions.
3. Alarm display column	Displays a occurring alarm.
4. Cursor buttons	The cursor buttons are used to scroll the waveform display, etc. (The START/STOP buttons are not used when the data current at the alarm occurrence is displayed.) ↑ ↓ : Use these to select the data item to be set. ← → : Use these to move the cursor left and right. START STOP : Switches the sampling start or stop.
5. Data selection menus	These menus differ according to the set item, and appear when display data or a movement target cursor is selected. These are also used to set and change the value data.
6. Function selection menus	When a function is selected and the menu is pressed, the display will be highlighted and the function setting will be enabled. Refer to the next page for explanations on each menu.

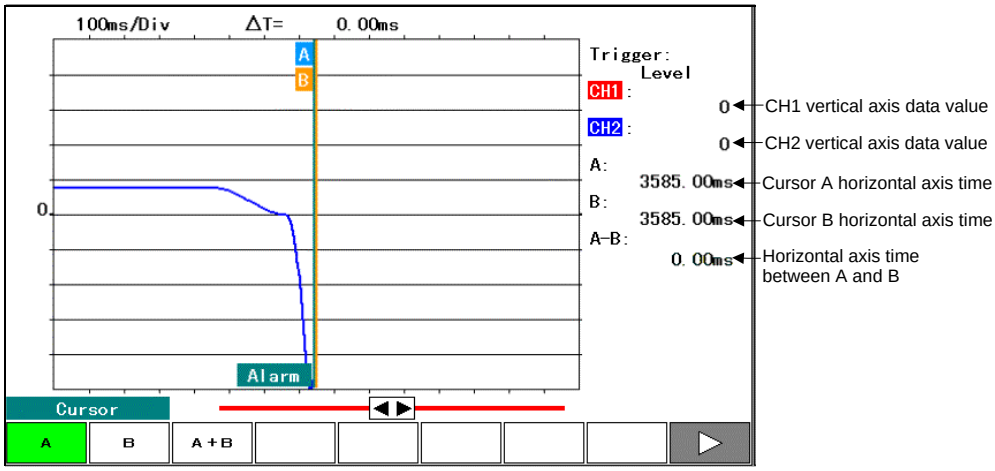
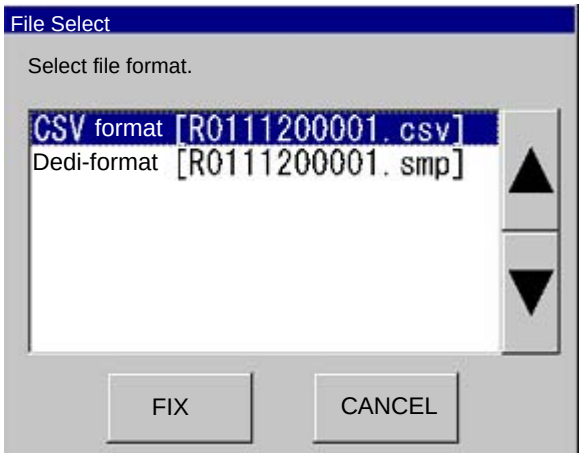
(2) Menus

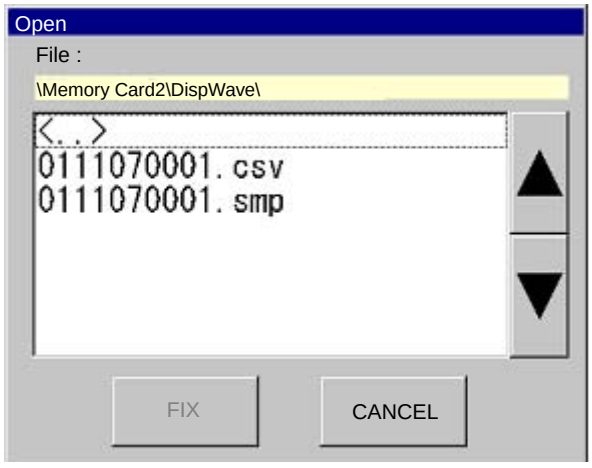
The setting data of “Setting”, “TIME/Div”, and “Trigger” are retained during real time mode.

Menu	Details
Setting	The following display data setting items will appear. Select the setting item by touching the line to be selected or by using the UP/DOWN cursor buttons. Select the display data from the data select button. Axis: Displays the axis to be displayed in waveform. If there are multiple axes, the displayed axis can be changed. Data: Select the display data from the following data select buttons. Servo axes:  “Torque” is displayed only when a servo axis is selected on “Axis” during the real time mode. To display the “Torque” correctly, set proper values to the servo parameters. Spindle axes:  “Sync-tap err width” and “Sync-tap err angle” are displayed only when a spindle is selected on “Axis” during the real time mode. The units of displayed data are as follows. SpeedFB: r/min Current: % CurrentFB: % Position: interpolation unit PositionFB: interpolation unit Droop: interpolation unit Torque: interpolation unit Sync-tap err width: μm Sync-tap err angle: 0.001 ° Data/Div: Set the vertical axis scale. (1 to 99999999) The setting can be changed with ▲▼. The setting digit can be selected with ◀▶. 

<table>
<tr><th>Menu</th><th>Details</th></tr>
<tr><td>Setting</td><td>Offset
Set the vertical axis' basic position. (-999999999 to 999999999)
The setting can be changed with ▲▼. The setting digit can be selected with ◀▶.
 
P-P
0 to 999999999
Displays the difference between the maximum value and the minimum value.</td></tr>
<tr><td>TIME /Div</td><td>Set the horizontal axis (display range) scale. When this menu is pressed, the display range in the waveform display area will be highlighted.
The setting can be changed with ▲▼. The setting digit can be selected with ◀▶. The scale can be set in the range of 0 to 9999.
  </td></tr>
<tr><td>Trigger</td><td>Pressing Trigger button turns the waveform display to the next screen, that is, Trigger condition setting screen.
On this screen, select the setting item by touching the item to be set or by using the UP/DOWN cursor buttons.
Trigger button cannot be used on the screen transit from NC Alarm Guidance Tool.
  </td></tr>
</table>

Menu	Details	
Trigger	TriggerCh. (CH1/CH2)	Designate the channel to set a trigger.
	Trigger Mode (single/repeat/free-run)	Designate the trigger mode. "Single" means one-shot trigger. "Repeat" starts next sampling when the trigger condition is satisfied again after the previous data sampling finishes. Select one from the followings. Single | Repeat | FreeRun | ▷
	Trigger Type (Level/X device/ Y device/Alarm)	Select the trigger type. Selecting "Level" sets a trigger when the data exceed the given value. Selecting "X device" or "Y device" sets a trigger when X or Y device turns. The condition to turn to 0 or 1 depends on the edge setting. When "Alarm" is selected, sampling stops when the servo alarm or the spindle alarm occurs. Select from menu. Level | Xdevice | Ydevice | Alarm | ▷
	Trigger Level (Level: 0 to 99999 X,Y device: 0 to FF Alarm: None)	When "Level" is selected on "Trigger Type" above, set the trigger level. At this time, current setting value is displayed on the waveform display area. When "X device" or "Y device" is selected, set the device address. Change the setting by ▼ and ▲ buttons. To select a digit to set, use ◄ and ► buttons.
	Trigger Edge (Off→On/On→Off/Both edge)	Select the edge to set a trigger on. Select from menu and set. Off→On | On→Off | Both edge | ▷
	Pre Trigger (0 to 9999ms)	Set the pre-trigger value. Change the setting by ▼ and ▲ buttons. To select a digit to set, use ◄ and ► buttons.
	Stop Type (X device/Y device /Alarm/None)	Select the stop type. When X or Y device is selected, sampling stops when X or Y device turns. The condition to turn to 0 or 1 depends on the edge setting. When "Alarm" is selected, sampling stops when the servo alarm or the spindle alarm occurs. Select from menu. Xdevice | Ydevice | Alarm | None | ▷
	Stop Device (X,Y device: 0 to FF Alarm: None)	Set the address of X or Y device when "X device" or "Y device" is selected on "Stop Type". Change the setting by ▼ and ▲ buttons. To select a digit to set, use ◄ and ► buttons.
	Stop Edge (Off→On/On→Off/ Both edge)	Set the turning edge of X or Y device when "X device" or "Y device" is selected on "Stop Type". Select from menu and set. Off→On | On→Off | Both edge | ▷

Menu	Details
Cursor	Two cursor signs will appear. Measure the time difference and data with these two cursor signs (A and B). The cursor signs can be moved with ← and → buttons.  **(Note)** CH1 and CH2 show the intersection coordinate value of the waveform when A is selected and the A cursor, this also applies when B is selected. The CH1 and CH2 vertical axes are not displayed when "A+B" is selected.
Save Data	The current sampling data and setting values are saved in the memory card. Select binary format or CSV format to save a file. The Format Select screen opens when this menu is pressed. Select proper format and press FIX button. Pressing CANCEL button abandons data saving.  Files will be stored into the memory card2 with the name "*date* + ID.smp" for binary format or "*date* + ID.csv" for CSV format. The character strings quoted by [ ] are the saved file name.

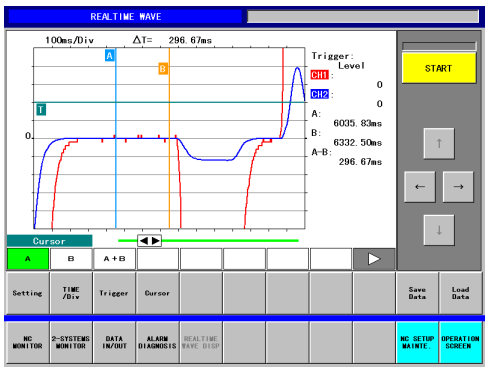
Menu	Details
Save Data	Output sample of CSV format. RealTime Setting:,CH1,CH2 Axis,Y1,X1 Data,SpeedFB, SpeedFB DataID,0,0 Data/Div,0,0 Offset,0,0 Time/Div,100 TriggerCH,CH1 TriggerMode,2,FreeRun TriggerType,4,Alarm TriggerLevel,0 TriggerEdge,0,None PreTrigger,0 StopType,2,Y Device StopDevice,23 StopEdge,2,Pulse Off -> On CursorPos,275,275 DisplayPos,1520 Trigger[ms]:,0 Data: Time[ms],SpeedFB(Y1), SpeedFB (X1) 0,0,0 1,0,0 2,0,0 3,0,0
Load Data	The data saved in the memory card2 can be read out. The File Select screen will open when this menu is pressed. Select the data to be displayed, and then press the FIX button to display the saved data. As for the loading format, whether the dedicated format or CSV format is recognized by the file extension. 

3.7.1 Operation Example of Waveform Display Tool

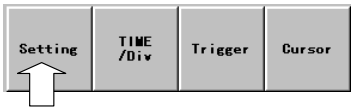
Operation examples of Servo data and Spindle data Waveform Display tool are described below.

3.7.1.1 Real Time Waveform Display

1) Press REALTIME WAVE DISP menu to use Servo data and Spindle data Waveform Display tool.



2) Press Setting menu to display the data setting screen.

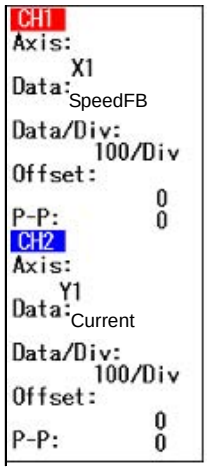


To display the waveform of synchronous tapping error:

CH1: Set as follows
- Axis: “X1”(servo axis)
- Data: “SpeedFB”
- Data/Div: “100”
- Offset: “0”

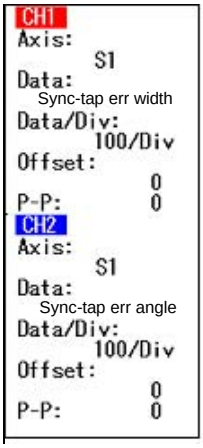
CH2: Set as follows
- Axis: “Y1”(servo axis)
- Data: “Current”
- Data/Div: “100”
- Offset: “0”

To display the waveform of speedFB and current command:

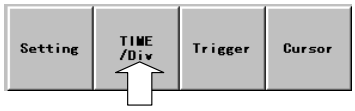


CH1: Set as follows
- Axis: “S1”(spindle)
- Data: “Sync-tap err width”
- Data/Div: “100”
- Offset: “0”

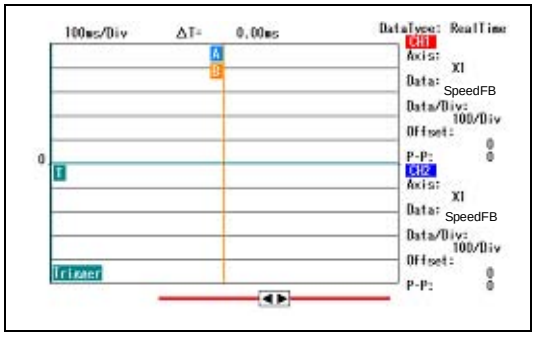
CH2: Set as follows
- Axis: “S1”(spindle)
- Data: “Sync-tap err angle”
- Data/Div: “100”
- Offset: “0”



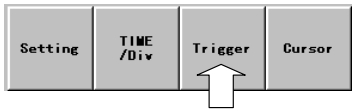
3) Press TIME/Div menu to display the setting screen.



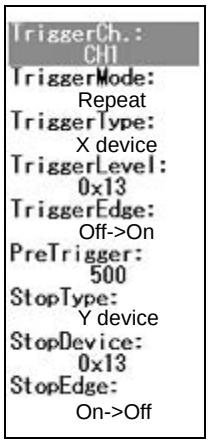
4) Set “100” as TIME/DIV value.



5) Press Trigger menu to display the setting screen.

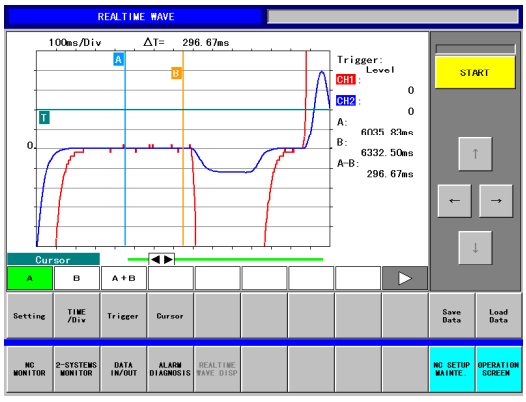


Set as follows;
- TriggerCh: “CH1”
- TriggerMode: “Repeat”
- TriggerType: “X device”
- TriggerLevel: “0x13”
- TriggerEdge: “Off->On”
- PreTrigger: “500”
- StopType: “X device”
- StopLevel: “0x13”
- StopEdge: “On->Off”



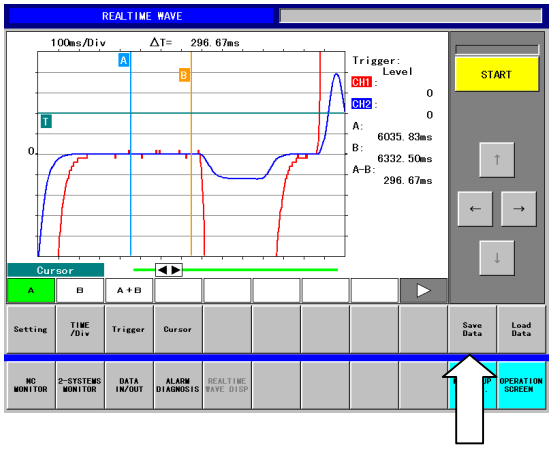
6) Press START button to start sampling.

7) Waveform display starts at the time when X device 0x13 is turned ON.
Waveform display stops when:
- X device 0x13 is turned OFF.
- START button is pressed again.
- 30 seconds have passed since waveform display started.

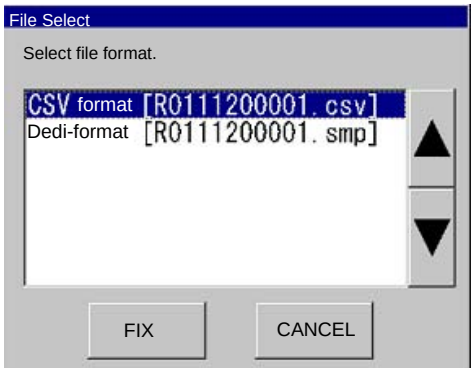


3.7.1.2 To Save the Waveforme Data

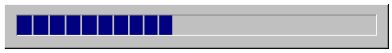
1) Press Save Data on the screen.



2) File Select dialog will open.
Select “CSV format” and press FIX button.

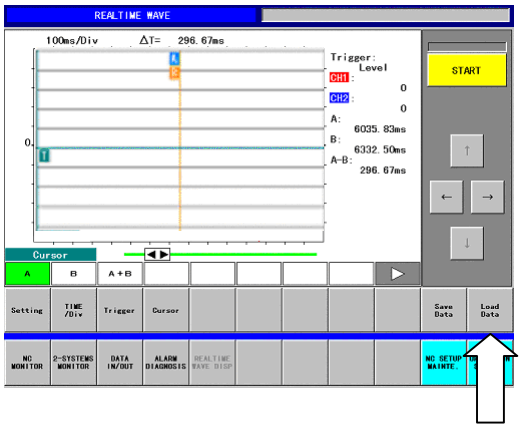


3) Progress bar indicates the file saving status. When the progress bar is disappeared, saving a file completes.

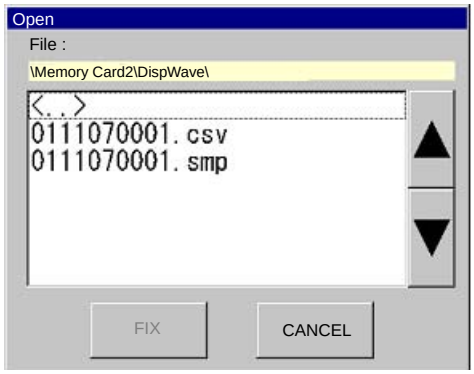


3.7.1.3 To Load the Waveform Data

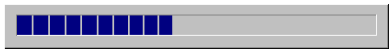
1) Press Load Data menu.



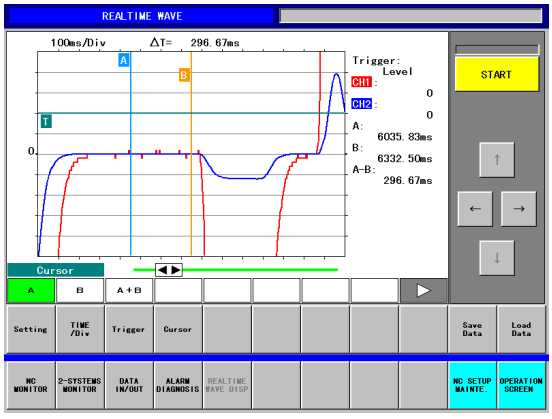
2) File Select dialog will open.
Select the file including the waveform data to be loaded and press FIX button.



3) Progress bar indicates the file loading status. When the progress bar is disappeared, loading a file completes.

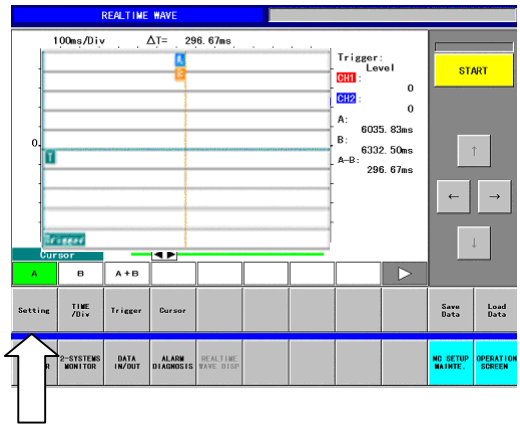


4) When loading completes, the loaded waveform data will be displayed on the screen.

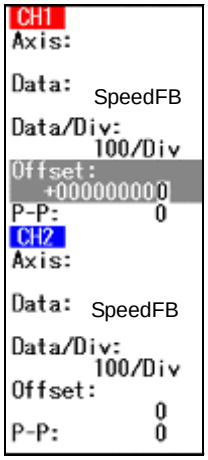


3.7.1.4 To Change a Numerical Parameter

1) Press Setting menu to display the setting screen.



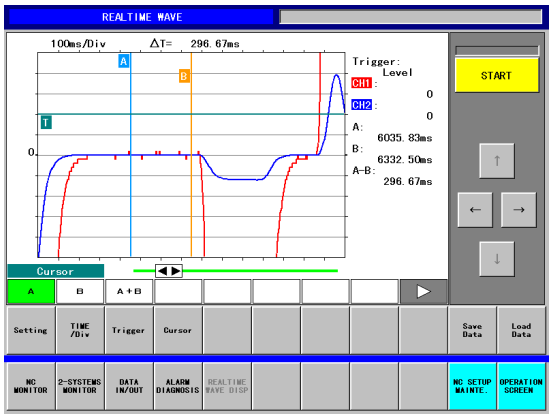
2) Click “Offset” of CH1.
Then the clicked item will be reversed.



3) Press ◀ button twice.
The 3rd figure of CH1 Offset data will be reversed.

4) Press ▲ button five times.
The value of the 3rd figure of CH1 Offset will increase by 5.

5) Press Setting menu again.
Parameter setting completes and normal screen will be recovered.



3.7.2 Synchronous Tapping Error Display

The width and the angle of synchronous tapping error is obtained by the formulas shown below.

[Synchronous tapping error width]

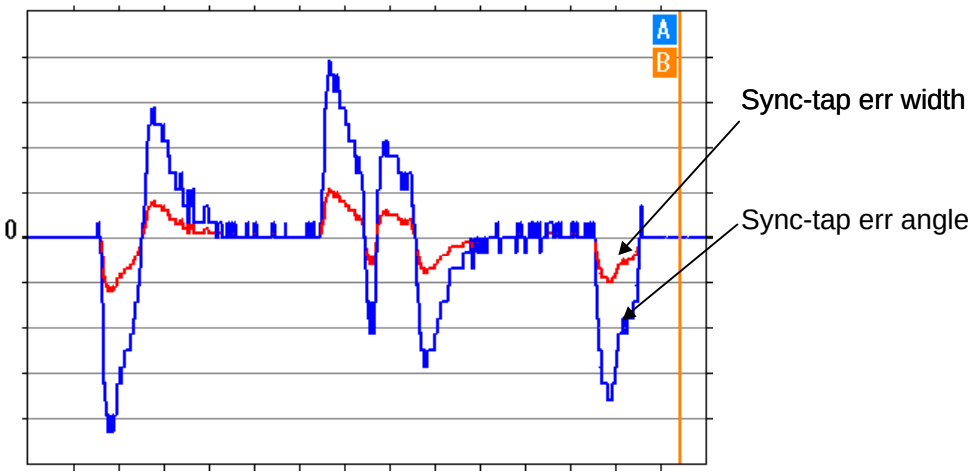
$$\text{Sync-tap err width } (\mu m) = \text{Tapping axis position deviation amount (i)} - \frac{\text{Spindle position deviation amount} - \text{screw lead(mm)} \times 1000}{4096 \times 5625}$$

[Synchronous tapping error angle]

$$\text{Sync-tap err angle } (0.001°) = \frac{\text{Sync-tap err wide } (\mu m) \times 360}{\text{Screw lead(mm)} \times 1000} \times 1000$$

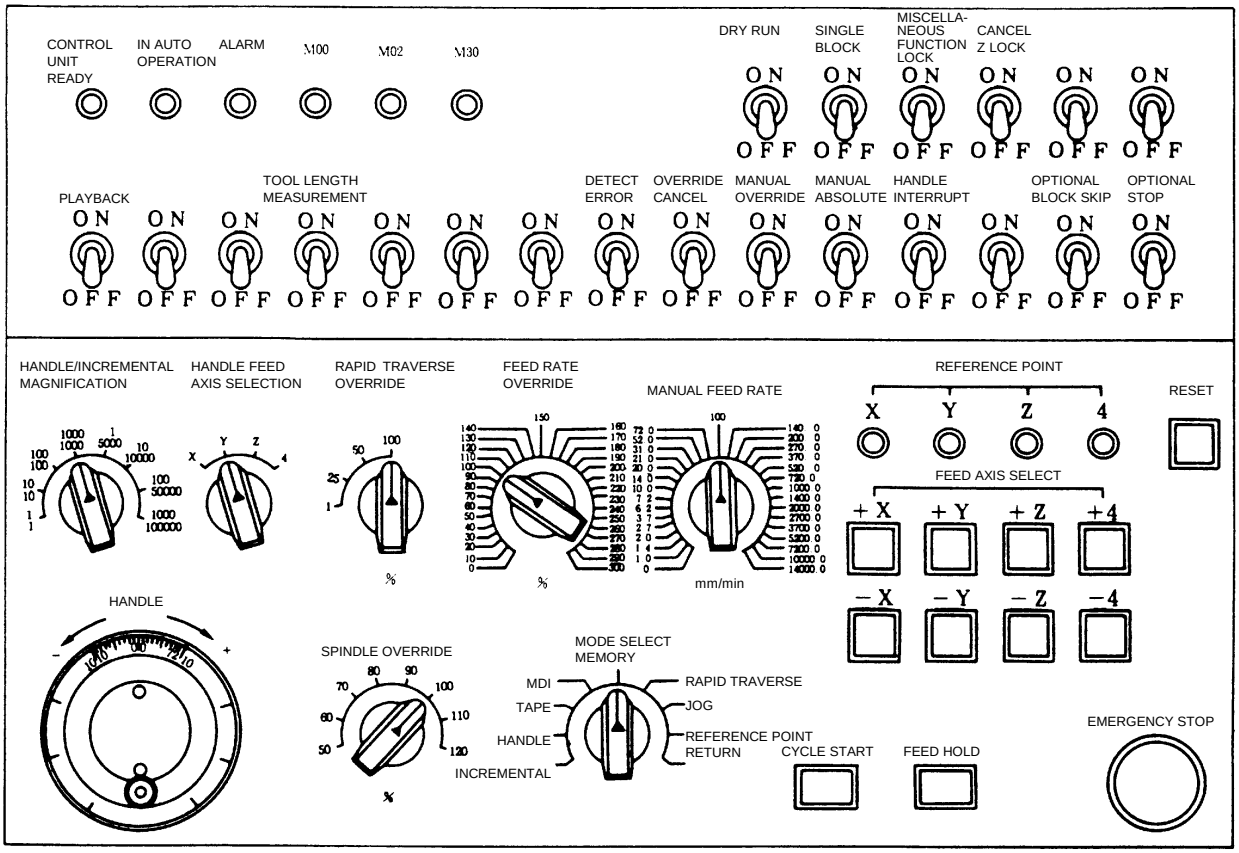
(Note) “i” is interpolation unit.

Waveform of the synchronous tapping will be displayed as shown below.



CHAPTER 2 MACHINE OPERATIONS

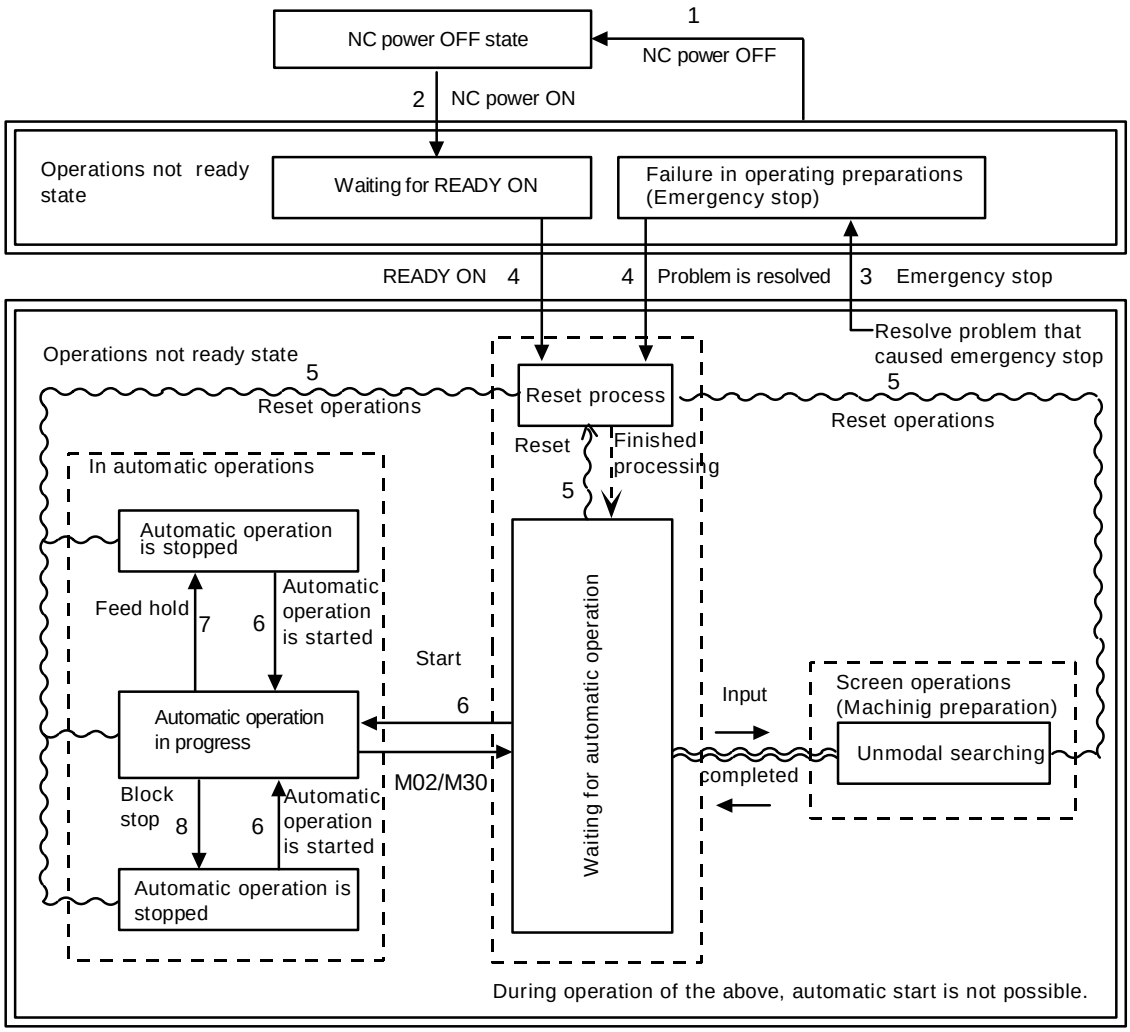
This chapter explains the functions and operation method of the machine operation switches for operation (automatic operation and manual operation) by using the illustration of the machine operation panel.
The actual machine operation and motion vary from one minute to another. Refer to the operation manual issued by the machine manufacturer. 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 moveable range of the machine during automatic operation. During rotation, keep hands, feet and face away from the spindle.

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.

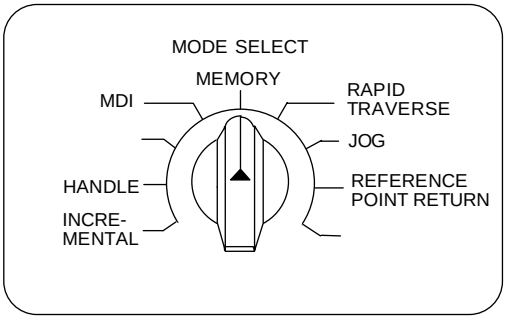
CAUTION

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.

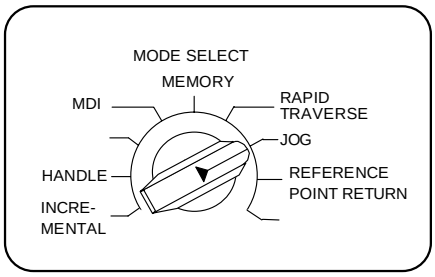
(Note 1) See 1.4 for the running state when a change is made to another mode during automatic operation.

4.2 Jog Feed Mode

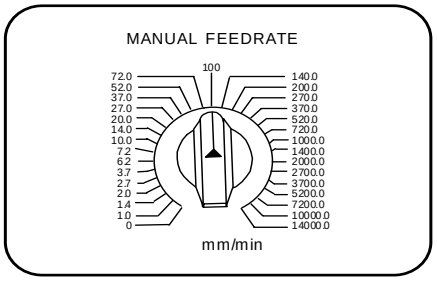
The jog feed mode enables the machine to be moved consecutively at the feedrate set by using the MANUAL FEED RATE switch manually. The jog feed mode is started by using the FEED AXIS SELECT switch.
See Section 5 for the MANUAL FEED RATE switch.

Operation procedure

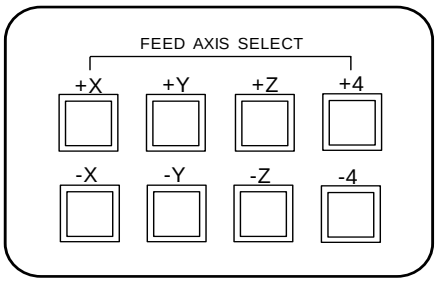
Using the MODE SELECT switch, select the jog mode.



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.

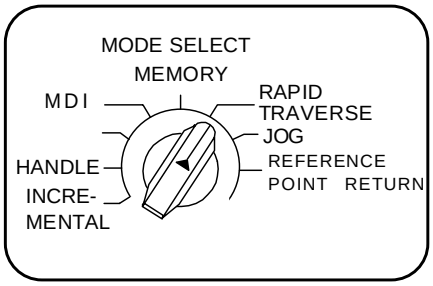
4.3 Rapid Traverse Feed Mode

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.

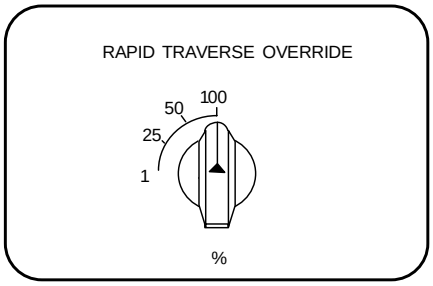
(Note 1) Refer to the manual issued by the machine manufacturer for the rapid traverse feedrate.
(Note 2) See Section 5 for the RAPID TRAVERSE OVERRIDE switch.

Operation procedure

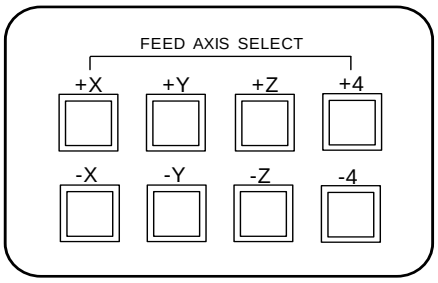
Using the MODE SELECT switch, select the rapid traverse feed mode.



Using the RAPID TRAVERSE OVERRIDE switch, set any desired override value.



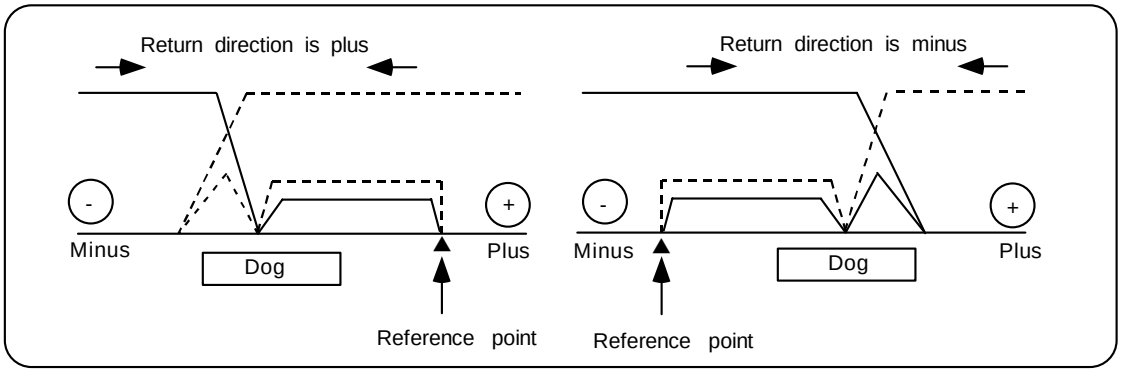
To move the controlled axis, turn on the FEED AXIS SELECT switch. The controlled axis is moved while the switch is turned on.
When the switch is turned off, the controlled axis stops with deceleration.



(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.



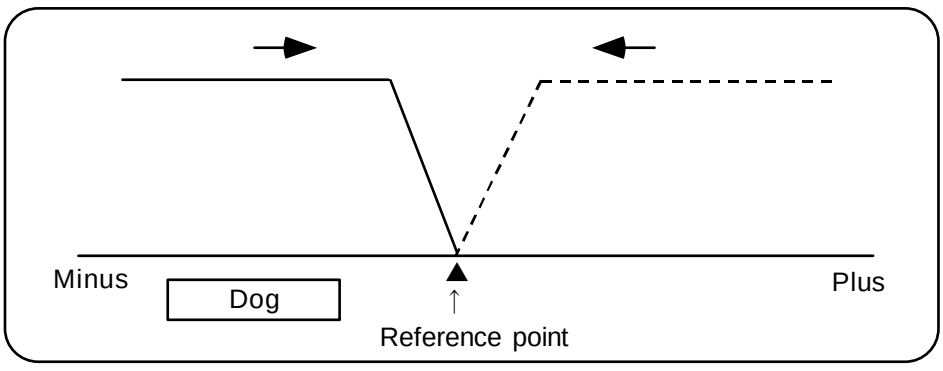
Dog mode return to reference position

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.

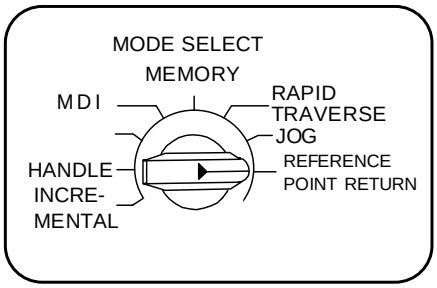
High speed return to reference position

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.

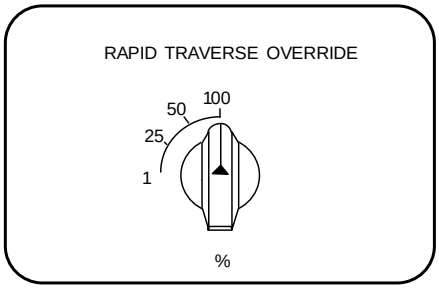


Operation procedure

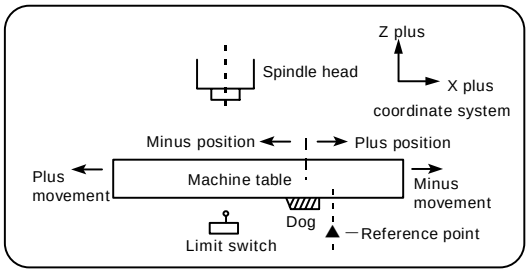
Using the MODE SELECT switch, select the return to reference position.



Using the RAPID TRAVERSE OVERRIDE switch, set any desired override value.

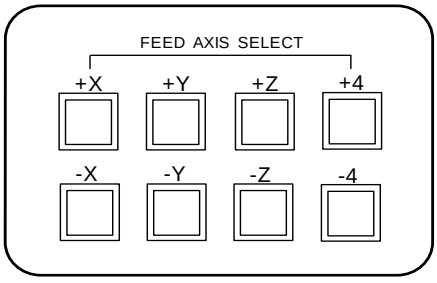


Check the current machine position.



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.

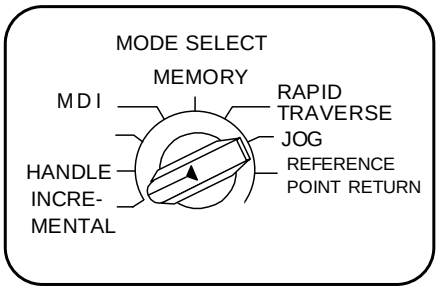
Hold the FEED AXIS SELECT switch on during return to reference position until the machine passes by the dog (dog mode) or the REFERENCE POSITION ARRIVAL lamp goes on (high speed return).

4.5 Incremental Feed Mode

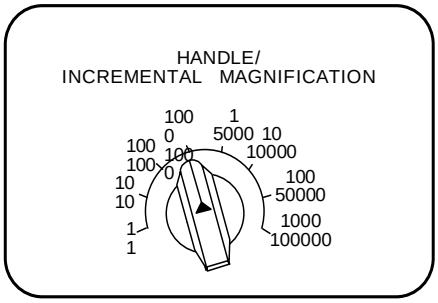
The incremental feed mode enables the controlled axis to be moved at a given distance selected by using the HANDLE/INCREMENTAL MAGNIFICATION switch at the manual feedrate when the FEED AXIS SELECT switch is on.

Operation procedure

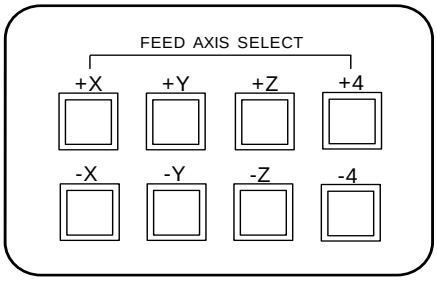
Using the MODE SELECT switch, select the incremental feed mode.



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.

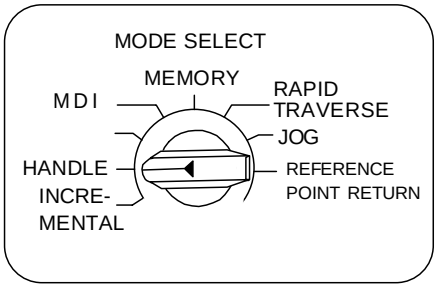


4.6 Handle Feed Mode

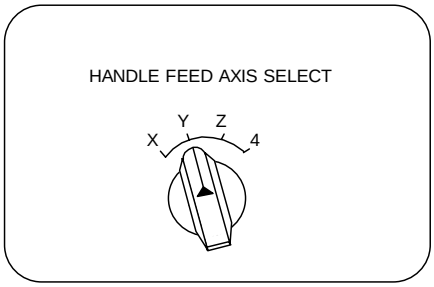
The controlled axis can be moved by turning the manual handle.
The travel distance per graduation of the handle depends on how the HANDLE/INCREMENTAL MAGNIFICATION switch is set.
The axis that can be moved by using the manual handle is determined by setting the HANDLE FEED AXIS SELECT switch.

Operation procedure

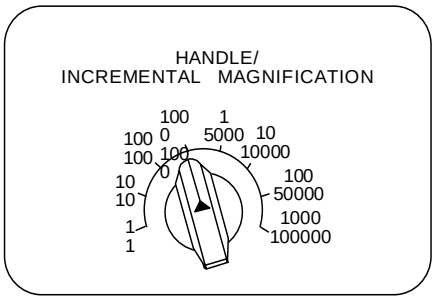
Using the MODE SELECT switch, select the handle feed mode.



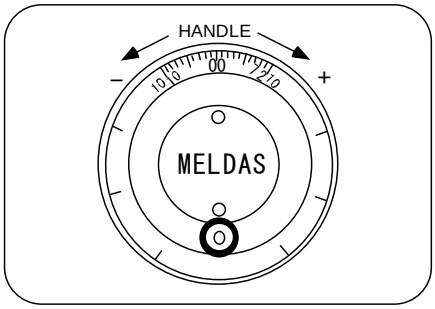
Using the HANDLE FEED AXIS SELECT switch, select the controlled axis to be moved.



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.

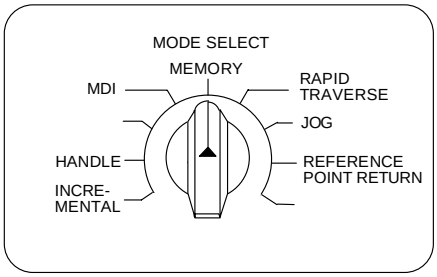
Operation procedure

Call the work program for memory operation by using the setting and display unit.

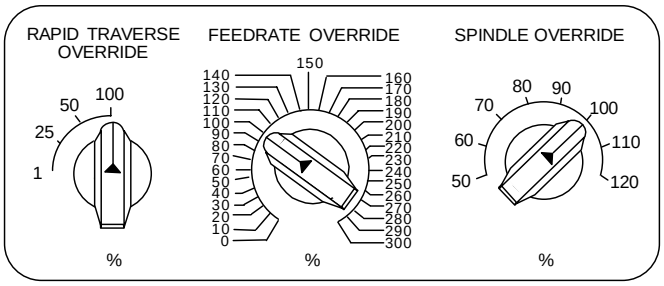
(Refer to the “Operation Search” section in the Operation Manual.)

Check whether or not the work program is called normally.

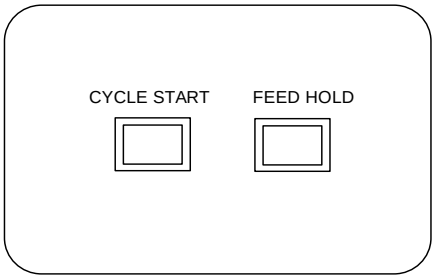
Using the MODE SELECT switch, select the memory mode.



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

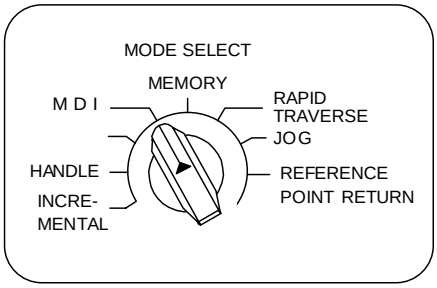
In the MDI operation mode, automatic operation is executed by using a program set on the CRT setting and display unit MDI screen.

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.



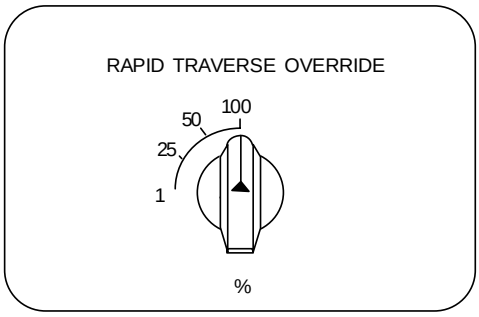
The following steps are the same as the memory operation steps.

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:

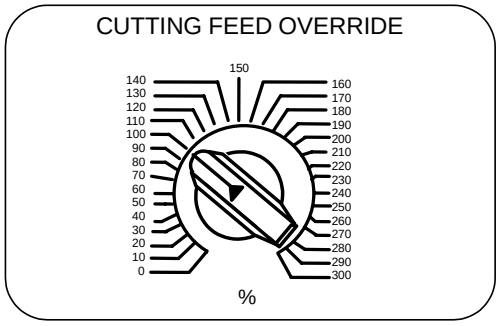
Automatic operation: G00, G27, G28, G29, G30
Manual operation: Rapid traverse, return to reference position, incremental feed



(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.

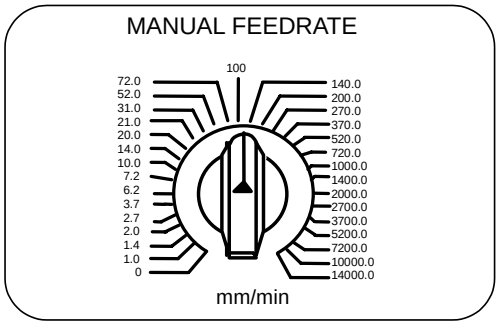


(Note 1) The dry run rate is the movement rate set by using the MANUAL FEEDRATE switch by overriding the programmed feedrate in automatic operation.
(Note 2) See Section 6.6 for feedrate override applied to manual feedrate.

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.

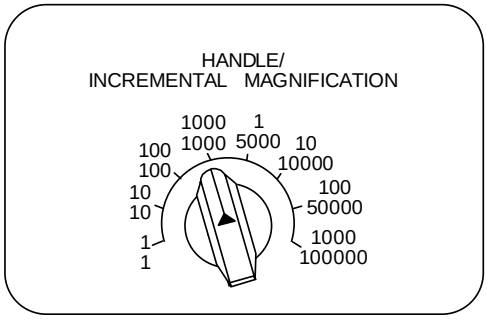
Manual feedrate (mm/min)				
0.	7.2	72	720	7200
1.0	10.0	100	1000	10000
1.4	14.0	140	1400	14000
2.0	20.0	200	2000	
2.7	27.0	270	2700	
3.7	37.0	370	3700	
5.2	52.0	520	5200	



5.4 Handle/Incremental Feed Magnification Factor

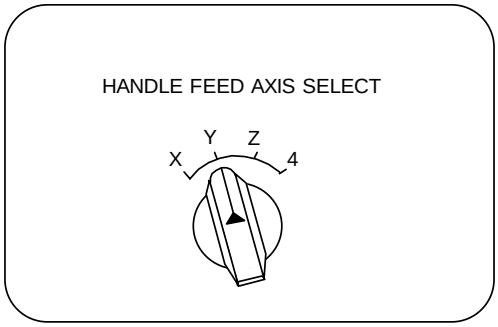
Use the HANDLE/INCREMENTAL MAGNIFICATION switch to set the travel distance specified when manual handle feed or incremental feed is made.
The travel distances for each axis are listed below:
(Up to 1000 can be set for handle feed.)

Handle	Incremental
1	1
10	10
100	100
1000	1000
1	5000
10	10000
100	50000
1000	100000



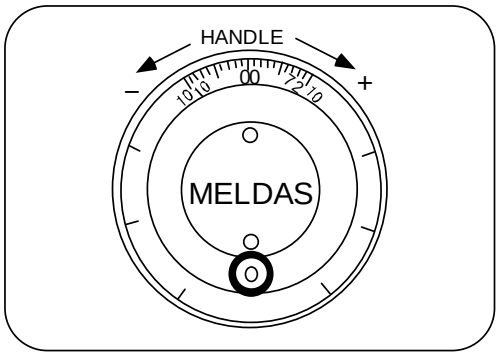
5.5 Handle Feed Axis Selection

Use the HANDLE FEED AXIS SELECT switch to select the axis moved by handle operation when the handle mode is selected.



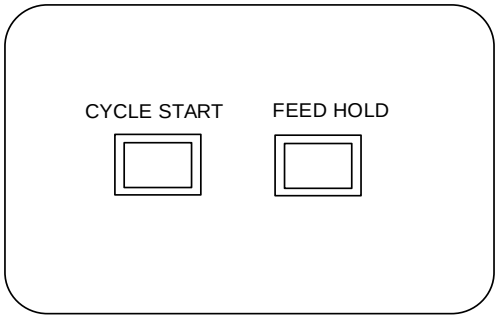
5.6 Manual Pulse Generator

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

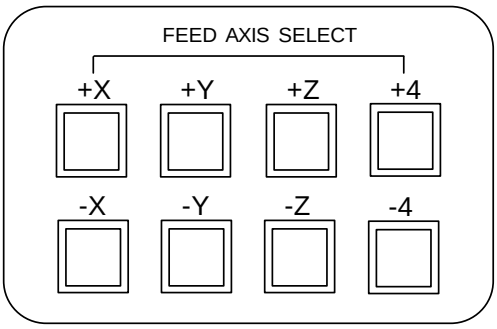
Use the CYCLE START switch to start automatic operation (memory, tape, or MDI). Automatic operation is executed by turning on the switch. Use also the switch for restart from stop by the FEED HOLD switch or the automatic operation stop state.



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) The automatic operation speed during machine lock can be selected as the commanded speed or machine lock speed using the parameters.
- Commanded speed – The movement is executed at the feedrate commanded in the program. Thus, the time is the same as actual machining.
- Machine lock speed – The movement commands are processed at the rapid traverse rate, and the dwell time, etc., is ignored. Thus, the program check, etc., can be carried out faster.

(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.

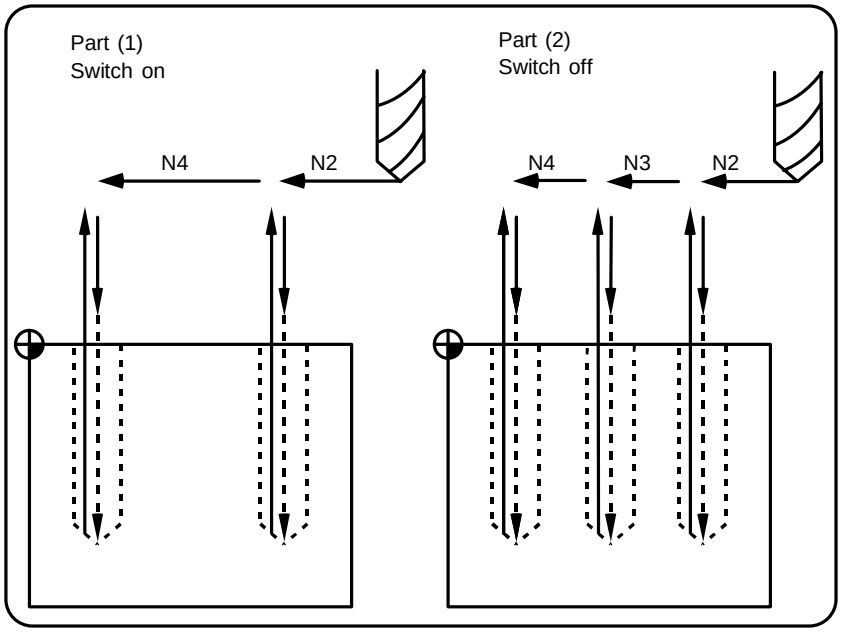
(2) The machine stops after the M01 block is executed.

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.

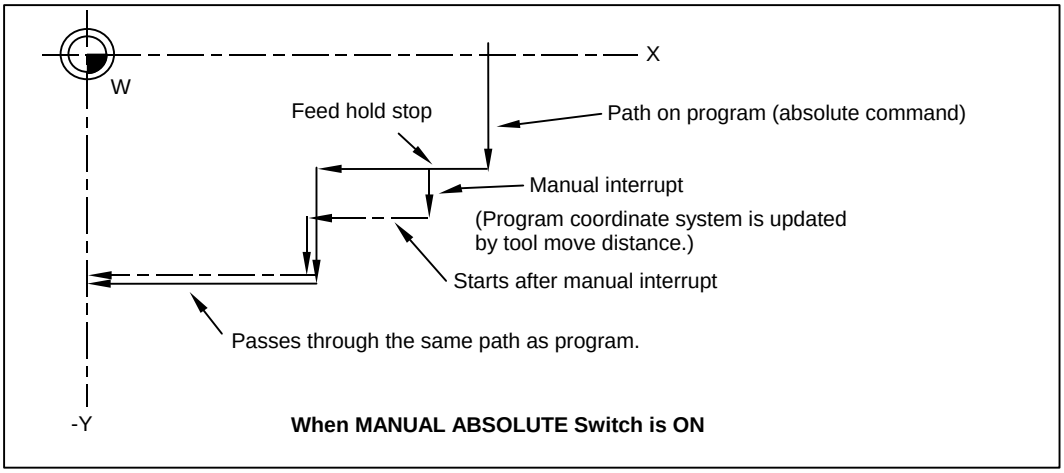
(Example) To work two parts as illustrated below, if the following program is prepared and work is made by turning on the OPTIONAL BLOCK SKIP switch, part (1) is provided; if work is made by turning off the switch, part (2) is provided:

```
Program  N1G54 ;
         N2G90G81X50. Z-20. R3. F100 ;
         /N3X30. ;
         N4X10. ;
         N5G80 ;
         M02 ;
```

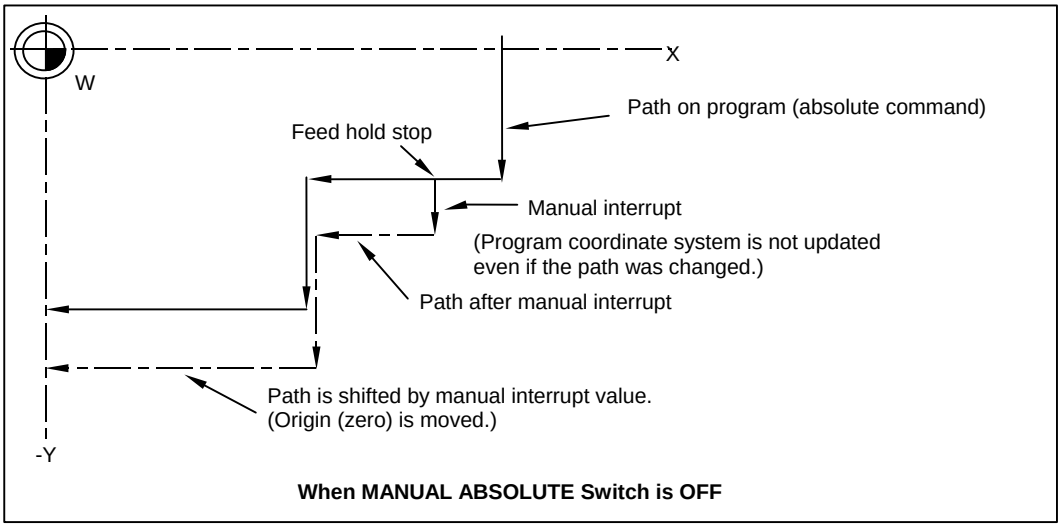


6.10 Manual Absolute

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

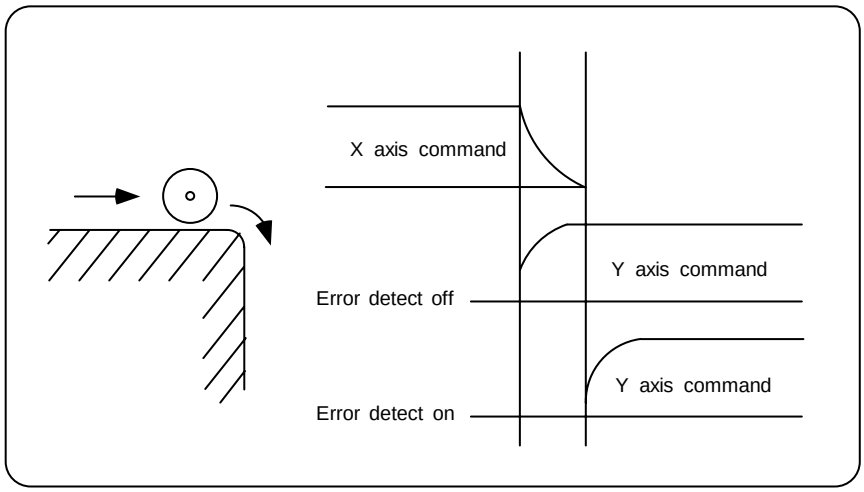


When MANUAL ABSOLUTE Switch is OFF

(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.

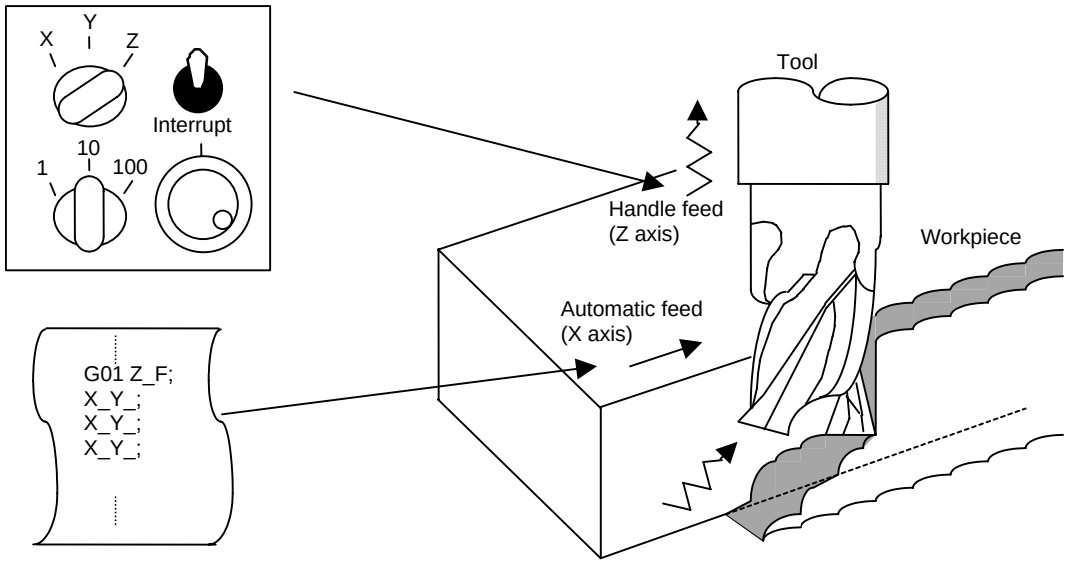
6.14 Manual/Automatic Synchronous Feed

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.)

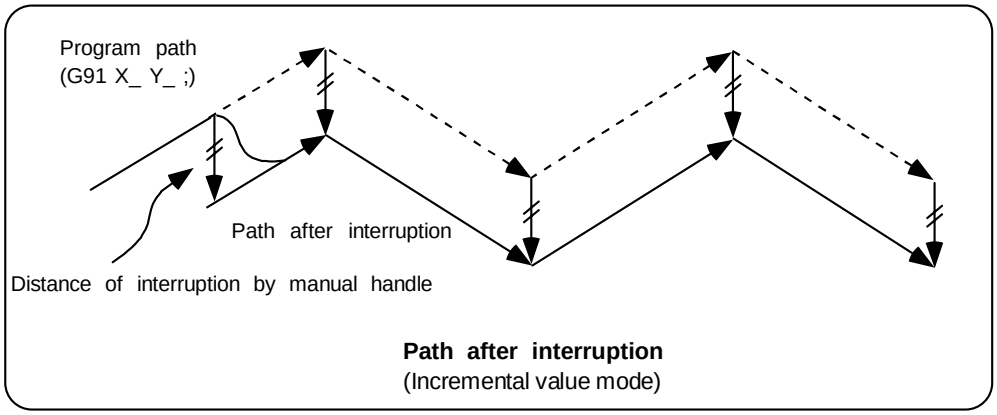
(4) If an attempt is made to execute interruption at a speed exceeding the clamp speed, the reading on the handle scale does not match the distance of interruption.

(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.

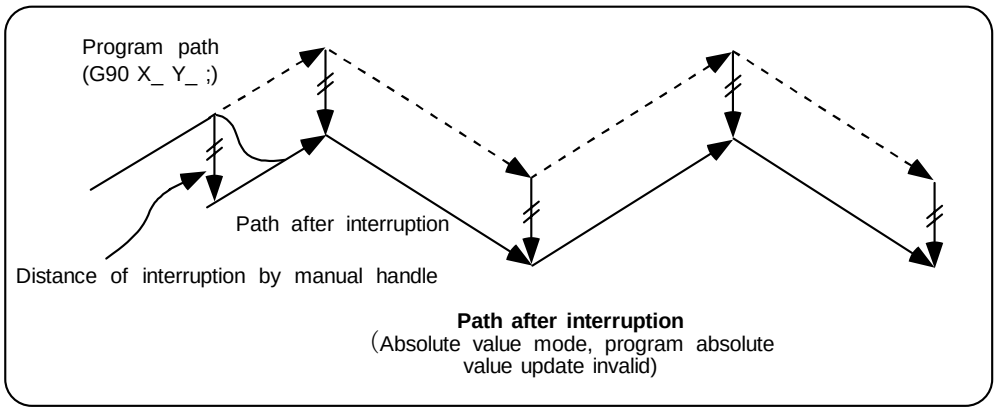
(Note) Each of the POSITION and MACHINE display includes the distance of handle interruption.

For automatic handle interruption, select whether absolute data is updated or not, as follows:

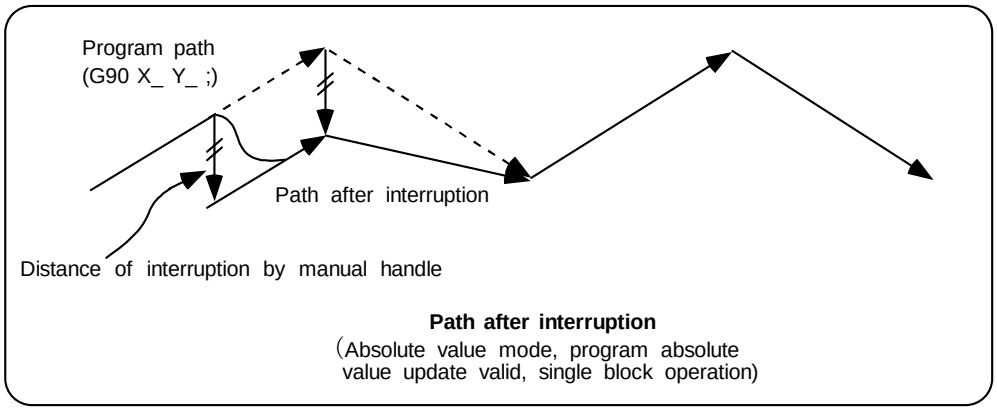
1) Using machine parameters, set whether the manual absolute changeover switch or parameters are used.
2) If the parameters are used, set whether or not absolute data is updated for each axis, by using machine parameters other than those in item.
3) If the manual absolute changeover switch is used, use the machine operation panel switch for selection.

Absolute value update conditions for automatic handle interruption

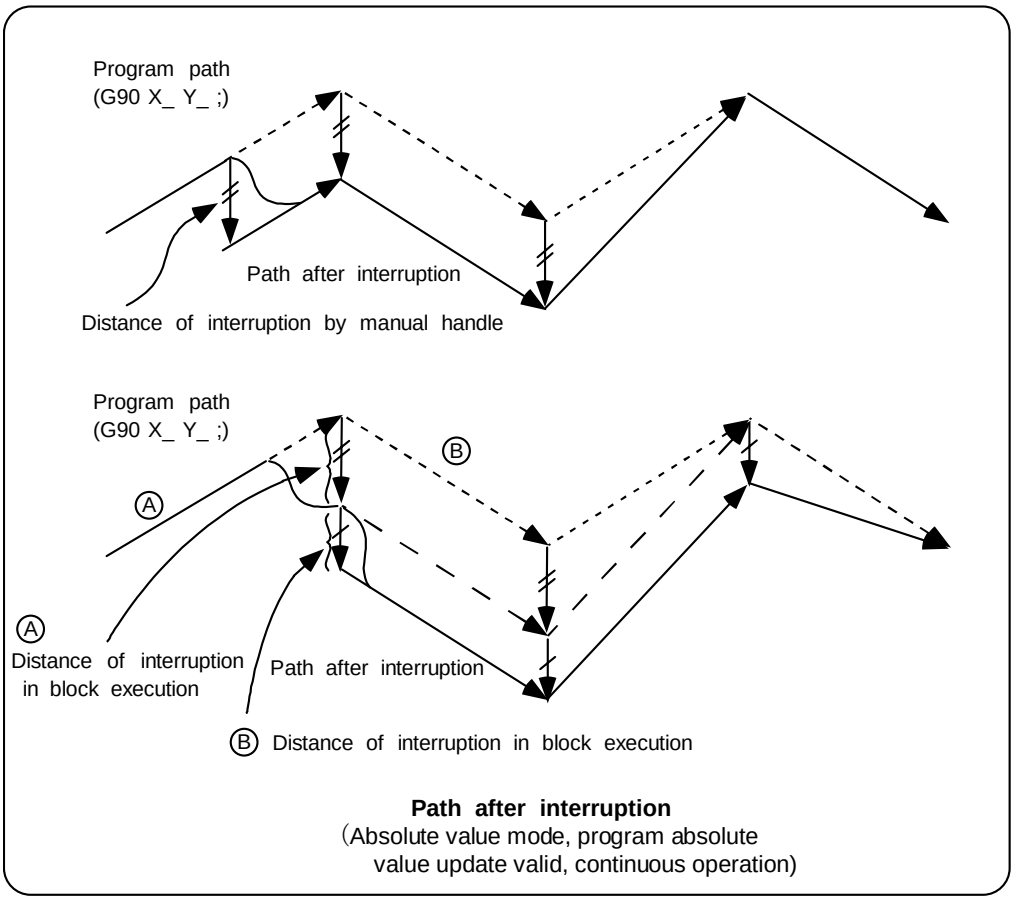
			Absolute data update		MONITOR 2 screen manual interruption distance display
Parameter "1145 I_abs"	On "1"	Parameter "#1061 intabs" (Every axis)	On "1"	Absolute value is updated.	Not updated.
			Off "0"	Absolute value is not updated.	Updated.
	Off "0"	PLC interface manual absolute switching	On	Absolute 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

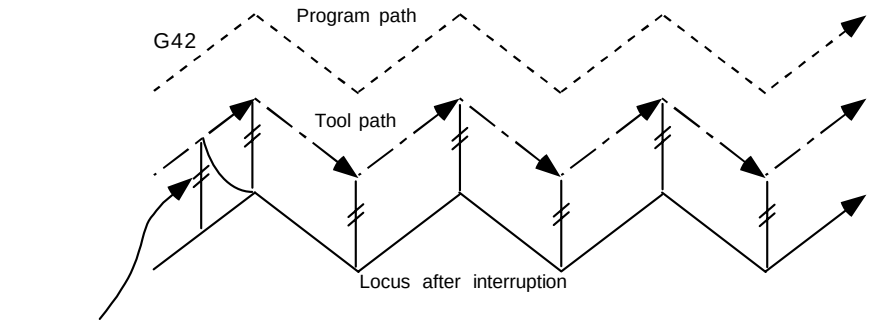
Special movement described below relates only to the tool radius compensation plane axis. It has no influence on the other axes.
At time of tool radius compensation (G41, G42):
In incremental value mode — The quantity of deviation equals the distance of interruption.
In absolute value mode — If handle 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.

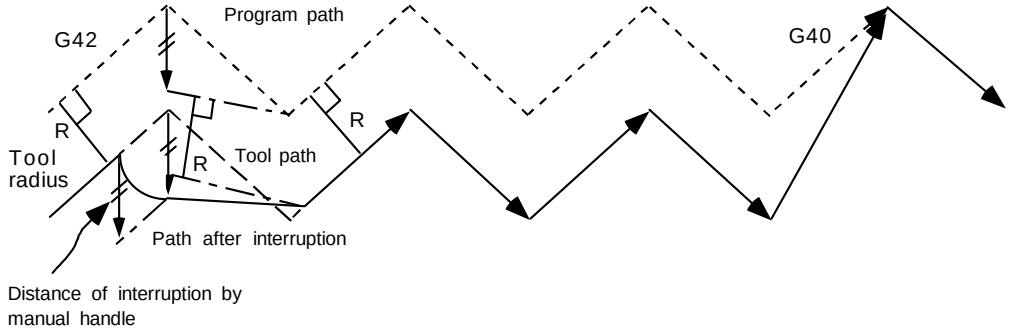
At this time, four or more blocks may not exist between

- Block for which the interruption has been completed and
- Block that contains the tool radius compensation cancel command (G40).

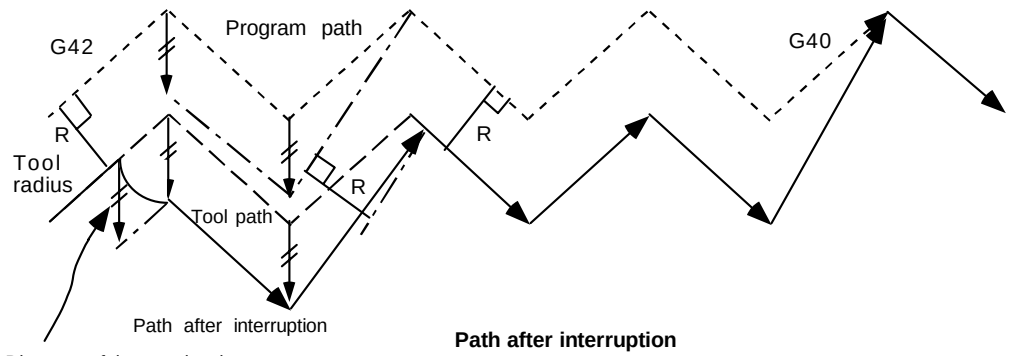
In this case, the proper tool path will return at the block next to the tool radius compensation cancel command (G40).



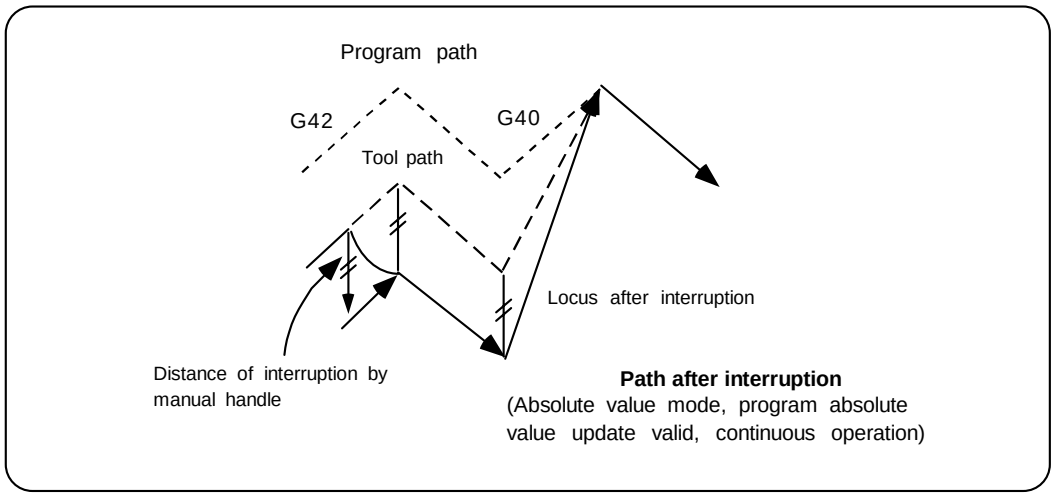
Path after interruption
(Incremental value mode, continuous operation)



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

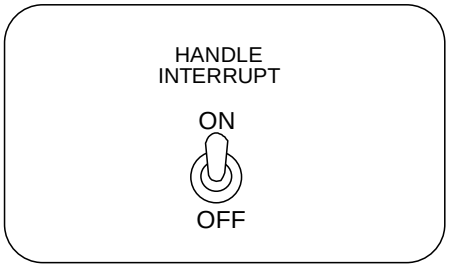
Interrupt amount is reset when
(1) dog reference position return is executed;
(2) emergency stop is released;
(3) reset rewind or reset 2 is executed; or
(4) reset 1 is executed when the interrupt amount reset parameter is ON

6.15.8 Operation Sequence

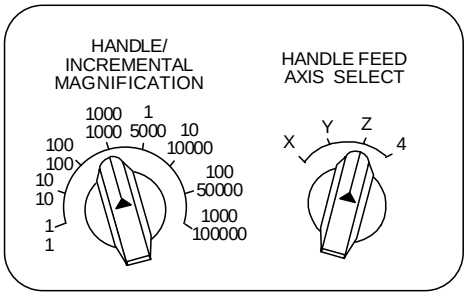
An operation example is given where automatic operation of XYZ axes is executed in the memory operation mode and the Z axis is used as a handle interrupt axis.

Perform automatic operation.

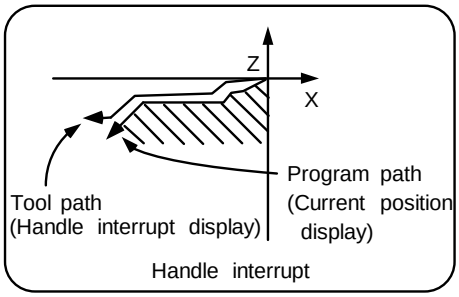
Turn on the HANDLE INTERRUPT switch on the machine operation panel.



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.

6.16 Deceleration Check

(1) Function

With the deceleration check function, a deceleration stop is executed at the block joints before the next block is executed, preventing corner roundness by reducing the machine shock that occurs when the control axis feedrate is suddenly changed.
The conditions for executing the deceleration check are as follows.

(a) Deceleration check during rapid traverse
During the rapid traverse mode, a deceleration check is always carried out when the block movement finishes before executing the next block.

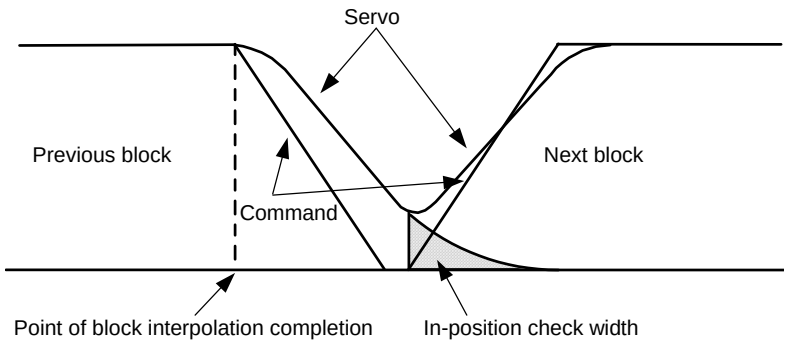
(b) Deceleration check during cutting feed
A deceleration check is carried out at the block joints before executing the next block when any one of the following conditions is valid during the cutting feed mode.

1) When the error detect switch (external signal) is ON.
2) When G09 (exact stop check) is commanded in the same block.
The G09 command is issued in the same block as the cutting command. It is an unmodal command.
3) When G61 (exact stop check mode) has been selected.
The G61 command is a modal command. The modal is canceled by the following commands.
G61.1 ..... High-accuracy control
G62 ..... Automatic corner override
G63 ..... Tapping mode
G64 ..... Cutting mode

(c) Deceleration check and parameters
In a deceleration check, the in-position width is set in the parameters (servo parameter #2224 SV024), and the check is carried out with that value as a reference.
In addition, using programmable in-position check function allows to designate temporary in-position width in a machining program.

(2) Deceleration check method

The servo system position error amount is confirmed to be less than the parameter (#2224 SV024 In-position width) setting value before the start execution of the next block.



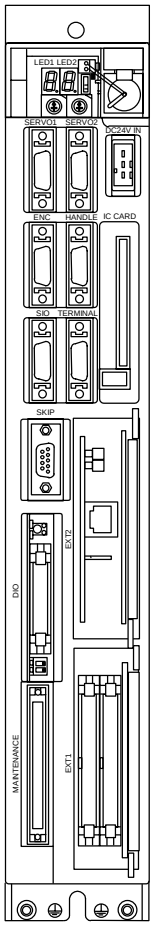
Separate in-position widths can be set for the rapid traverse and cutting feed. In blocks where there is little danger of interference with the workpiece, a reduction in acceleration time can be achieved by setting a large in-position value.
The in-position width to be designated in the machining program is unmodal, and commanded in the same block as G09.
The parameter setting value is used when there is no in-position width command in the machining program.

CHAPTER 3 MAINTENANCE

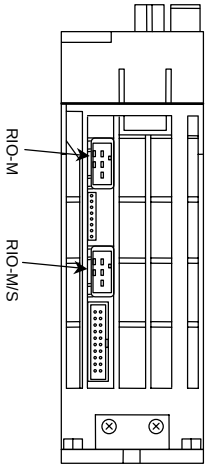
1. Various Switches

1.1 Layout of Rotary Switches on Control Unit

<NC body>



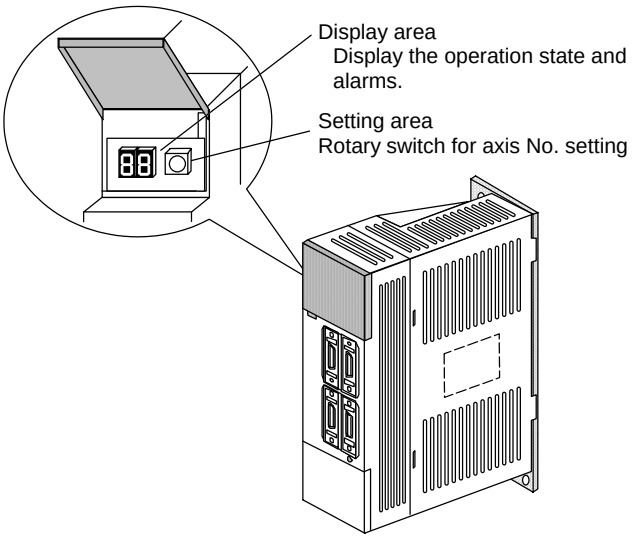
Front view



Bottom view

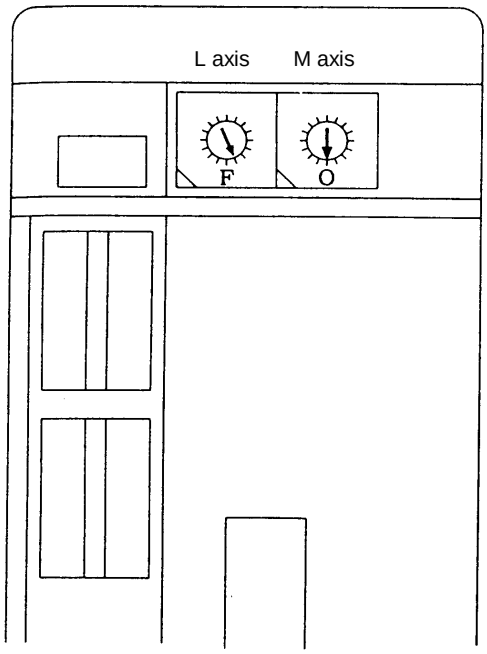
<SVJ2 Series drive unit>

The axis No. must be set with the rotary switches before turning the power ON. The rotary switch setting is validated when the power is turned ON.



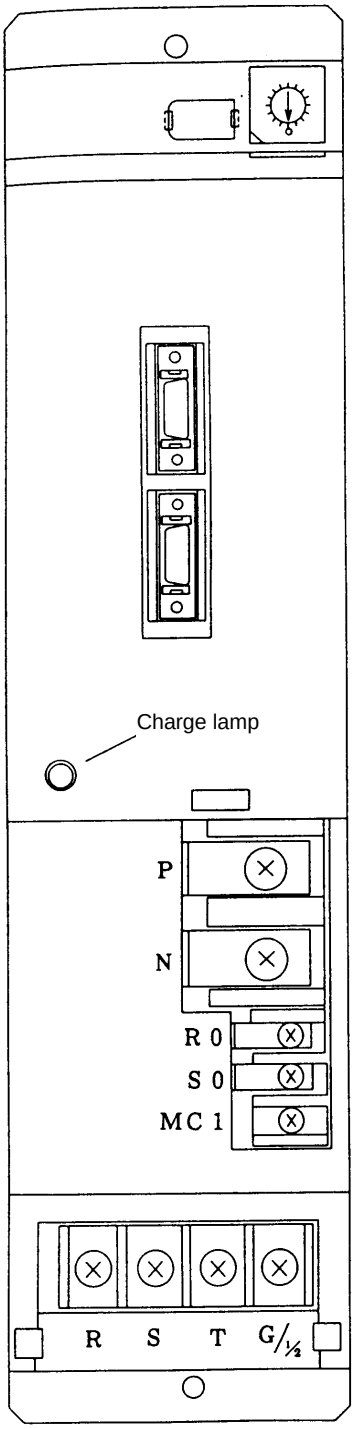
Rotary switch setting	Set axis No.
0	1st axis
1	2nd axis
2	3rd axis
3	4th axis
4	5th axis
5	6th axis
6	7th axis
7	Use not possible
8	
9	
A	
B	
C	
D	
E	
F	Axis not used

<V Series drive unit>



Function	Setting	Meaning
Axis No. setting CS	0	1st axis
	1	2nd axis
	2	3rd axis
	3	4th axis
	4	5th axis
	5	6th axis
	6	7th axis
	7 to E	Use not possible
	F	Non-used axis selection

<CV Series power supply>



The rotary switch (SW1) is set according to the following table.

SW1. setting	CV usage state
0	During operation with contactor (deposits are detected)
1	During operation without contactor
2	Setting prohibited
3	
4	
5	
6	
7	
8	Setting prohibited
9	
10	
11	
12	
13	
14	
15	

2. Startup and Adjustment Procedures

2.1 Checking the Connection

Refer to the Connection Manual and check the connection of each unit and communication terminal, etc.
Pay special attention to the position of the input power supply and connectors, etc. It is recommended to leave the servo motor and spindle motor power cables disconnected until setting of the parameters, etc., is completed.

⚠ CAUTION

Always ground the signal cable to ensure stable operation of the system. Use one-point grounding to provide the same potential at the control unit body, distribution panel and machine.

2.2 Setting the Various Switches

(1) Control unit setting switch (CS1)

A rotary switch: CS1 (left side) is located at the top front of the control unit.
Normally, the rotary switch is set to "0". Set it as shown below as necessary.

Switch	Operation	Application
0	Normal operation mode	Normal operation
1	PLC program stops	PLC development work
2	Not used	
3	Maintenance mode used by manufacturer	Do not use.
4	Not used	
5	Maintenance mode used by manufacturer	Do not use.
6	Not used	
7	Memory all erase	Do not use.
8 to F	Maintenance mode used by manufacturer	Do not use.

(2) Control unit setting switch (CS2)

A rotary switch: CS2 (right side) is located at the top front of the control unit.
This switch is used when two or more C6/C64 control units are to be connected to a single communication terminal. (Use it along with CS1 and SW1 switches.)

Switch	Operation
0	Normal operation mode
1	Designation of the station No. "n0" for multiple C6/C64 controllers.
2	Designation of the station No. "n1" for multiple C6/C64 controllers.
:	:
F	Designation of the station No. "n15" for multiple C6/C64 controllers.

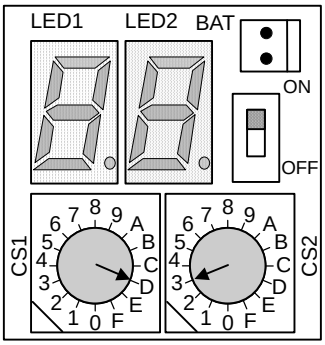
Procedure

(a) Turn the C6/C64 controller power OFF. (To set the station No.)

Setting of rotary switches (CS1, CS2) and slide switch (SW1)

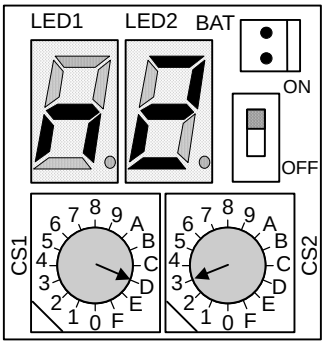
Rotary switch (CS1)	:0 to D	setting mode for multiple C6/C64 controllers
Slide switch (SW)	:OFF to ON	setting mode for multiple C6/C64 controllers
Rotary switch (CS2)	:0 to 1~F	station No.(n0:1, n1:2, ... n15:F)

(ex.) Following figure indicates that the station No. is set to "2".



(b) Turn the C6/C64 controller power ON. (To confirm the station No.)

(ex.) Following LED indicates that the station No. is set to "2".



(c) Turn the C6/C64 controller power OFF. (To make the operation enable)

Restore the settings of rotary switches (CS1, CS2) and slide switch (SW1) to their previous setting.

(3) Remote I/O unit setting switches

A lever type switch (DS1) and rotary switch (CS) are provided at the front center of the remote I/O unit.
The DX100 has one each of these switches, and the DX110/120 has two each of these switches.
Set the two levers of all DS1 switches to "OFF: left side".
CS1 can be set between "0" and "7". Refer to the following table and set. Note that the output (D0) of the PCB on the right of the DX110/120, looking from the front, is a 16-point output.
Make sure that the CS setting No. is different from the other CS setting Nos.

Remote I/O unit CS setting

The device used by the PLC is determined by the setting of the rotary switch for setting the No. of channels.

Rotary switch No.	Device No. read in		Output device No.		Analog output (AO)
	RIO-M	RIO-M/S	RIO-M	RIO-M/S	
0	X00~X1F	X100~X11F	Y00~Y1F (Y0F)	Y100~Y11F (Y10F)	The rotary switches correspond to the file registers R100 to R103 in order of small numbers.
1	X20~X3F	X120~X13F	Y20~Y3F (Y2F)	Y120~Y13F (Y12F)	
2	X40~X5F	X140~X15F	Y40~Y5F (Y4F)	Y140~Y15F (Y14F)	
3	X60~X7F	X160~X17F	Y60~Y7F (Y6F)	Y160~Y17F (Y16F)	
4	X80~X9F	X180~X19F	Y80~Y9F (Y8F)	Y180~Y19F (Y18F)	
5	XA0~XBF	X1A0~X1BF	YA0~YBF (YAF)	Y1A0~Y1BF (Y1AF)	
6	XC0~XDF	X1C0~X1DF	YC0~YDF (YCF)	Y1C0~Y1DF (Y1CF)	
7	XE0~XFF	X1E0~X1FF	YE0~YFF (YEF)	Y1E0~Y1FF (Y1EF)	

The values shown in parentheses are the device range when mounted to the right side of the unit.

(4) Servo drive unit setting switch

A rotary switch for setting the axis No. is located in the window at the top front of the servo drive unit. Refer to the following table and set the axis No.

Rotary switch No.	Axis No.
0	1st axis
1	2nd axis
2	3rd axis
3	4th axis
4	5th axis
5	6th axis
6	7th axis
F	Non-used axis selection

2.3 Turning the Power ON, Initializing the Memory, and Setting the Parameters

(1) Turning the power ON

Always check the cable connections, etc., before turning the power ON. Take special care to the power system connection state.
After turning the power ON, confirm that the screen can be changed.

(2) Setting the basic specification parameters and initializing the memory

1) Refer to section "2.4.4 Setup Parameter" in the "Chapter 1 Screen Operations", and display the screen for the setup parameters.
2) Refer to the Parameter Manual, and set the base specifications parameters (#1001 to #1019, #1025 to #1043).
3) Set "#1060 SETUP" (normally set to "0") to "1", and execute the one-touch setup. The memory will be initialized according to the basic parameters.
4) Turn the control unit's input power OFF and ON.

This completes the initialization of the memory.

(3) Setting the parameters

1) Refer to section "2.4.4 Setup Parameter" in the "Chapter 1 Screen Operations", and display the screen for the setup parameters.
2) Refer to the Parameter Manual, and set the parameters.

⚠ CAUTION

⃠ Do not change the setup parameters without consent from the machine maker.

3. Adjustment of Dog-type Reference Point Return

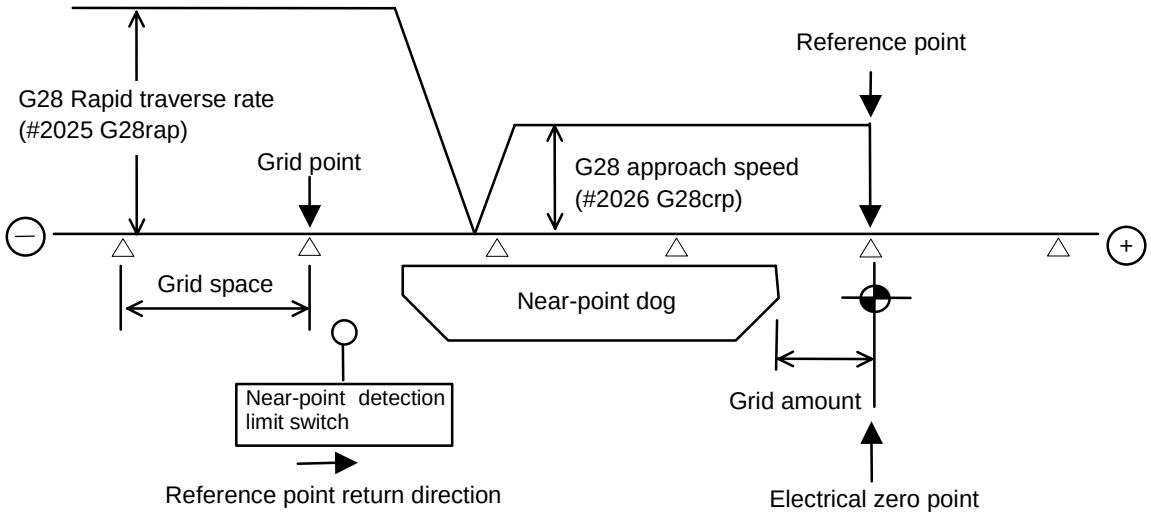
3.1 Outline

The relative position detection and absolute position detection type position detection systems are available. The methods of returning to the reference point include the dog-type reference point return and the dogless-type reference point return.
The method of adjusting the dog-type reference point return using the relative position detection is described in this section. Refer to the section "4. Absolute Position Detection System" for details on adjusting the absolute position detection.

3.2 Dog-type Reference Point Return

(1) Reference point return operation

When the dog-type reference point return is executed, the machine will move at the commanded speed. When the near-point detection limit switch kicks the near-point dog (when the near-point detection signal turns OFF), the machine decelerates to a stop. After decelerating to a stop, the machine moves at the G28 approach speed, and positions to the first grid point after leaving the near-point dog.
This grid point is called the electrical zero point. Normally, this electrical zero point position is the reference point.
For the reference point return using the relative position detector, the first return after turning the power ON is carried out with the dog-type reference point return. The second and following returns are carried out with high-speed reference point return. High-speed reference point return is a function that directly positions to the reference point saved in the memory without decelerating at the near-point dog.
If reference point return has not been executed even once after turning on the power while using relative position detection, the program error (P430) will occur when movement commands other than G28 are executed.



(2) Reference point

The reference point is the point positioned to when the dog-type reference point return is executed. Note that a separate setting method is used for the absolute position detection.
The reference point is the point positioned to with the manual reference point return and G28 command in the machining program.
Using parameters, the reference point can be shifted from the electrical zero point position.

(3) Grid point

The position detector has a Z-phase that generates one pulse per rotation. The 0-point position of this Z-phase is the grid point. Thus, there is a grid point per rotation of the position detector, and the machine has many grid points at a set pitch.
The grid point can be set per grid space by setting the grid space (SETUP PARAM. #2029 grspc). Thus, multiple grid points can be set per detector rotation.

(4) Grid space

The distance between the grid points is the grid space. The grid space can be set in mm units with the SETUP PARAM. #2029 grspc.

(5) Grid amount

The grid amount expresses the distance from when the near-point detection limit switch leaves the near-point dog and reaches the grid point (electrical zero point) when the dog-type reference point return is executed.
The grid amount can be confirmed with "GRID" on the "ALM/DIAG" "SERVO MONITOR (2)" screen of the setting and display unit.
After setting the grid mask, the grid amount shows the distance from the grid mask OFF to the grid point.

(6) Basic machine coordinate system zero point

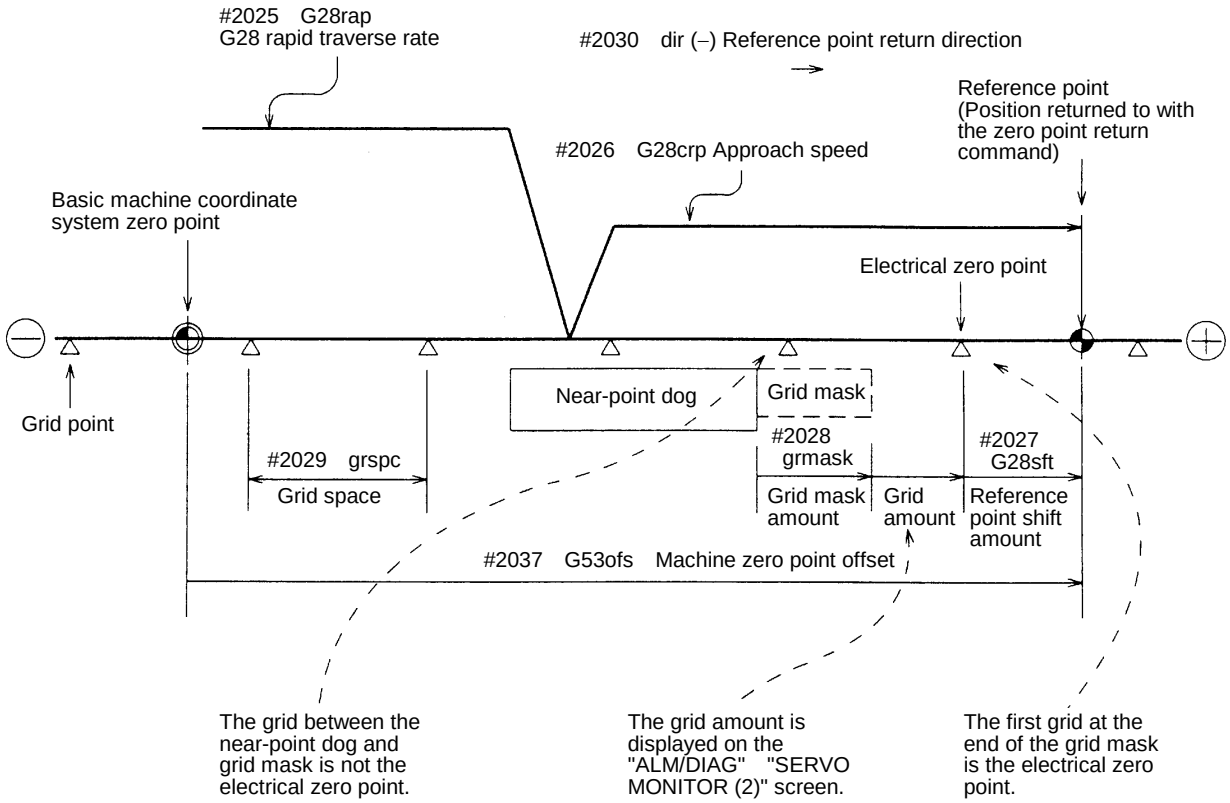
The basic machine coordinate system is a coordinate system that expresses a position decided characteristically for the machine.
The following positions are expressed with the machine coordinate system.

- No. 1 to No. 4 reference point position
- Workpiece coordinate system zero point position
- Stored stroke end position
- Soft limit position
- Chuck barrier position

Usually, the machine coordinate zero point position is the same as the reference point position, but it can be shifted with the parameters.

3.3 Reference Point Return Parameters

(1) Reference point return operation and parameter related drawing



(2) G28 rapid traverse rate (#2025 G28rap)

This parameter designates the feedrate for dog-type reference point return in manual operation and automatic operation.
The feedrate during high-speed reference point return will be the rapid traverse rate (SETUP PARAM. #2001 rapid).

(3) G28 approach speed (#2026 G28crp)

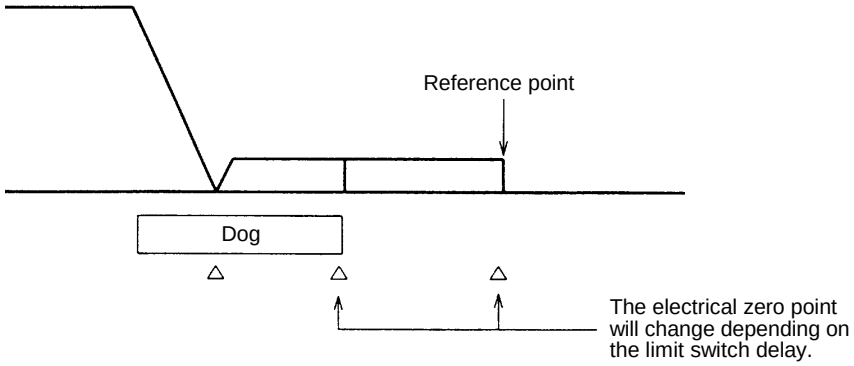
This parameters sets the approach speed (creep speed) to the reference point after decelerating to a stop by the near-dog detection. The creep speed is accelerated and decelerated in steps (acceleration/deceleration zero), so if the speed is fast, mechanical shock, etc., could occur. The creep speed should be set between 100 and 300 mm/min., and within 500 mm/min. at the fastest.

(4) Reference point shift amount (#2027 G28sft)

This parameter can set the shift amount for shifting the reference point from the electrical zero point.
The shift direction can be set only in the reference point return direction.
If the reference point shift amount is "0", the grid point (electrical zero point) will be the reference point.

(5) Grid mask amount (#2028 grmask)

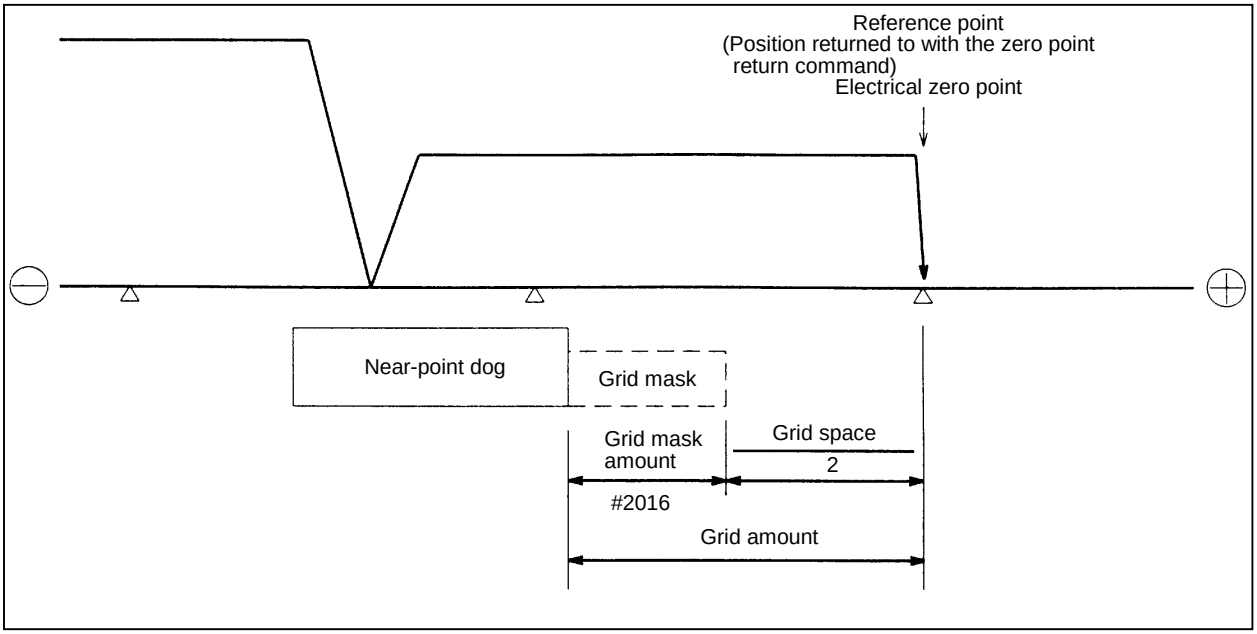
The electrical zero point is the first grid point after the dog is kicked.
If the grid point is at the position where the near-point dog is kicked, the electrical zero point will be the grid point at the position where the dog is kicked because of the delay of the limit switch operation. Thus, the next grid point will be applied, or in the end, the reference point position may be deviate by the amount of the grid space.
Thus, the position that the dog is kicked must be at the approximate center of the grid space.



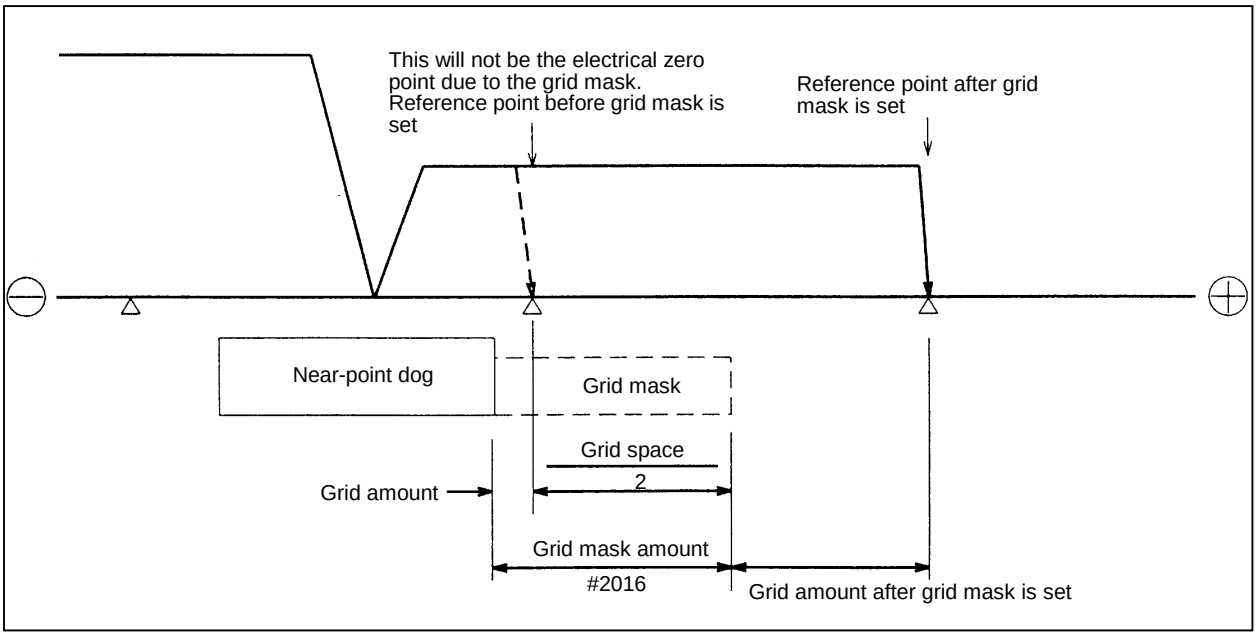
Adjustments can be made by changing the near-point dog or by setting the grid mask amount.
Setting the grid mask has the same effect as lengthening the near-point dog.
If the grid amount is near the grid space or 0, the grid point will be at the position that the near-point dog is kicked, so set a grid mask.
The grid mask amount is set so that the grid mask is one-half of the grid space.
The grid mask amount can be set only in the reference point return direction.
The grid amount and grid space can be confirmed on the "SERVO MONITOR" screen.
Refer to the grid mask amount calculation expressions on the next page for the grid mask amount values.

Grid mask amount calculation expression

When $\frac{\text{Grid space}}{2}$ < Grid amount Grid mask amount = Grid amount − $\frac{\text{Grid space}}{2}$



When $\frac{\text{Grid space}}{2}$ > Grid amount Grid mask amount = Grid amount + $\frac{\text{Grid space}}{2}$



(6) Grid space (#2029 grspc)

This parameter sets the distance between grids.
The normal grid space is the ball screw pitch (SETUP PARAM. #2218 PIT) value or the movement amount per motor rotation set as a mm unit.
To make the grid space smaller, set a divisor of the grid space.

Calculation expression for movement amount per motor rotation

- When linear feed mechanism is ball screw

$$\text{Movement amount per motor rotation} = \frac{\text{Motor side gear ratio}}{\text{Machine side gear ratio}} * \text{Ball screw pitch}$$

- When linear feed mechanism is rack & pinion

$$\text{Movement amount per motor rotation} = \frac{\text{Motor side gear ratio}}{\text{Machine side gear ratio}} * \text{No. of pinion gear teeth} * \text{Rack pitch}$$

- For rotary axis

$$\text{Movement angle per motor rotation} = \frac{\text{Motor side gear ratio}}{\text{Machine side gear ratio}} * 360$$

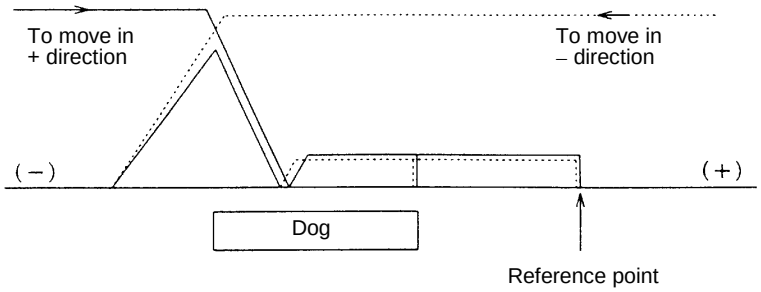
$$N = \frac{PC1}{PC2} * PIT$$

N = Movement amount per motor rotation
PC1 = Motor side gear ratio
PC2 = Machine side gear ratio
PIT = Ball screw pitch

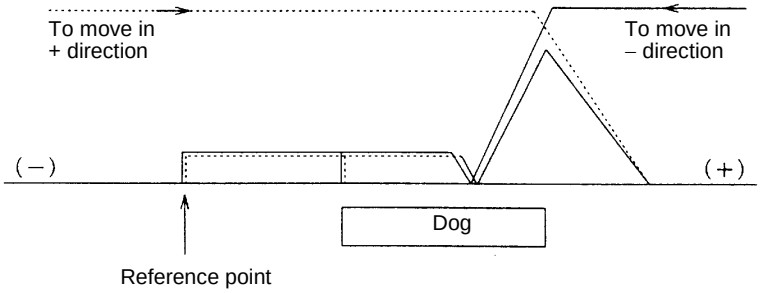
(7) Reference point return direction (#2030 dir (–))

This parameter sets the direction to move after the limit switch kicks the dog causing a deceleration stop during dog-type reference point return. The direction is either positive "0" or negative "1".
If the reference point position is in the positive direction from the near-point dog, set "0".
If the reference point position is in the negative direction from the near-point dog, set "1".

(a) When reference point return direction is positive (+)



(b) When reference point return direction is negative (–)



(8) Axis with no reference point (#2031 noref)

"0" is set for the axis to carry out dog-type reference point return and the axis for absolute position detection.
"1" is set for the axis that does not carry out reference point return during relative position detection.

(9) Machine coordinate system offset (#2037 G53ofs)

This parameter is set when the basic machine coordinate system zero point position is to be shifted from the reference point position.
When "0" is set, the reference point position will be the position of the basic machine coordinate system zero point.
G53ofs sets the reference point position from the basic machine coordinate system zero point as a basic machine coordinate system coordinate value. The machine value becomes this value with reference point return after the power is turned on, and the basic machine coordinate system is established.

3.4 Dog-type Reference Point Return Adjustment Procedures

Adjust the dog-type reference point return with the following steps.

(1) Set the zero point return parameter.
Set the reference point shift amount to 0.
Set the grid mask amount to 0.
(2) Turn the power off and on, and then execute reference point return.
(3) Display the "ALM/DGN" "SERVO MONITOR (2)" screen on the setting and display unit.
The grid space and grid amount values can be read.
(4) Calculate the grid mask amount with the grid mask amount calculation method.
(5) Display the "SETUP PARAM" screen.
Set the grid mask amount.
(6) Turn the power off and on, and then execute reference point return.
(7) Display the "ALM/DGN" "SERVO MONITOR (2)" screen on the setting and display unit.
The grid space and grid amount values can be read.
If the grid amount value is approx. half of the grid space, the grid mask amount has been correctly set.
If the value is not approx. half, repeat the procedure from step (1).
(8) Set the reference point shift.
(9) Turn the power off and on, and then execute reference point return.
(10) Set G53ofs.

4. Absolute Position Detection System

4.1 Outline

The absolute position detection function detects the machine movement amount while the power is OFF. This allows automatic operation to be started without carrying out zero point return after the power is turned ON. This function is reliable as it carries out a mutual check of the feedback amount from the detector, and checks the absolute position unique to the machine, etc.
To carry out absolute position detection, the machine zero point must be positioned, and the absolute position must be established. This machine uses only the dogless-type absolute position detection.

(1) Dogless-type absolute position detection

The absolute position is established by setting a random coordinate at a random position without using the dog.
The basic position can be determined with the following two methods.
- Machine end stopper method
- Marked point alignment method

For the machine end stopper method, the manual initialization and automatic initialization methods can be used.

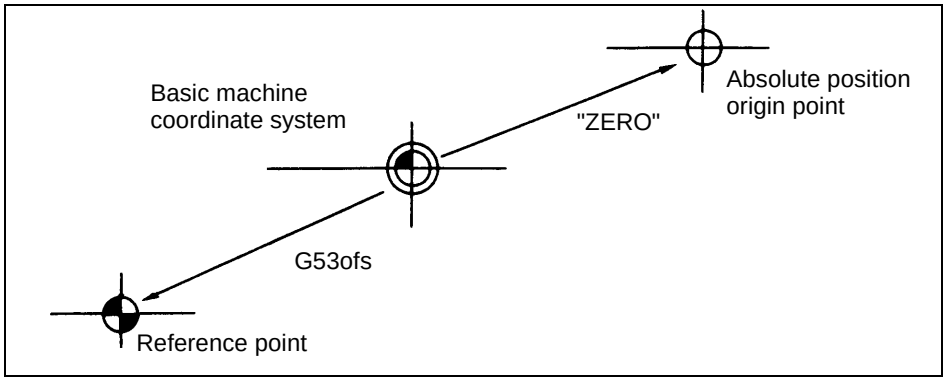
(2) Dog type absolute position detection

The absolute position is established by executed dog type zero point return.

The validity and method of the absolute position detection system can be selected with parameters for each axis. Note that the servo amplifier and detector must have the absolute position detection specifications.

4.2 Absolute Position System Coordinate System

Using the mechanical basic position (machine end or origin point) or the electrical basic position (machine end or grid point just before origin point) as the absolute position origin point, the basic machine coordinate system zero point is set at the position obtained by reversing the "ZERO" parameter sign. The reference point is set at the "G53ofs" parameter from the basic machine coordinate system's zero point.



Dogless absolute position coordinate system

"ZERO" : Coordinate value of absolute position origin point looking from basic machine coordinate system zero point (ABS. POSITION SET screen "#2 ZERO").
"G53ofs" : Coordinate value of reference point looking from basic machine coordinate system zero point (parameter "#2037 G53ofs").

(Note) Whether to use the mechanical basic position or electrical basic position as the absolute position origin point is selected with parameter "#2059 zerbas".

4.3 Starting Up the Absolute Position Detection System

The zero point must be initialized before the absolute position detection system is started up. The coordinate system is established and operation is enabled when the zero point is initialized.

(1) Operation when absolute position is not established

If the zero point has not been initialized even once or if the absolute position is lost, the following alarm and non-initialized axis will be output. The coordinate system is not set in this state, so the limits given in following table will be applied to each mode. Initialize the zero point and establish the coordinate system.

Alarm: Z70 ABS. ILLEGAL
Z71 DETECTOR ERROR

Operation in each mode

Operation mode	Absolute detection method	
	Dogless-type	**Dog type**
Memory/MDI	Movement command invalid **(Note 1)** (Including G28)	Movement command invalid **(Note 1)** (Valid for G28)
Jog feed	Valid	Valid
Rapid traverse	Valid	Valid
Handle	Valid	Valid
Step	Valid	Valid
Zero point return	Starting not possible **(Note 2)**	Starting possible

(Note 1) The program error "P430 AXIS NOT RET." will occur.
(Note 2) If the axis is started before the absolute position is established, the error "M01 OPERATION ERROR 0024" will occur.
(This is valid for an axis for which the absolute position has been established.)

(2) Selecting the zero point initialization method

Select the zero point initialization method with the following parameter.
#2049 type 1: Dogless-type Machine end stopper method
2: Dogless-type Marked point alignment method
3: Dog-type

(3) Zero point initialization

The zero point is initialized using the ABS POSITION SET screen and JOG or handle. The execution status of the initialization process is displayed at "STATE" on the ABS POSITION SET screen.
The operation methods differ according to the zero point initialization method.
Each operation procedure is described below.

■ Machine end stopper method

The machine end stopper method includes the manual initialization and automatic initialization methods.

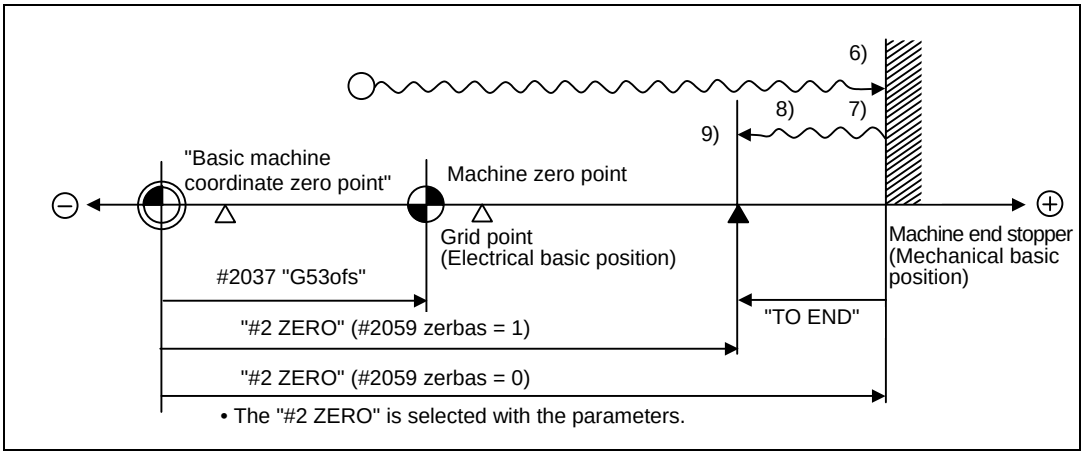
(a) Manual initialization

With this method, the axis is pressed against the machine end stopper using handle or JOG.
Set the following parameters before initializing the zero point.
(Refer to the Appendices for details.)
#2054 clpush

[Operation procedure/operation]

Operation procedure	"STATE" display
1) Select the "ABS POSITION SET" screen.	• NG when absolute position has been lost • OK when absolute position has been established
2) Select the handle mode or the JOG mode.	
3) Check that the stopper method is applied for the axis for which the zero point is to be initialized. ("TYPE" on the ABS POSITION SET screen is STOPPER.)	
4) Set "1" for "#0 INIT. SET" of the axis for which the zero point is to be initialized.	STOPPER
5) Set "#2 ZERO".	
6) Push against the machine end stopper.	
7) Check that the "STATE" display indicates pushing. (When the axis is pushed against the machine end stopper and the current limit is reached continuously for a set time, the display will change to RELEASE. The distance between the machine end and previous grid will appear at "TO END".)	RELEASE
8) Move in the opposite direction of the pushed direction.	ORIG-RTN
9) When the previous grid is reached, the movement will automatically stop. • The basic machine coordinate system will be automatically set. The absolute position is established at this stage.	OK
10) This completes the zero point initialization. Initialize all axes and then turn the power ON again.	
11) Output the parameters.	

When changing only the basic machine coordinate zero point, carry out steps 4) and 5) above and then turn the power OFF and ON.



Manual initialization operation diagram

(Note 1) If the grid point is not passed even once after the power is turned ON before pushing against the machine end, the message "NOT PASS" will appear. In this case, return one grid back, and repeat from step 6).

(Note 2) At the step 9), if the first grid point is on the grid mask ("#2028 grmask"), the axis will stop at the next grid point.
Note that the zero point shift ("#2027 G28sft") is invalid.

(b) Automatic initialization

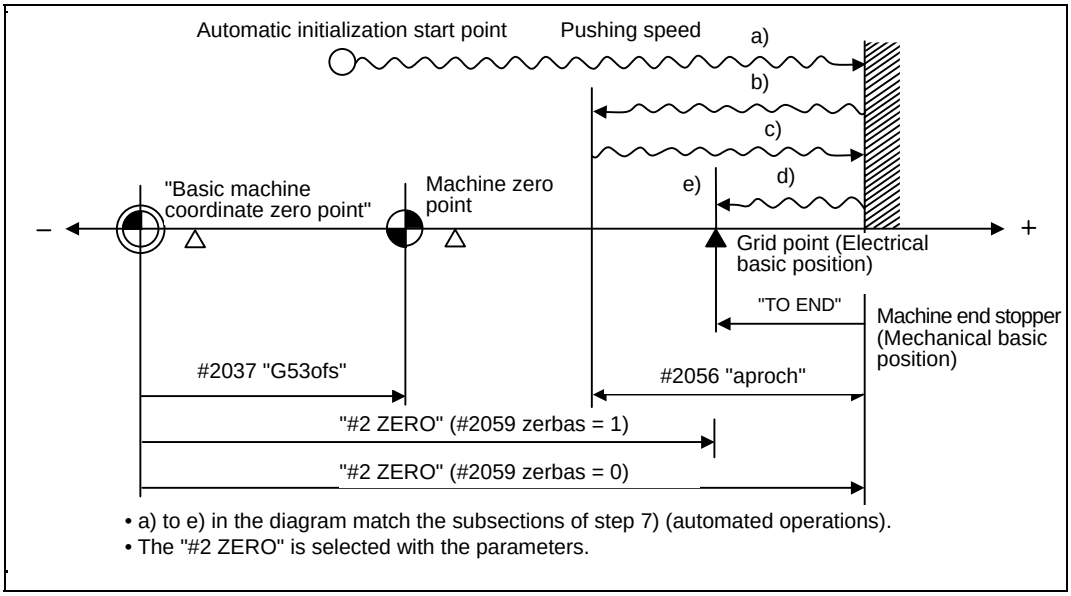
With this method, the axis is pushed against the machine end stopper, and can be used when the "INIT-SET" mode is selected. This method has the following features compared to the manual initialization method.

- The axis is pushed with the same conditions (feedrate, distance) each time, so inconsistencies in the zero point position can be reduced.
- Part of the operations is automated to simplify the zero point initialization.
  Set the following parameters before initializing the zero point.
  (Refer to the Appendices for details.)
  #2054 clpush : Current limit
  #2055 pushf : Pushing speed
  #2056 aproch : Approach point

[Operation procedure/operation]

Operation procedure		"STATE" display
1) Select the ABS POSITION SET screen.		• NG when absolute position has been lost • OK when absolute position has been established
2) Select the "INIT-SET" mode.		
3) Check that the stopper method is applied for the axis for which the zero point is to be initialized. ("TYPE" on the ABS POSITION SET screen is STOPPER.)		
4) Set "1" for "#0 INIT. SET" of the axis for which the zero point is to be initialized.		JOG-START
5) Set "#2 ZERO".		
6) Start the axis for which the zero point is to be initialized with JOG. • JOG can be started only in the "#2 ZERO" sign direction (machine end stopper direction). (If the JOG start direction is illegal, a message will appear.)		STOPPER 1
7) Automated operations	a) The axis will move at the "pushing speed" in the machine end stopper direction.	
	b) When the axis pushes against the machine end stopper and the current limit is reached continuously for a set time, the axis will return toward the approach point at the "pushing speed".	ZP-RTN
	c) After reaching the approach point, the axis will move in the machine end stopper direction at the "pushing speed".	STOPPER 2
	d) When the axis pushes against the machine end stopper and the current limit is reached continuously for a set time, the axis will return to the previous grid at the "pushing speed".	ORIG-RTN
	e) When the previous grid is reached, the movement will automatically stop. • The basic machine coordinate system will be automatically set. The absolute position is established at this stage.	OK
8) This completes the zero point initialization. Initialize all axes and then turn the power ON again.		
9) Output the parameters.		

When changing only the basic machine coordinate zero point, carry out steps 4) and 5) above and then turn the power OFF and ON.



Automatic initialization operation diagram

(Note 1) If the grid point is not passed even once after the power is turned ON, the message "NOT PASS" will appear when the approach point is reached after pushing. In this case, return one grid back, and repeat from step 6).

(Note 2) Smoothing is turned OFF (step feed) for the acceleration/deceleration during movement at the pushing speed.

(Note 3) If 0 is set for the parameter "#2056 aproch", the machine zero point will be the approach point.

(Note 4) The automatic initialization will be stopped in the following cases after starting. When stopped, "STATE" will change to JOG-START (when the mode has been changed, after the "INIT-SET" has been selected), so repeat from step 6).

- When a new absolute position detection alarm has occurred
- When READY has been turned OFF
- When the mode has been changed
- When the program has been reset

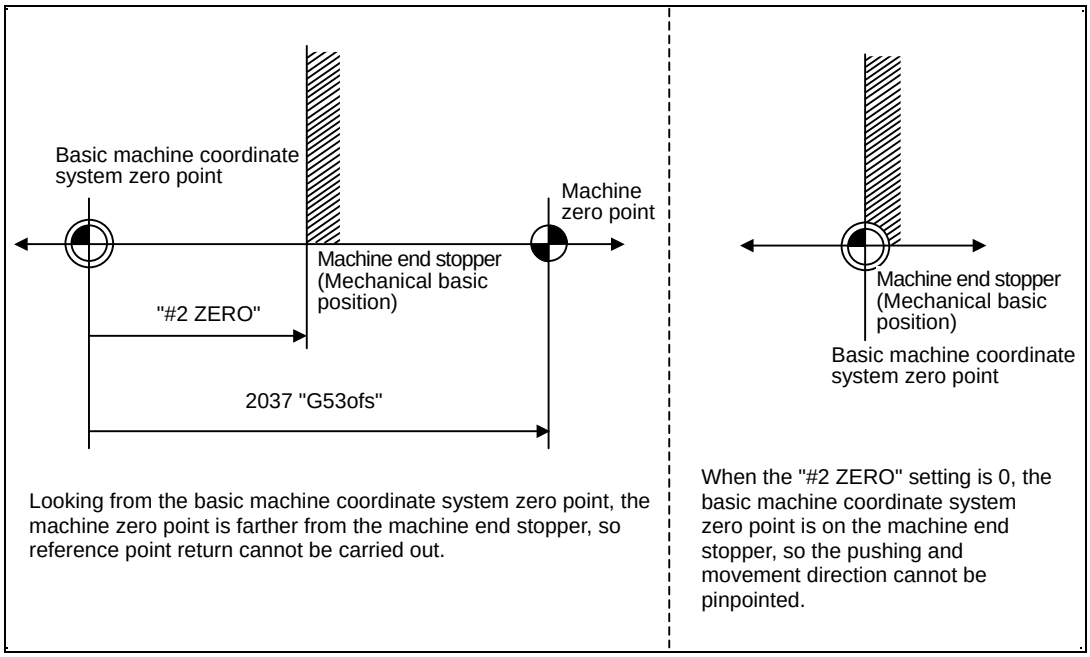
If the "STATE" is OK before starting the automatic initialization, and the power is turned ON again without resuming operation, "STATE" will return to OK.

(Note 5) At the step 7) e), if the first grid point is on the grid mask ("#2028 grmask"), the axis will stop at the next grid point.
Note that the zero point shift ("#2027 G28sft") is invalid.

(Note 6) Automatic initialization cannot be started in the following cases. The message "T01 CAN'T CYCLE ST" will appear if starting is attempted.

- When "#0 INIT. SET" is not set.
- When the "#2 ZERO" setting is inappropriate.
- When "#2055 pushf" is not set.
- When "Z71 DETECTOR ERROR 0005" has occurred.

In the above cases, if the "#2 ZERO" setting is inappropriate, this means that the relation of "#2 ZERO" and "2037 G53ofs" is inappropriate. In other words, if "#2 ZERO" is smaller than the "#2037 G53ofs", the machine end stopper will be located between the basic machine coordinate system zero point and machine zero point, so automatic initialization cannot be started. If "#2 ZERO" is set to 0, the machine end stopper direction will not be set, so automatic initialization cannot be started.
(Refer to the following diagram)



Explanatory diagram of factors preventing automatic initialization startup

■ **Marked point alignment method**

With this method, the axis is aligned to the machine's basic point (marked point) using handle or JOG.
Set the following parameters before initializing the zero point.
(Refer to the Appendices for details.)

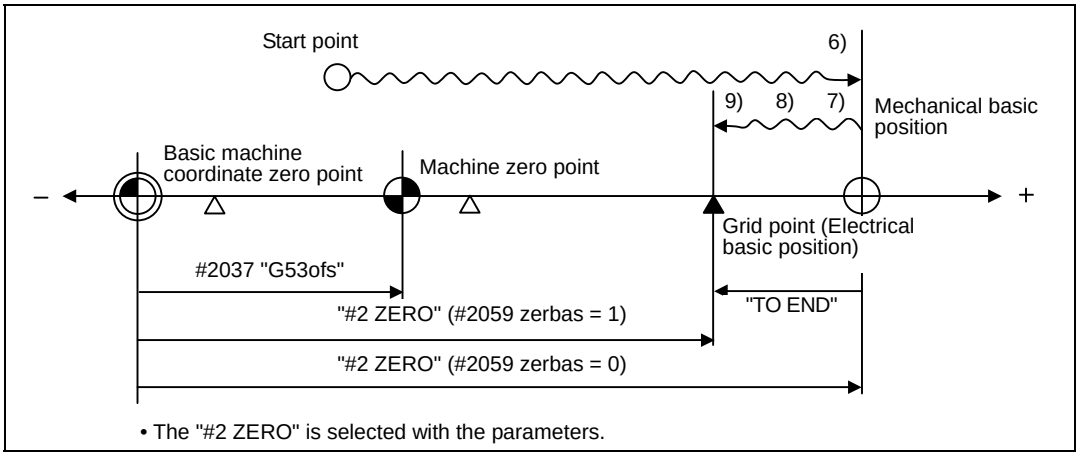
#2050 absdir :Base point of Z-direction

#2054 clpush :Current limit

[Operation procedure/operation]

Operation procedure	"STATE" display
1) Select the ABS POSITION SET screen.	• NG when absolute position has been lost • OK when absolute position has been established
2) Select the handle mode or the JOG mode.	
3) Check that the marked point alignment method is applied for the axis for which the zero point is to be initialized. ("TYPE" on the ABS POSITION SET screen is NO-STOPPER.)	
4) Set "1" for "#0 INIT. SET" of the axis for which the zero point is to be initialized.	ORIGIN SET
5) Set "#2 ZERO".	
6) Move to the machine basic position and align to the marked point.	
7) Set "1" in "#1 ORIGIN".	
8) Move in the direction designated with the parameter "#2050 absdir".	ORIG-RTN
9) When the first grid is reached, the movement will automatically stop. The basic machine coordinate system will be automatically set. The absolute position is established at this stage.	OK
10) This completes the zero point initialization. Initialize all axes and then turn the power ON again.	
11) Output the parameters.	

When changing only the basic machine coordinate zero point, carry out steps 4) and 5) above and then turn the power OFF and ON.



Marked point alignment operation diagram

(Note 1) If the grid point is not passed even once after the power is turned ON before setting the origin point, the message "NOT PASS" will appear. In this case, return one grid back, and repeat from step 6).

(Note 2) At the step 9), if the first grid point is on the grid mask ("#2028 grmask"), the axis will stop at the next grid point.
Note that the zero point shift ("#2027 G28sft") is invalid.

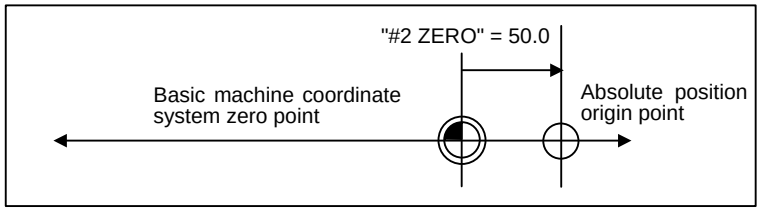
(Note 3) If movement does not take place in step 8), recheck the absdir direction.
If set to 0, the axis will move only in the forward direction, and when set to 1, will move only in the reverse direction.

(4) Common precautions for dogless-type absolute position detector

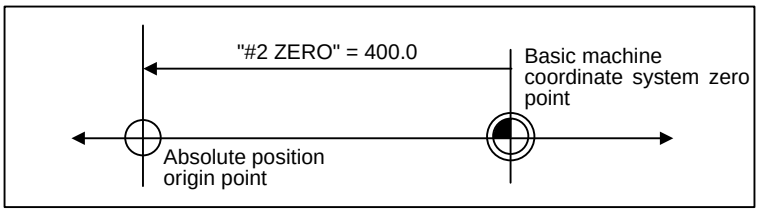
(a) Example of setting "#2 ZERO" parameter

For the "#2 ZERO" parameter, set the coordinate value of the absolute position origin point (mechanical basic position or electrical basic position" looking from the basic machine coordinate system zero point.

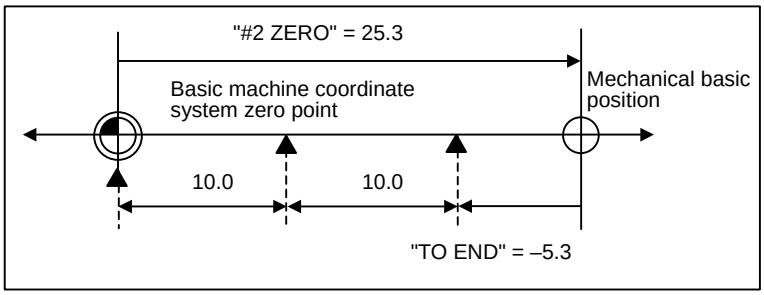
(Example 1) To set the zero point at 50.0mm before absolute position origin point on + end



(Example 2) To set the zero point at 400.0mm before the machine basic position or absolute position origin point on – end.



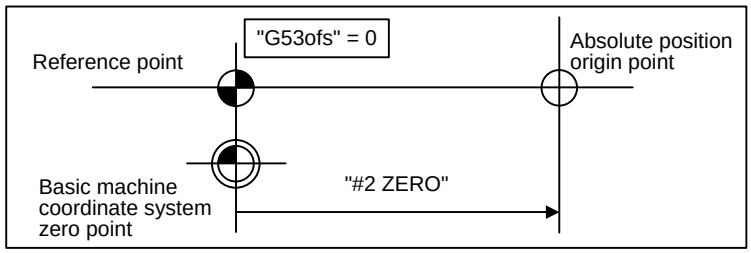
(Example 3) To set the basic machine coordinate system zero point on the grid point, calculate the "#2 ZERO" parameter setting value as shown below using the value displayed at "TO END". "TO END" shows the distance from the mechanical basic position to the previous grid point.
(Note that when setting the electrical basic position coordinate value in "#2 ZERO", the "TO END" value does not need to be considered.)
To set the third gird point as the zero point when the "TO END" display is –5.3 at the + end basic position. (Example of 10.0mm grid interval.)



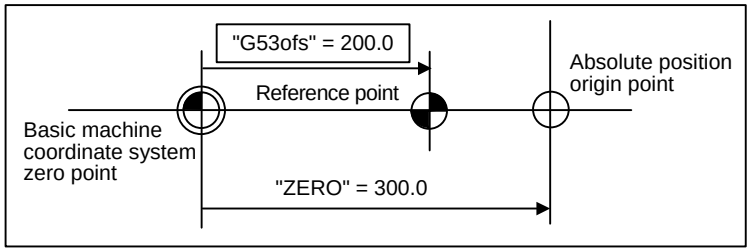
(b) Setting the reference point

The reference point can be set as shown below by setting the "#2037 G53ofs".

(Example 1) To set the reference point to the same position as the basic machine coordinate system zero point.



(Example 2) To set the reference point at a position 200.0mm to the + side from the basic machine coordinate zero point.
(To set the basic machine coordinate system zero point 300.0mm front of the absolute position origin point.)



(c) Common precautions for initialization operations

- The "#0 INIT. SET" parameter (axis for which zero point is to be initialized) can be set simultaneously for all axes or individually for each axis.
- The "#0 INIT. SET" parameter cannot be turned OFF with the keys. It is turned OFF when the power is turned ON again. If this parameter is ON for even one axis, the message "INITIAL SET (ABS)" will appear in the operation status display area of all screens, and axis in initialization will be output. The automatic, MDI and manual zero point return operations will be interlocked at this time.
- "#2 ZERO" can be set at any time as long as "#0 INIT. SET" is set to "1".
- The grid point must be passed at least once after turning the power ON before initializing the zero point. If the grid point has not been passed, "NOT PASS" will appear at the "MAC POS" display.
- When the absolute position is established, the required data will be stored in the memory.

(5) Zero point initialization for dog type absolute position detection

By executing dog type reference point return with the manual reference point return mode or automatic reference point return command (G28), the zero point will be initialized. The execution stage of the initialization will be showed in the STATE column of the "ABS POSITION SET" screen. The #0 "INIT. SET", #1 "ORIGIN" and #2 "ZERO" settings are invalid during dog type absolute position detection.

[Operation procedure/operation]

Operation procedure	"STATE" display
1) Select the "ABS POSITION SET" screen.	• NG if the absolute position is lost. • OK if the absolute position is established.
2) Confirm that the axis to be zero point initialized is a "dog type" axis. (See TYPE on the "ABS POSITION SET" screen.)	
3) Perform manual or automatic dog type reference point return.	ZP-RTN
4) The basic machine coordinate system will be established when the reference point is reached. The absolute position is established at this stage, and the zero point initialization is completed.	COMPLETE
5) Output the parameters after reference point return is completed for all axes.	

(Note 1) If the dog type reference point return is stopped by resetting, the previous state (OK or NG) will display in the STATE column.
(Note 2) After the absolute position has been established, save the necessary data in the memory.
(Note 3) With dog type reference point return, reference point return can be executed again even if the STATE is OK.

5. Zero Point Initialization for Auxiliary Axis

The following two methods can be used for zero point initialization for auxiliary axis.

- Method by commanding from PLC
- Method by operating on the auxiliary axis monitor screen (when the operation test mode signal from PLC is ON)

Dog-type method conforms to the method by commanding from PLC, and dogless-type method to the method by operating on the auxiliary axis monitor screen.

5.1 Dog-type Reference Point Return for Auxiliary Axis

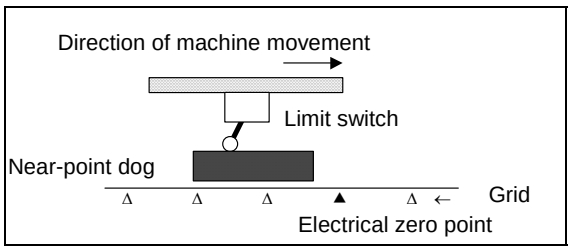
The dog-type reference point return is a method for establishing the coordinate zero point in an incremental system. The coordinate zero point is determined with the electrically determined reference point (machine specific point) used as a reference. This reference point is determined by the signals (near-point dog signals) turned ON/OFF by the near-point dog and limit switch.
In the motor end position detector there is a Z phase signal that is output once per rotation. Looking from the movable section of the machine driven by the motor, a Z phase signal is output for every set movement amount. The position at which this Z phase is output is called the grid. One specific point of these grid points is recognized as the electrical zero point by the servo amplifier. The dog signal is used as a means to designate/recognize which grid point is the electrical zero point in the servo amplifier.

Electrical zero point	→	Reference point	→	Coordinate zero point
		Determined by the electrical zero point and reference point shift amount (ZRNshift). The default shift amount is 0, and the electrical zero point and reference point are in the same position.		Determined by the reference point and the offset amount. The default offset amount is 0, and the electrical zero point, reference point, and coordinate zero point are in the same position.

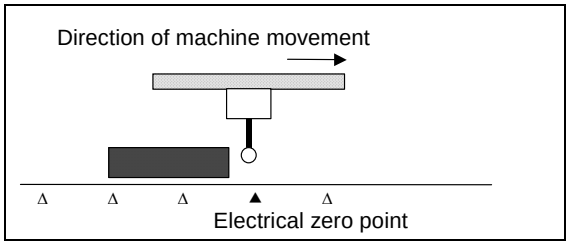
(1) Operation principle

The operation to determine the electrical zero point is explained below.
The dog signal is OFF when the limit switch is on the near-point dog. The dog signal is a B contact that is ON, when the limit switch is not on the near-point dog.

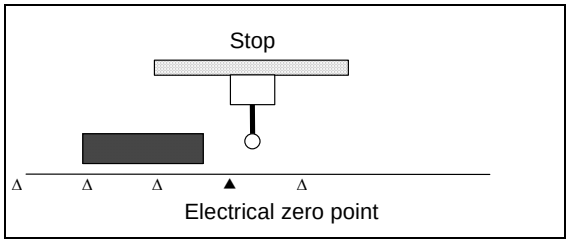
(1) When the machine movable parts are moved, the dog signal limit switch is ON according to the near-point dog, and the dog signal is OFF.



(2) When the machine movable parts are moved further in the same direction, the limit switch leaves the dog, and the dog signal turns ON.

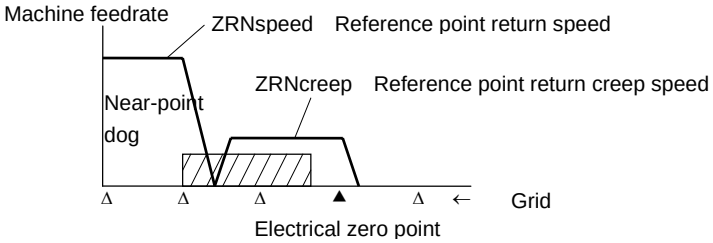


(3) The servo amplifier recognizes the first grid point after the dog signal turns ON as the electrical zero point.



(2) Execution procedure

The execution procedure for dog-type reference point return is shown below.

(1) Initial setting	Confirm that the parameter "#101 Cont1 bit D (No zero point)" setting is to 0 (zero). **< Memo >** When "#101 Cont1 bit D (No zero point)" is 1, the specification will be that there is no reference point. The machine position when the power is turned ON becomes the reference point.
(2) Set the speed	Set the parameters that designate the axis feedrate during reference point return "#110 ZRNspeed (Reference point return speed)" and "#111 ZRNcreep (Reference point return creep speed)". **< Memo >** If the reference point return speed is too fast, it may not be able to decelerate fully when the limit switch is ON, and an alarm may occur. If this alarm occurs, decrease the reference point return speed. 
(3) Designate the reference point return direction	Determine the motor rotation direction for reference point return execution with parameter "#101 Cont1 bit8 (Reference point return direction)". **#101 Cont1 bit8 Reference point return direction** / **Approach direction** 0 / Motor rotates CW and approaches 1 / Motor rotates CCW and approaches
(4) Select the reference point mode	When the "reference point return mode (ZRN)" signal is turned ON, and the start signal is turned ON, reference point return will be executed. The axis automatically stops at the electrical zero point. **< Memo >** The default settings of electrical zero point, reference point, and coordinate zero point are the same. Refer to the next section when setting the reference point and coordinate zero point to a different position than the electrical zero point.

(3) Supplemental remarks

<table>
<tr><th>No.</th><th>Abbrev.</th><th>Parameter name</th><th>Default value</th><th>Unit</th><th>Explanation</th><th>Setting range</th></tr>
<tr><td>#101</td><td>∗Cont1</td><td>Control parameter 1</td><td colspan="5">This is a HEX setting parameter. Set bits without a description to their default values.
<table>
<tr><th>bit</th><td>F</td><td>E</td><td>D</td><td>C</td><td>B</td><td>A</td><td>9</td><td>8</td><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td></tr>
<tr><th>Default value</th><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>1</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td></tr>
</table>
<table>
<tr><th>bit</th><th>Meaning when "0" is set.</th><th>Meaning when "1" is set.</th></tr>
<tr><td>1</td><td>High-speed zero point return after zero point establishment</td><td>Dog-type method for each zero point return operation</td></tr>
<tr><td>8</td><td>Reference point return direction (+)</td><td>Reference point return direction (–)</td></tr>
<tr><td>9</td><td>Rotation direction determined by DIR</td><td>Rotation direction in the shortcut direction</td></tr>
<tr><td>A</td><td>Machine basic position becomes the origin point</td><td>Electrical zero point becomes the origin point</td></tr>
<tr><td>D</td><td>Coordinate zero point creation valid</td><td>Zero point established at power supply ON position</td></tr>
<tr><td>E</td><td>Rotation direction in DIR or in the shortcut direction</td><td>Rotation direction is the same as the random position command sign</td></tr>
<tr><td>F</td><td>Stopper direction is positioning direction</td><td>Stopper direction is the same as the stopper amount sign</td></tr>
</table></td></tr>
<tr><td>#110</td><td>ZRNspeed</td><td>Reference point return speed</td><td>1000</td><td>deg/min (mm/min)</td><td>Set the clamp value for the feedrate when a reference point return is carried out. The feedrate becomes the manual operation speed of the parameter group selected at that time, but it is clamped by this parameter setting value.</td><td>1~100000</td></tr>
<tr><td>#111</td><td>ZRNcreep</td><td>Reference point return creep speed</td><td>200</td><td>deg/min (mm/min)</td><td>Set the approach speed to the reference point after dog detection during a reference point return.</td><td>1~65535</td></tr>
</table>

5.2 Absolute Position Detection for Auxiliary Axis

5.2.1 Dog-type Reference Point Return Method

The coordinate zero point is established with the dog-type reference point return operation. The operation method is the same as the dog-type reference point return using the incremental system. Refer to the section "5.1 Dog-type Reference Point Return for Auxiliary Axis".

5.2.2 Machine Stopper Method

Jog feed is carried out with the torque (current) limit set, and the axis is pushed against the machine end, etc., to determine the absolute position origin point.

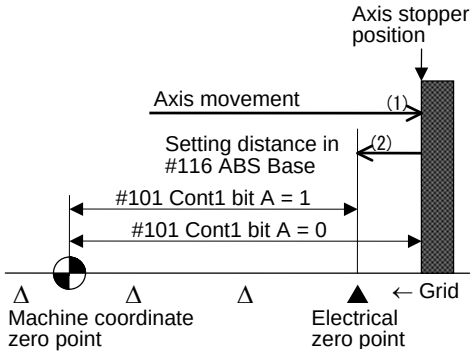
(1) Initialization

The operation parameter group 4 will be automatically selected during the reference point initialization mode. Set the torque limit value (TL4) and excessive error detection width (OD4) to values appropriate for the pushing operation.

Operation procedure	State
1. Select the Auxiliary axis monitor screen.	
2. Press the page key to select the axis to be set and set the axis name.	If the axis that has not been set is selected, error will occur.
3. Press [J] to turn ON the "Ope test mode" signal.	"Ope test mode" signal can be turned ON only when the auxiliary axis is stopped and "Ope test mode valid" signal from PLC is turned ON.
4. Press [Z].	The absolute position initializing state is on.
5. Press [M]. When the "Ope test mode" is INCREMENTAL or HANDLE, move the cursor to "Scale" and set the feedrate per pulse.	Operation mode can be selected from the followings: JOG, INCREMENTAL, HANDLE
6. Press [P].	Speed and torque limit value can be designated.
7. Move the axis to set the marked point. (Press [↑] or [↓] to start.)	

(1) The axis is pushed against the machine end stopper with jog or handle feed. When the torque (current) reaches the limit value due to this pushing, the limiting torque (TLQ) is output, and the position is saved as the "absolute position origin point".

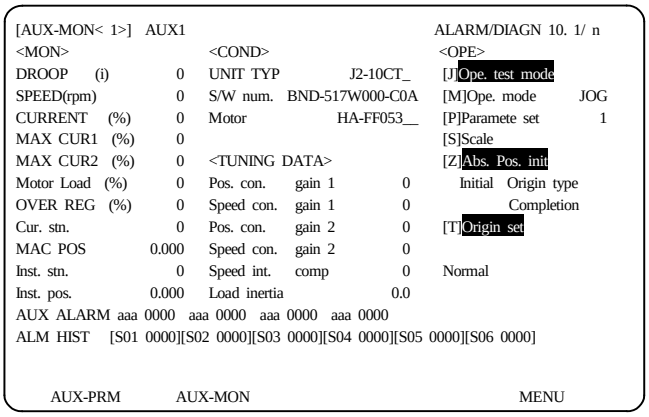
(2) The axis is moved in the direction opposite the pushing direction. When the axis moves and reaches the first grid point, the axis automatically stops, and the absolute position coordinates are established.
If parameter "101 Cont1 bit A" is set to "1", the electrical zero point (grid) will be set as the "absolute position origin point" instead of the pushed position.



(3) In this state, the absolute position origin point will become the coordinate zero point. To set a point other than the push position or electrical zero point as the absolute position coordinate zero point, move the machine coordinate zero point with parameter #116 ABS Base (Absolute position zero point).

8. After initializing, turn on the power again to run the normal operation.	

(2) Supplemental remarks



The adjustment of the auxiliary axes is carried out in the <OPE> area on the AUXILIARY AXIS MONITOR screen.

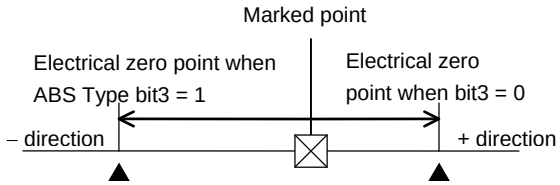
Function	Operation key	Details
Ope. mode	M	This changes the operation mode. The operation mode changes over as follows every time the key is pressed. JOG → INCREMENTAL → MANUAL OPERATION → HANDLE → (dog-type method only) ZERO RTN → JOG The initial display becomes "JOG" when the operation adjustment mode is validated from automatic adjustment.
Paramete set	P	Set the operation parameter group No. The operation parameter group changes as follows every time the key is pressed. 1 → 2 → 3 → 4 → 1
Scale	S	The settings for this function are only validated when the operation mode is the "Incremental" mode or the "Handle" mode. Incremental mode: Set the feed amount per startup. The feed amount changes as follows every time the key is pressed. 1/1° → 1/10° → 1/100° → 1/1000° → 1/1° Handle mode: Set the handle rotation magnification. The handle rotation magnification changes as follows every time the key is pressed. 1 → 10 → 100 → 1000
Abs. Pos. init	Z	The settings for this function are only validated when the zero point is initialized by the "Stopper" method or the "Marked point alignment" method. This function operates when initializing the absolute position. The function is turned ON/OFF every time the key is pressed.
Origin set	T	The settings for this function are only validated when the zero point is initialized by the "Marked point alignment" method. Press this operation key to set the origin point. The function is turned ON/OFF every time the key is pressed.

Function	Operation key	Details
Normal rotation	↑	This starts the rotation in the forward run direction. The rotation will stop if this key is pressed while starting.
Reverse rotation	↓	This starts the rotation in the reverse run direction. The rotation will stop if this key is pressed while starting.
Stop	Press any key other than the SHIFT key. **(Example)** A key	This stops the rotation during forward or reverse run.

<table>
<tr><th>No.</th><th>Abbrev.</th><th>Parameter name</th><th>Default value</th><th>Unit</th><th>Explanation</th><th>Setting range</th></tr>
<tr><td>#101</td><td>∗Cont1</td><td>Control parameter 1</td><td colspan="4">This is a HEX setting parameter. Set bits without a description to their default values.
<table>
<tr><th>bit</th><td>F</td><td>E</td><td>D</td><td>C</td><td>B</td><td>A</td><td>9</td><td>8</td><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td></tr>
<tr><th>Default value</th><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>1</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td></tr>
</table>
<table>
<tr><th>bit</th><th>Meaning when "0" is set.</th><th>Meaning when "1" is set.</th></tr>
<tr><td>1</td><td>High-speed zero point return after zero point establishment</td><td>Dog-type method for each zero point return operation</td></tr>
<tr><td>8</td><td>Reference point return direction (+)</td><td>Reference point return direction (–)</td></tr>
<tr><td>9</td><td>Rotation direction determined by DIR</td><td>Rotation direction in the shortcut direction</td></tr>
<tr><td>A</td><td>Machine basic position becomes the origin point</td><td>Electrical zero point becomes the origin point</td></tr>
<tr><td>D</td><td>Coordinate zero point creation valid</td><td>Zero point established at power supply ON position</td></tr>
<tr><td>E</td><td>Rotation direction in DIR or in the shortcut direction</td><td>Rotation direction is the same as the random position command sign</td></tr>
<tr><td>F</td><td>Stopper direction is positioning direction</td><td>Stopper direction is the same as the stopper amount sign</td></tr>
</table></td></tr>
<tr><td>#178</td><td>TL4</td><td>Operation parameter group 4
Torque limit value</td><td>500</td><td>%</td><td>Set the value 100% or below that may not exceed the torque limit at the accelerated speed when pushing against the machine end.</td><td>1 ~ 500</td></tr>
<tr><td>#179</td><td>OD4</td><td>Operation parameter group 4
Excessive error detection width</td><td>100</td><td>deg (mm)</td><td>Set the value that may not generate the excessive error detection width alarm.</td><td>0 ~ 32767</td></tr>
</table>

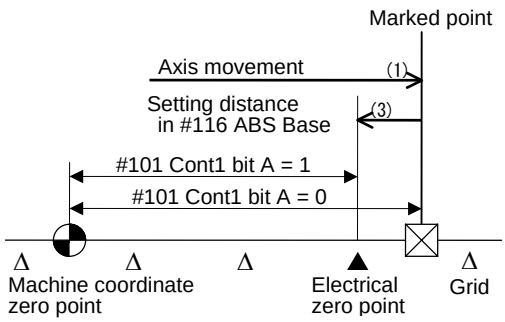
5.2.3 Dogless-type Marked Point Alignment Method

The absolute position origin point is determined by setting the axis to the marked point within the machine's moveable range.

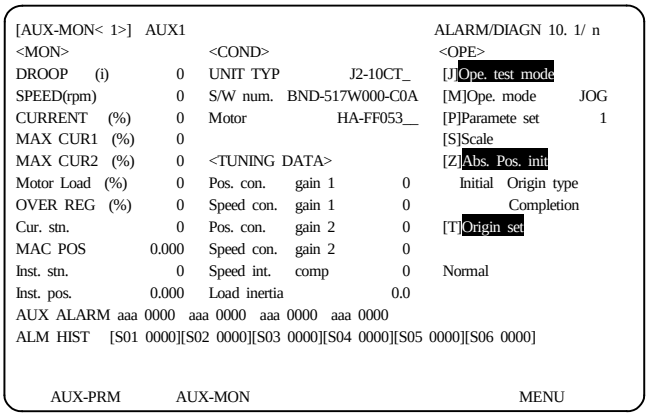


(1) Initialization

Set the direction from the marked point to the grid to be used as the electrical zero point in parameter #120 ABS Type (Absolute position detection) parameter bit3.

Operation procedure	State
1. Select the Auxiliary axis monitor screen.	
2. Press the page key to select the axis to be set and set the axis name.	If the axis that has not been set is selected, error will occur.
3. Press [J] to turn ON the "Ope test mode" signal.	"Ope test mode" signal can be turned ON only when the auxiliary axis is stopped and "Ope test mode valid" signal from PLC is turned ON.
4. Press [Z].	The absolute position initializing state is on.
5. Press [M]. When the "Ope test mode" is INCREMENTAL or HANDLE, move the cursor to "Scale" and set the feedrate per pulse.	Operation mode can be selected from the followings: JOG, INCREMENTAL, HANDLE
6. Press [P].	Speed and torque limit value can be designated.
7. Move the axis to set the marked point. (Press [↑] or [↓] to start.)	
(1) Using jog, handle or incremental feed, set the axis position to the position to become the "absolute position origin point". (2) Turn the zero point setting (ZST) signal ON. (3) Using jog, handle or incremental feed, move the axis in the direction of the grid to be the electrical zero point. When the axis reaches the grid to be the electrical zero point, it will automatically stop, and the absolute position coordinates will be established. If parameter #101 Cont1 bit A is set to "1", the electrical zero point (grid) will be set as the "absolute position origin point" instead of the position where zero point setting was turned ON. (4) In this state, the absolute position origin point will become the coordinate zero point. To set a point other than the position where zero point setting (ZST) was turned ON or the electrical zero point as the absolute position coordinate zero point, move the machine coordinate zero point with parameter #116 ABS Base (Absolute position zero point).	
8. After initializing, turn on the power again to run the normal operation.	

(2) Supplemental remarks



The adjustment of the auxiliary axes is carried out in the <OPE> area on the AUXILIARY AXIS MONITOR screen.

Function	Operation key	Details
Ope. mode	M	This changes the operation mode. The operation mode changes over as follows every time the key is pressed. JOG → INCREMENTAL → MANUAL OPERATION → HANDLE → (dog-type method only) ZERO RTN → JOG The initial display becomes "JOG" when the operation adjustment mode is validated from automatic adjustment.
Paramete set	P	Set the operation parameter group No. The operation parameter group changes as follows every time the key is pressed. 1 → 2 → 3 → 4 → 1
Scale	S	The settings for this function are only validated when the operation mode is the "Incremental" mode or the "Handle" mode. Incremental mode: Set the feed amount per startup. The feed amount changes as follows every time the key is pressed. 1/1° → 1/10° → 1/100° → 1/1000° → 1/1° Handle mode: Set the handle rotation magnification. The handle rotation magnification changes as follows every time the key is pressed. 1 → 10 → 100 → 1000
Abs. Pos. init	Z	The settings for this function are only validated when the zero point is initialized by the "Stopper" method or the "Marked point alignment" method. This function operates when initializing the absolute position. The function is turned ON/OFF every time the key is pressed.
Origin set	T	The settings for this function are only validated when the zero point is initialized by the "Marked point alignment" method. Press this operation key to set the origin point. The function is turned ON/OFF every time the key is pressed.

Function	Operation key	Details
Normal rotation	↑	This starts the rotation in the forward run direction. The rotation will stop if this key is pressed while starting.
Reverse rotation	↓	This starts the rotation in the reverse run direction. The rotation will stop if this key is pressed while starting.
Stop	Press any key other than the SHIFT key. **(Example)** A key	This stops the rotation during forward or reverse run.

No.	Abbrev.	Parameter name	Default value	Unit	Explanation	Setting range
#101	∗Cont1	Control parameter 1	This is a HEX setting parameter. Set bits without a description to their default values.			

bit	F	E	D	C	B	A	9	8	7	6	5	4	3	2	1	0
Default value	0	0	0	0	0	0	1	0	0	0	0	0	0	0	0	0

bit	Meaning when "0" is set.	Meaning when "1" is set.
1	High-speed zero point return after zero point establishment	Dog-type method for each zero point return operation
8	Reference point return direction (+)	Reference point return direction (–)
9	Rotation direction determined by DIR	Rotation direction in the shortcut direction
A	Machine basic position becomes the origin point	Electrical zero point becomes the origin point
D	Coordinate zero point creation valid	Zero point established at power supply ON position
E	Rotation direction in DIR or in the shortcut direction	Rotation direction is the same as the random position command sign
F	Stopper direction is positioning direction	Stopper direction is the same as the stopper amount sign

No.	Abbrev.	Parameter name	Default value	Unit	Explanation	Setting range
#116	ABS base	Absolute position zero point	0.000	deg (mm)	Set the movement amount when the machine coordinate zero point is to be moved from the origin point during absolute position initialization.	–99999.999 ~99999.999
#120	ABS Type	Absolute position detector parameter	This is a HEX setting parameter. Set bits without a description to their default values.			

bit	F	E	D	C	B	A	9	8	7	6	5	4	3	2	1	0
Default value	0	0	0	0	0	0	0	0	0	0	0	0	0	1	0	0

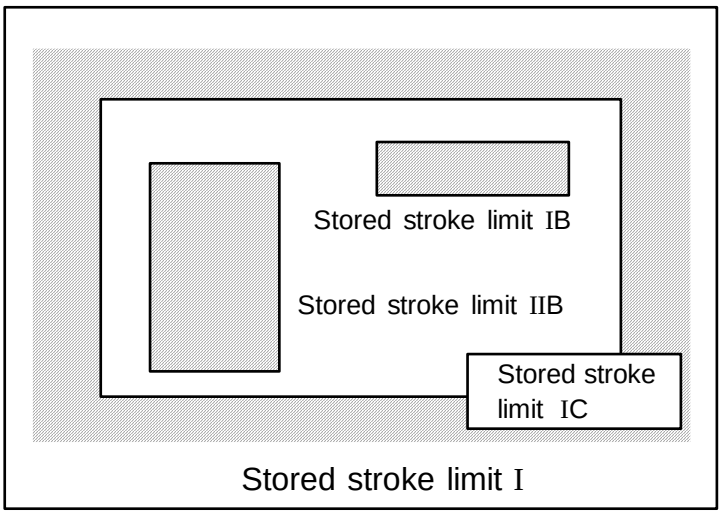
bit	Meaning when "0" is set.	Meaning when "1" is set.
1	Dog-less type initialization	Dog-type initialization
2	Machine stopper method initialization	Marked point alignment initialization
3	Electrical zero point direction +	Electrical zero point direction –

6. Stored Stroke Limit

(1) Outline

Three tool entry prohibited ranges can be set with stored stroke limit I, stored stroke limit II or IIB and stored stroke limit IB. Part of the prohibited range on the outside of soft limit I can be validated with stored stroke limit IC.
As for the stored stroke limit II and IIB, the prohibited range is selected by the parameter.

II :Prohibited area is outside
IIB :Prohibited area is inside



If the tool tries to move over the set range, an alarm will appear, and the axis will decelerate to a stop.
If the prohibited range is entered and an alarm occurs, movement will be possible only in the direction opposite the entry direction.
This function is an option.

[Valid Conditions of Stored Stroke Limit]
When using the relative position detection system, the stored stroke limit is invalid until the reference point return is completed after the power is turned ON.
The stored stroke limit can be validated even if the reference point return is not yet completed, by setting parameter #2049 type = 9.

(Note) If the absolute position detection is valid when using the absolute position detection system, the stored stroke limit will be validated immediately after the power is turned ON.

[Stored stroke limit coordinates]
The stored stroke limit check is carried out in the basic machine coordinate system established by the reference point return. To validate the stored stroke limit even when the reference point return is not yet completed, check the stored stroke limit using the temporary basic machine coordinate system (basic coordinate system defined when the power was previously turned OFF).
When the 1st dog-type reference point return is completed after the power is turned OFF, the correct coordinate system is established.

CAUTION
Always set the stored stroke limit. Failure to set this could result in collision with the machine end.

(Note) The axis movement possible when the reference point return has not yet completed is limited to manual and handle feed only. Automatic operation is validated after the reference point return is completed.

(2) Detailed explanation

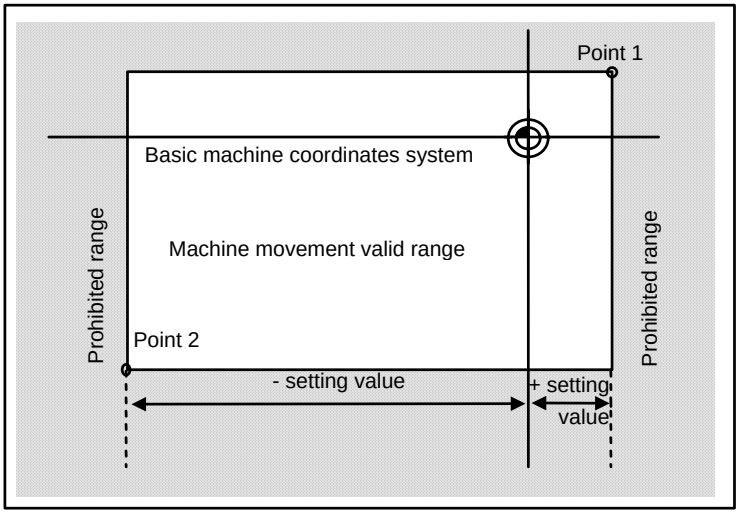
The stored stroke limit sets a prohibited range with the parameters or program command. The minimum and maximum values of the prohibited range are set as the coordinate value (radius value) on the machine coordinate system for each axis. The stroke is not checked for axes set to the same value. This function is valid only for the axis returned to the reference point after the power was turned ON.
Before the machine enters the prohibited range, an error "M01 Operation error 0007" (S/W stroke end) will occur, and the machine movement will stop. The alarm can be reset by moving the erroneous axis in the opposite direction.
During automatic operation, if an alarm occurs with even one axis, all axes will decelerate to a stop. During manual operation, only the axis that caused the alarm will decelerate to a stop. The axis will always stop at a position before the prohibited range. The distance between the prohibited range and stop position will depend on the feedrate, etc.

The stored stroke limits I, II, IIB, IB and IC are handled as follows.

Type	Prohibited range	Explanation		Range setting parameter	Validating conditions
I	Outside	• Set by the machine maker. • When used with II, the confined range designated by the two functions becomes the movement valid range. • Can be rewritten with DDB.		#2013 OT– #2014 OT+	• Reference point return is completed. • #2013 and #2014 are not set to the same value.
II	Outside	• Set by the user. • Select II or IIB with the parameters. • Can be rewritten with DDB.	• #8210 OT-INSIDE:0 • Used with I.	#8204 OT-CHECK-N #8205 OT-CHECK-P	• Reference point return is completed. • #8204 and #8205 are not set to the same value. • #8202 OT-CHECK OFF:0
IIB	Inside		• #8210 OT-INSIDE:1		
IB	Inside	• Set by the machine maker.		#2061 OT_1B– #2062 OT_1B+	• Reference point return is completed. • #2061 and #2062 are not set to the same value.
IC	Outside	• Set by the machine maker. • Can be rewritten with DDB.		#2061 OT_1B– #2062 OT_1B+	• #2061 and #2062 are not set to the same value. • #2063 OT_1B type:2

6.1 Stored Stroke Limit I

This is a stroke limit function used by the machine maker. The boundary is set with the parameters (axis specification parameters "#2013 OT–" and "#2014 OT+"). The outside of the set boundary is the prohibited range.
When used with the stored stroke limit II function, the confined range designated by the two functions becomes the movement valid range.



The following values are set with the coordinate values in the basic machine coordinate system.
Point 1: #2014 OT+
Point 2: #2013 OT–

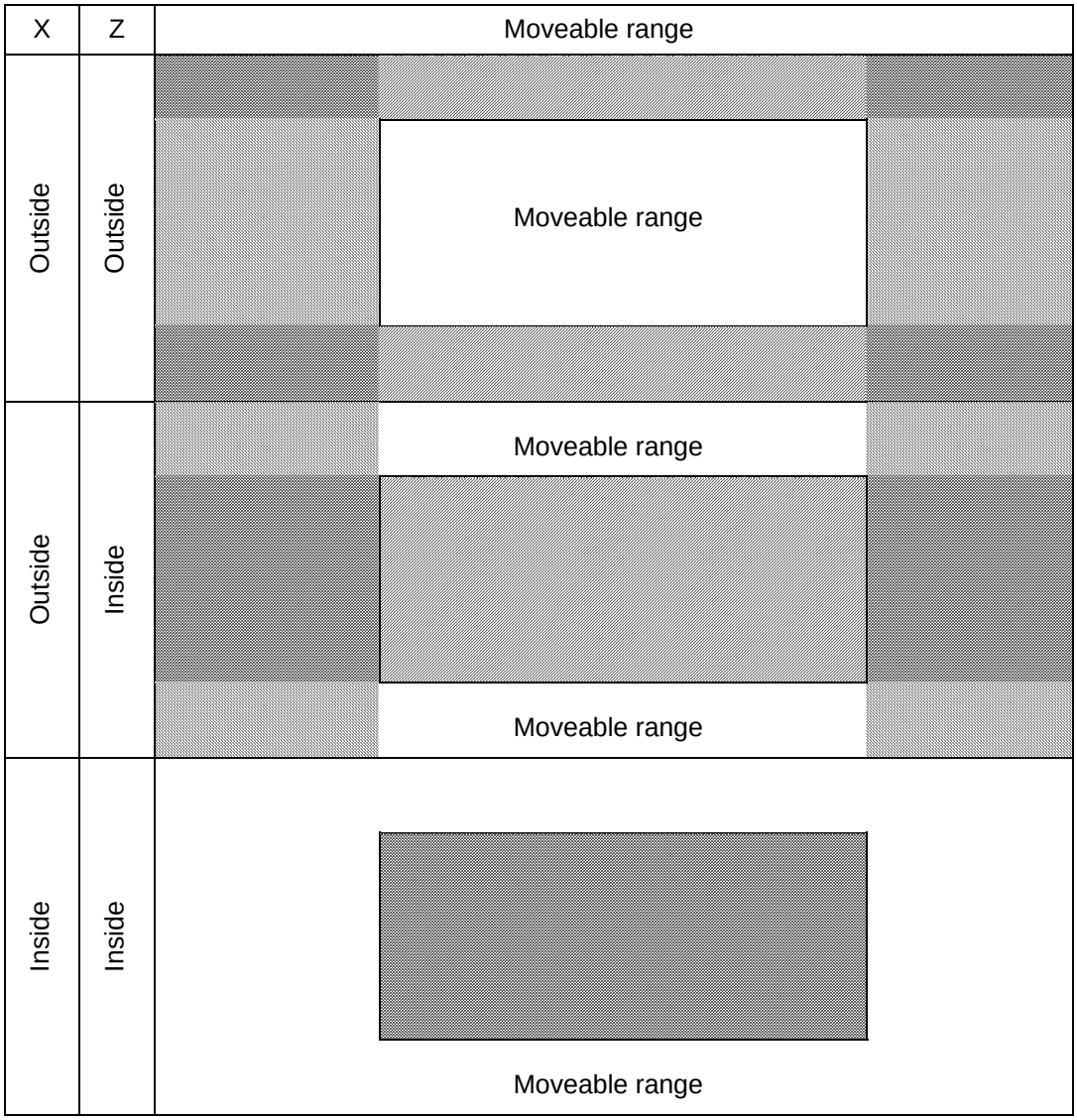
(Note1) This function will be invalid if the same value excluding "0" is set to the parameters "#2013 OT-" and "#2014 OT+".

(Note2) When using indexing axis for C64T, inside of the specified range will be prohibited if the "#2013 OT-" value is larger than that of "#2014 OT+".

6.2 Stored Stroke Limit II

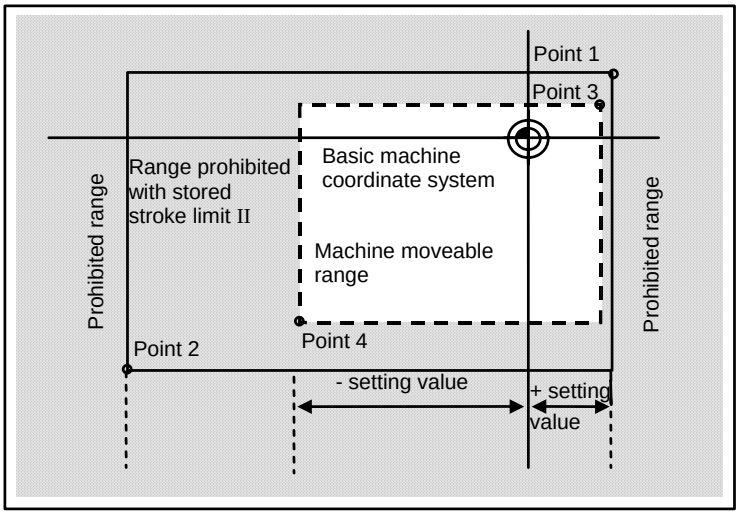
The boundary is set with the parameters (axis parameters #8204 OT-CHECK-N,#8205 OT-CHECK-P) or with the program commands. Either the inside or the outside of the set boundary is the prohibited range. Whether the inside or outside of the range is prohibited is determined by parameter (#8210 OT-INSIDE). When the inside is selected, this function is called stored stroke limit IIB.
When using program commands, entry of the tool into the prohibited range is prohibited with G22, and entry into the prohibited range is enabled with G23.
The stored stroke limit II function can be invalidated for each axis with the parameter setting (#8202 OT-CHECK OFF:1).

"#8210 OT-INSIDE" can be set for each axis, but when used in combination, the following type of operation is possible.



(1) Stored stroke limit II (When prohibited range is on outside)

When used with the stored stroke limit I function, the confined range designated by the two functions becomes the moveable range.

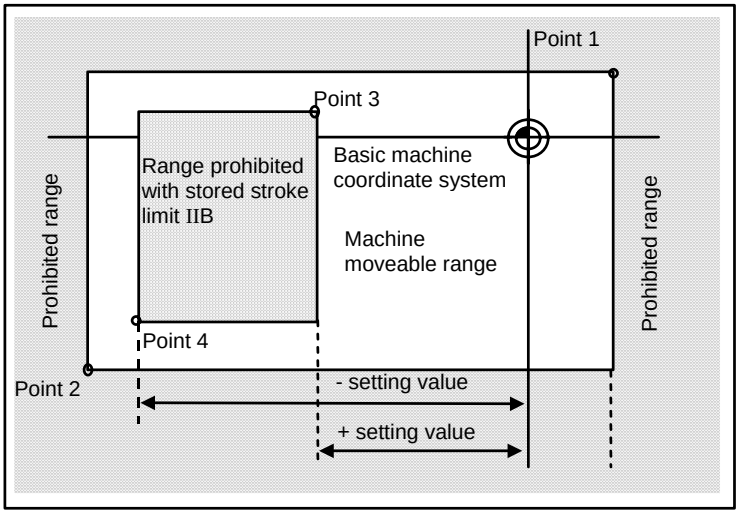


The following values are set with the coordinate values in the basic machine coordinate system.
Point 3 and 4:
#8205 OT-CHECK-P
#8204 OT-CHECK-N

The area determined by points 1 and 2 is the prohibited range set with stored stroke limit I.

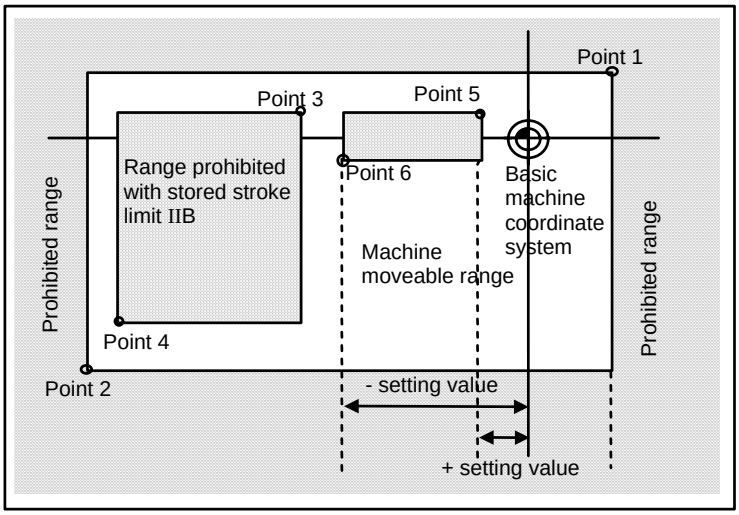
(2) Stored stroke limit IIB (When prohibited range is on inside)

A range except for that of the stored stroke limit I becomes the movement prohibited range.



6.3 Stored Stroke Limit IB

The boundary is set for each axis with the parameters (axis parameters "#2061 OT_1B-" and "#2062 OT_1B+"). The inside of the set boundary is the prohibited range.

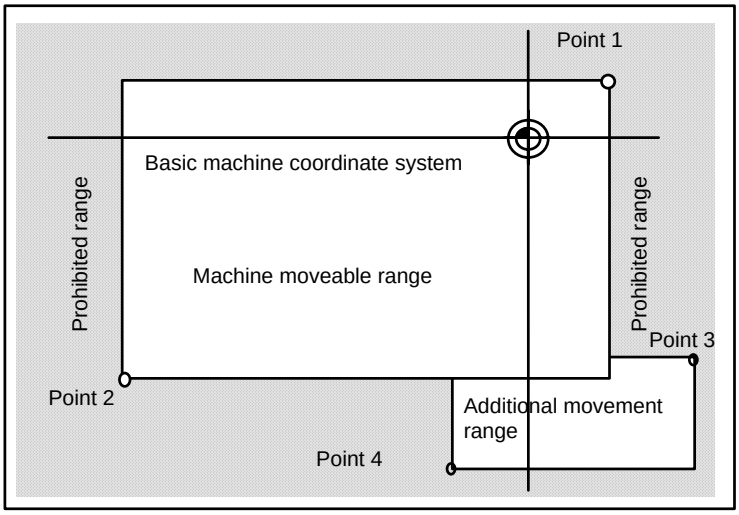


The following values are set with the coordinate values in the basic machine coordinate system.
Point 3 : #2062 OT_1B+
Point 4 : #2061 OT_1B–

The area determined by points 1 and 2 is the prohibited range set with stored stroke limit I, and the area determined by points 3 and 4 is the prohibited range set with stored stroke limit II.

6.4 Stored Stroke Limit IC

The boundary is set for each axis with the parameters (axis parameters "#2061 OT_1B-" and "#2062 OT_1B+"). The inside of the set boundary is the machine moveable range.
This is valid when the axis parameter #2063 is set to 2, and cannot be used with soft limit IB.



The following values are set with the coordinate values in the basic machine coordinate system.
Point 3 : #2062 OT_1B+
Point 4 : #2061 OT_1B–

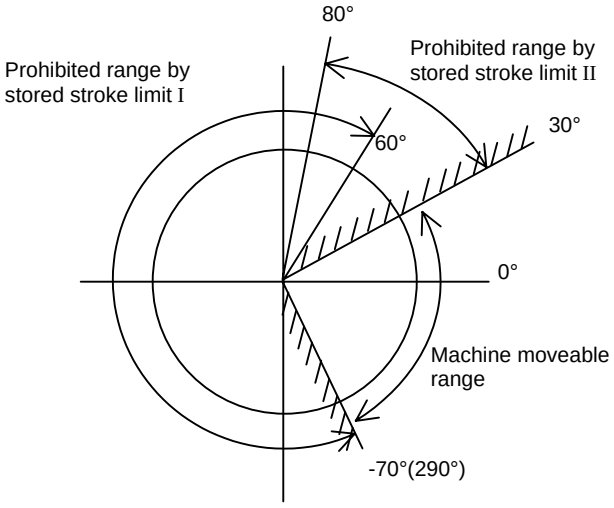
The area determined by points 1 and 2 is the prohibited range set with stored stroke limit I.

6.5 Stored Stroke Limit for Rotation axis

Use the stored stroke limit I and II for rotary axes. The area regarded as the prohibited range must be located between the maximum value and the minimum value, moreover, excluding the zero point of the basic machine coordinate system.
As for the rotary axis, even if the "maximum" parameter value is smaller than the minimum value, the stored stroke limit works the same as the maximum is larger.

(ex.)
Stored stroke limit I (When maximum value and minimum value of the parameters determining the prohibited range are such as;)
#2013 OT- : -70.000°
#2014 OT+ : 60.000°

Stored stroke limit II (When maximum value and minimum value of the parameters determining the prohibited range are such as;)
#8204 OT-CHECK-N : 30.000°
#8205 OT-CHECK-P : 80.000°



(Note) Make sure not to use the stored stroke limit IB, IIB, and IC. Make these parameters invalid by setting as follows.
#8210 OT-INSID : Set "0" (II:valid and IIB:invalid)
#2061 and #2062 : Set same value excluding "0" (IB and IC :invalid)

6.6 Cautions

(1) When the maximum value and minimum value of the stored stroke limit's prohibited range are set to the same value, note that the following will occur.
   (a) When the maximum value and minimum value are set to 0, if the outside is the prohibited range, the entire range will be prohibited. If the inside is the prohibited range, the entire range will be the moveable range.
   (b) If data other than 0 is set for the maximum value and minimum value, the entire range will be the moveable range.

(2) When using indexing axis for C64T, inside of the specified range will be prohibited if the "#2013 OT-" value is larger than that of "#2014 OT+".

(3) The stored stroke limit IC is valid when the axis parameter #2063 is changed.
If changed during automatic operation, the function will be validated after the smoothing for all axes reaches 0.

(4) Make sure that the lower limit value of the stored stroke limit IC setting value is smaller than the upper limit value.

(5) The axis parameters #2061, #2062 and #2063 can be changed with DDB.

7. Daily Maintenance and Periodic Inspection

7.1 Daily Inspection

7.1.1 Checking the External View

(1) Machine oil (cutting oil, lubrication oil) have been scattered onto the servomotor, detector, or control unit, or is leaking.
(2) Damage is found on the cables of the movable blocks, or the cables are twisted.
(3) Filter clogging
(4) A door of the control panel is not open.
(5) Ambient vibration
(6) The unit is located in a dusty location.
(7) Something that causes high frequency is placed near the NC unit.

7.1.2 Checking the Inside of the Control Panel

(1) Cable connectors are loosened.
(2) Installing screws are loosened.
(3) Attachment amplifier screws are loosened.
(4) The cooling fan operates abnormally.
(5) Cable damage
(6) PCB have been inserted abnormally.

7.2 Maintenance Tool

(1) Measuring instruments

The following measuring instruments are used to confirm that the voltage is being supplied correctly to the NC unit, to confirm that the wiring to the NC unit is correct, and to carry out simple troubleshooting.

Tool	Condition	Application
Tester		To check that the wiring to the NC unit is correct before turning the power ON.
AC voltmeter	Measure the AC power voltage. The tolerable error is ±2% or less.	To measure the AC power voltage being supplied to the external 24VDC power supply unit.
DC voltmeter	Max. scale 30V. The tolerable error is ±2% or less.	To measure the DC power voltage. External power supply 24V (control section, machine input/output interface) Battery voltage HR851 SA output
Synchroscope		General measurement and simple troubleshooting

(Note 1) Currently, a high precision digital multi-meter is commonly used as a tester. This digital multi-meter can be used as both an AC voltmeter and a DC voltmeter.

(2) Tools

Screwdriver (large, medium, small)
Radio pliers

7.3 Maintenance Items

Maintenance is categorized into daily maintenance items (items to be carried at set intervals) and periodic maintenance (replacement of parts when life is reached).
Some parts will not function in a hardware manner when the life is reached, so these should be replaced before the life is reached.

Class	Name	Life	Inspection/replacement	Remarks
Daily maintenance	Escutcheon (when using communication terminal)		Once/two months (Accordingly when dirty)	Refer to section 7.3.1.
Periodic maintenance	Battery (lithium battery)	Cumulative data holding time 45,000h	When battery drop caution alarm occurs (Guideline: approx. 5 years)	Refer to "Connection and Maintenance Manual."
	Cooling fan (control section)	30,000h	Refer to left.	Refer to "Connection and Maintenance Manual."
	LCD display unit (when using communication terminal)	10,000h (Specified by the power ON time that the brightness drops to less than 50%.)	Replace when backlight darkens.	Refer to section 7.3.2.

7.3.1 Escutcheon

(1) Cleaning the escutcheon

(a) Keep the rear side of the escutcheon as clean as possible.
(b) Wipe the escutcheon with a soft, clean, dry cloth. If cleaning is still required, put some neutral detergent on a cloth and wipe. Do not use alcohol, thinner, etc.

7.3.2 LCD Panel

(1) Handling the LCD panel

[Precautions for use]

(a) The polarizing plate (display surface) of the LCD panel surface can be easily scratched, so be careful during handling.
(b) Glass is used in the LCD panel. Be careful not to drop the LCD panel or allow it to hit hard objects, as the glass may chip or break.
(c) The polarizing plate may be stained or discolored if drops of water, etc., adhere to it for long periods, so be sure to wipe off any moisture immediately.
(d) Wipe off any dirt, dust, etc., on the polarizing plate using absorbent cotton or other soft cloth.
(e) A CMOS LSI is used in the LCD panel, so be careful of static electricity when handling.
(f) Never disassemble the LCD panel. Doing so will damage the panel.

[Precautions for storage]

(a) Do not store the LCD panel in locations having a high temperature or humidity. (Store within the storage temperature range.)
(b) When storing the LCD panel as an individual unit, be sure that other objects do not touch or hit the polarizing plate (display surface).
(c) When storing the LCD panel for extended periods, be sure to store in a dark place away from exposure to direct sunlight or fluorescent light.

(2) Other precautions for use

(a) Backlight life

The life of the backlight is 25,000 hours/25°C. (Time for luminance to drop to 50% of the initial value.)

The backlight life is dependent on the temperature. The life tends to be shorter when used continuously at lower temperatures.

(b) Luminance start

Due to the characteristics of the backlight, the luminance could drop slightly at lower temperatures. It will take approx.10 to 15 minutes for the luminance to reach the rated value after the power is turned ON.

(c) Unevenness, luminescent spots and irregularities

Uneven brightness, small luminescent spots or small dark spots may appear on LCD, but this is not a fault.

(d) Contrast

The contrast of STN method LCD panels changes with temperature fluctuation. If this happens and the panel is difficult to see, open the operation box door and adjust the contrast with the contrast adjustment potentiometer on the LCD signal interface PCB.

When using the 10.4 LCD, the brightness can be adjusted with the parameter settings.

7.3.3 IC Card

(1) Handling the IC card

The general handling methods for the IC card are described below.

Refer to the instruction manual of the IC card used for details.

[Precautions for use]

(a) Insert the card in the correct direction.
(b) Do not touch the connector area with the hands or metal.
(c) Do not apply excessive force to the connector area.
(d) Do not subject the card to bending or strong impacts.
(e) Do not open the cover or disassemble the card.
(f) Do not use the card in dusty locations.

[Precautions for storage]

(a) Do not store the card in locations having a high temperature or humidity.
(b) Do not store the card in dusty locations.

8. Fault Diagnosis and Action

8.1 Checking the Fault Occurrence Status

Check the following:

- When did the fault occur?
- During what operation did the fault occur?
- What fault occurred?

(1) When did the fault occur?

Time of day when the fault occurred.

(2) During what operation did the fault occur?

What NC operation mode?

- For automatic operation..... Program number, sequence number, and contents of program
- For manual operation ..... Mode?
  Operating procedure?
  Preceding and succeeding operations?
- Setting display unit's screen?
- During I/O operation?
- Machine system status?
- During tool change?
- Controlled axis hunting?

(3) What fault occurred?

- What does the alarm display of the setting display unit's alarm diagnosis screen indicate? Display the alarm diagnosis screen to check the contents of alarm.
- What does the machine sequence alarm indicate?
- Is the screen normal?

(4) Frequency of fault?

- When did the fault occur? Frequency? (Did the fault occur during operation of another machine?) If the frequency is too small, or the fault occurred during operation of another machine, the cause may be noises of the supply voltage, for example. In this case, check that (i) the supply voltage is normal (does momentary drop occur during operation of another machine?) and (ii) measures have been taken against noises.
- In specific mode?
- When the ceiling crane moved?
- Frequency for the same kind of workpiece?
- Does the fault occur when the same operation is made?
  (Repeatability check)
- Change the conditions (override, contents of program, operating procedure, etc.). Does the same fault occur?
- What was the ambient temperature?
  (Was there a sudden temperature change? Was the fan on the top of the control unit working?)
- Was there a contact defect or insulation defect on the cable?
  (Was there any oil or cutting water splashed on the cable?)

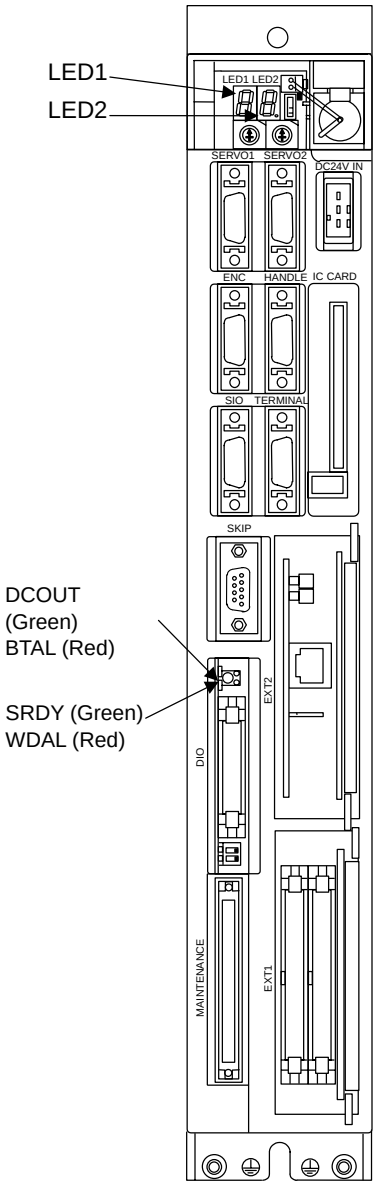
8.2 Fault Examples

When there is a problem or dissatisfaction with the system operation, check the following items before contacting the service center.

– Fault examples –

- The power cannot be turned ON.
- The power fails after being turned ON.
- The screen does not display.
- The operation keys do not work.
- Machining operations cannot be carried out.

List of Unit LEDs



[Explanation of LED functions]

Name	Function	Color	Status		Remedy for error
			When normal	During error	
LED1 LED2	7-segment system status display	Red	Depends on system status		Refer to (2).
DCOUT	During internal power output	Green	Lit	Not lit	Refer to (1).
BTAL	Battery drop warning	Red	Not lit	Lit	Refer to (1).
SRDY	Servo READY (SA)	Green	Lit	Not lit	Refer to (2).
WDAL	Watch dog error	Red	Not lit	Lit	Refer to (2).

(1) Problems related to the power supply

The power does not turn ON.

Cause	Remedy
The door interlock is applied.	If the control panel door is not completely closed, close it. If the door interlock is applied even when the door is closed, the door interlock circuit is damaged.
The external power supply's input voltage is not as specified.	Check that the input voltage is within 200 to 230VAC +10 to –15%.
The external power supply is faulty.	Check that the power can be turned ON with the external power supply only. **Note)** Depending on the external power supply, the power may not turn ON in the no-load state, so install a slight load and check.

The external power turns ON but the NC control power does not turn ON.

Cause	Remedy
The external power supply output is not correct.	Disconnect the cable between the NC unit and the external power supply, and check that the external power supply output is normal.
The power cable is disconnected or broken.	Check the cable connected between the NC unit and external power supply, and securely insert it. Check that the cable is not broken, and replace if broken.
The cable connected from the NC unit to the peripheral device is short-circuited.	Disconnect the cable connected to the peripheral device one at a time and check that the power turns ON. Check that there are no short-circuited cables.
There is a short circuit in the configuration card.	Remove the removable cards one at a time and check that the power turns ON. Check that there are no short-circuited cards.

The power turns OFF.

Cause	Remedy
There is a problem in the power supply.	Check whether the voltage fluctuates at certain time zones. Check whether an instantaneous power failure has occurred.
A problem occurs when the peripheral device starts operating.	Check whether the voltage drops instantaneously when the peripheral device operation starts.

The BTAL (red) LED on the control unit lights.

Cause	Remedy
This lights when the voltage of the battery connected to the BAT connector drops to 2.6V or less.	Replace the battery.

⚠ CAUTION

- ⚠ Do not apply voltages other than those indicated in the Connection and Maintenance Manual on the connector. 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 the PCBs while the power is ON.

(2) Problems when starting the system

The NC does not start up correctly.

Cause	Remedy
8 is displayed on the control unit 7-segment display LED1 (left side).	Check that the rotary switch CS1 (left side) is set to 0.
E or F is displayed on the control unit 7-segment display LED1 (left side).	Contact the Mitsubishi Service Center.
The WDAL (red) LED on the control unit lights.	Contact the Mitsubishi Service Center.

Servo READY (SA) does not turn ON.

Cause	Remedy
The SRDY (green) LED on the control unit is not lit.	Check the emergency stop conditions.
The SRDY (green) LED on the control unit is lit.	Check that the wiring past the DIO connector is correct. (Check the relay coil's diode connection, etc.) If there is no output even when the wiring is corrected, the control unit may be faulty. Check the voltage with a tester. (The voltage is correct if it is approx. 24V across the relay coil end or connector DIO's 1B pin (+) and 1A pin (–) when the LED is lit.)

⚠ CAUTION

- Do not apply voltages other than those indicated in the Connection and Maintenance Manual on the connector. 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 the PCBs while the power is ON.

(3) Problems related to remote I/O

The communication alarm LED ALM (red) lights.

Cause	Remedy
The remote connection cable is not connected. The cable is disconnected or has a connector contact fault.	Check the connection of the R211 cable between the NC control section and remote I/O unit.
The remote I/O unit is faulty.	Contact the Mitsubishi Service Center, and then replace the card.

The power system LED POWER (green) turns OFF.

Cause	Remedy
The input power is not being supplied.	Supply a +24V±5% voltage to the Remote I/O unit.
(The input power is not within the tolerable range, or the internal power is faulty.)	Check that the input voltage is not +20V or less. If it is being supplied correctly, contact the Mitsubishi Service Center.

(4) Problems related to Ethernet

A communication error message is appeared on the display, and the communication LEDs RX (green) and TX (green) turn OFF.
In other cases, a communication error message is appeared on the display, and after the OK button is pressed, the communication error message appears again and the communication LEDs RX (green) and TX (green) turn OFF.

Cause	Remedy
The Ethernet cable is not connected, is broken, or there is a contact fault at the connector.	Check the cable connected between the NC control unit and display.
A straight type Ethernet cable is in use.	Use a cross type Ethernet cable.
The Ethernet unit is not inserted to the very back.	Insert the Ethernet unit so that the plate is flush with the front panel.
The Ethernet unit is faulty.	Contact the Mitsubishi Service Center and replace the unit.

⚠ CAUTION

- Do not apply voltages other than those indicated in the Connection and Maintenance Manual on the connector. 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 the PCBs while the power is ON.

(5) Others

Check the following.

- Is the mode selected correctly?
- Are the starting conditions satisfied?
  (Depending on the machine, the start may be locked until the specified conditions are satisfied. Check with the instruction manual issued by the machine manufacturer.)
- Are the override and manual speed set to "0"?
- Has a reset signal been issued?
- Has a feed hold signal been issued?
- Is the machine lock ON?

9. Maintenance Functions

The machining programs, parameters and tool data, etc., are saved in the memory. However, these contents could be lost due to the battery life, etc. To avoid total loss, save the machining programs, tool data and parameters in an input/output device.

9.1 Data Input/Output Function

The data can be collectively input/output or collated in the data input/output screen or ATA card in/out screen. This function can be used to protect data and investigate failure causes at the customer premises. It can handle three types of data shown below. Hex data is converted and output in the ISO/EIA format. Individual data items can also be input/output or collated separately. This function is valid only while the user PLC is halted.

(1) Table data: The data area is reserved in advance.
- Parameter (system, common, axis, machine error)
- PLC data (PLC timer, PLC counter, PLC constants, bit selection, ATC data, axis control)
- Workpiece offset data
- Backup data (PLC latch relay, PLC switch, programs)

(2) File data: The file format is required before data input.
- Tool offset data
- Common variables
- Tool life management data

(3) PLC program data: PLC program area data
- Ladder (ROM: Ladder, message)
- APLC data

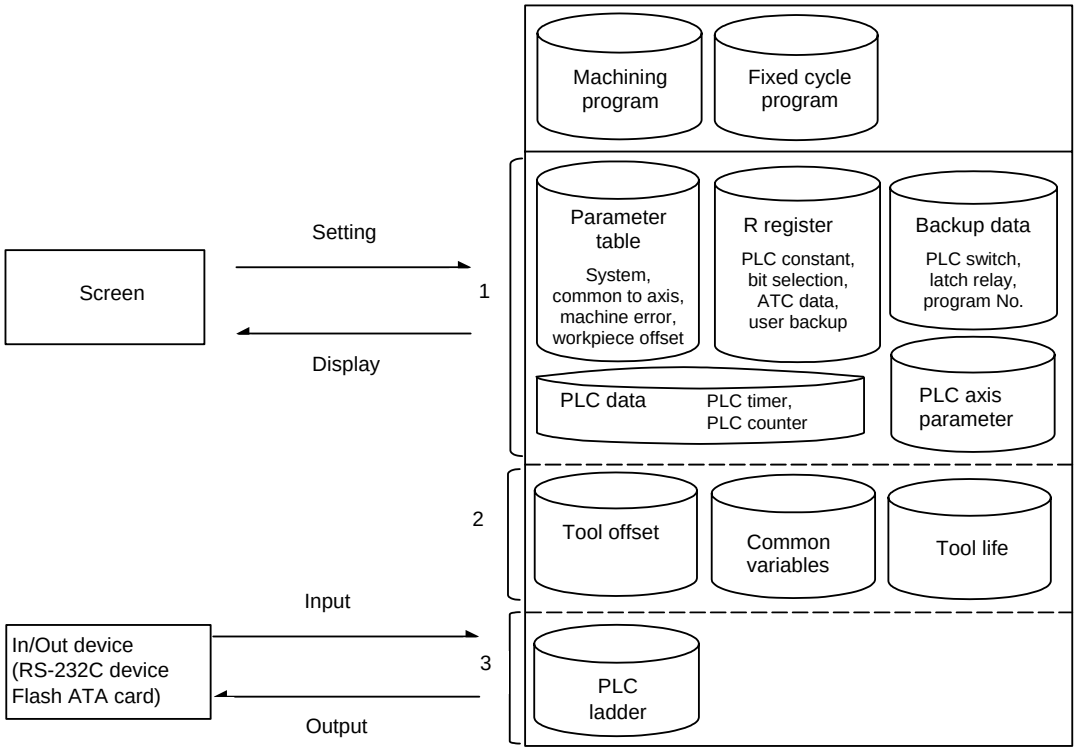


Table data 1 (No.100~149)

No.	Data details	ALL output
100	System common parameters	ALL1
102	Axis independent parameters (including PLC axis)	
103	Machine error offset data, offset amount	
105	PLC constants, work counter (R4500 to R4599)	
106	PLC timer	
107	PLC counter	
108	Bit selection parameter (R4600 to R4647)	
109	(Not used)	
110	Workpiece offset	
111	Tool registration (R4700 to R5389)	
112	R register user backup (R6400 to R7199)	
113	PLC latch relay	
114	PLC switch	
115	Program No.	
116	Timer cumulative output	
117	Counter cumulative output	
130	Tool life management I (R4700 to R5339)	
133	Position switch	
134	PLC timer cumulative current value	
135	PLC counter cumulative current value	
136	R register system backup (R60 to R79)	
138	Tool life management II (R5520 to R6319)	
140	Spindle parameter	
141	Communication parameters	
142	Common variable name	

Table data 2 (No.150~199)

No.	Data details	ALL output
150	Data history (operation history)	—
152	Special process backup	—

Table data 3 (No.200~231)

No.	Data details		ALL output
	M system	L system	ALL2
200	Tool length offset (shape)	Tool offset (X shape)	
201	Tool length offset (wear)	Tool offset (X wear)	
202	Tool radius offset (shape)	Tool offset (Z shape)	
203	Tool radius offset (wear)	Tool offset (Z wear)	
204	—	Tool offset (Y shape)	
205	Tool life	Tool offset (Y wear)	
206	—	Nose R (shape)	
207	—	Nose R (wear)	
208	—	Nose point No. data	
209	Common variables for all part systems (#500 to)		
210	Common variable valid flags for all part systems (#500 to)		
230	Common variables independent for part systems (#100 to)		
231	Common variable valid flags independent for part systems (#100 to)		

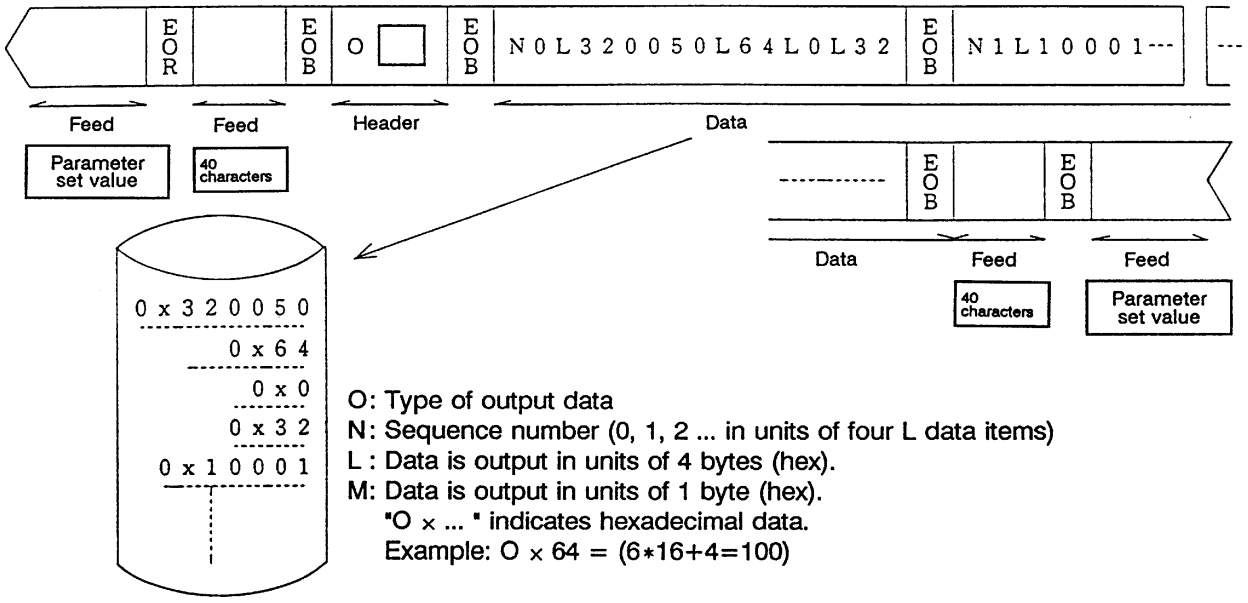
PLC program (No.250~299)

No.	Data details	ALL output
250	Ladder	ALL3
251	APLC software module	-
252	APLC data stored in the maintenance memory	ALL3

9.2 Data Input/Output by RS-232C

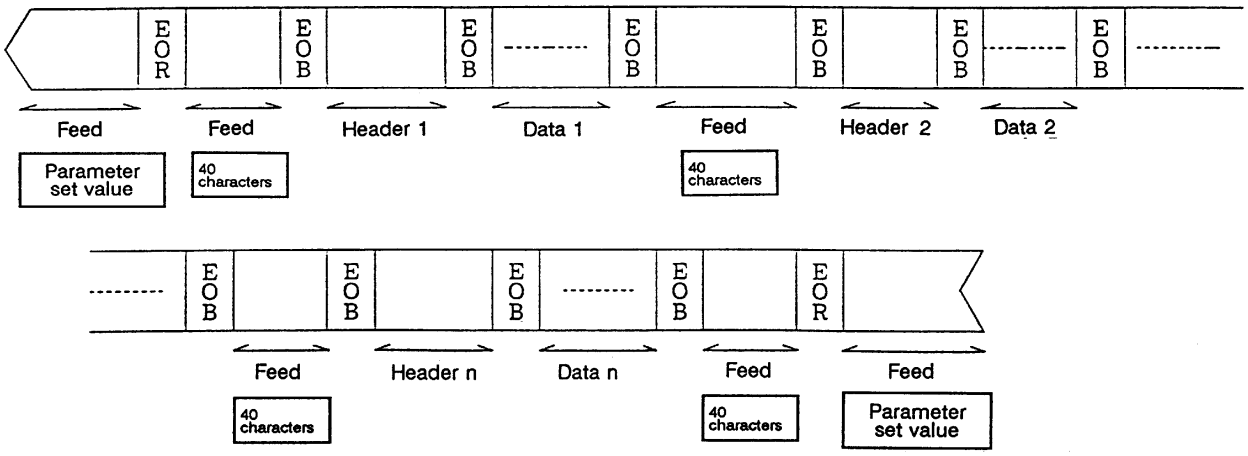
9.2.1 Data Format

The tape output format of single data is as follows:



O: Type of output data
N: Sequence number (0, 1, 2 ... in units of four L data items)
L : Data is output in units of 4 bytes (hex).
M: Data is output in units of 1 byte (hex).
"O × ... " indicates hexadecimal data.
Example: O × 64 = (6*16+4=100)

The output format of continuous data (batch output data) is as follows:



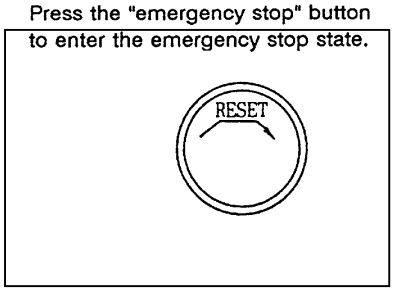
9.2.2 Data Output

(1) Explanation of function

When the user PLC is stopped, the various data can be converted from HEX to ISO/EIA and output to the external RS-232C device on the output screen.

User PLC stopping operation

This function can be used only when the user PLC is stopped.
To stop the user PLC, enter the emergency stop state by pressing the emergency stop button, and set the rotary switch NCSYS to "1".



Data group output operation

Setting area operation	Output data (No.)	
# (99) DATA (ALL1)	Plain data	100 to 149
# (99) DATA (ALL2)	File data	200 to 231
# (99) DATA (ALL3)	PLC program	250, 252 to 299

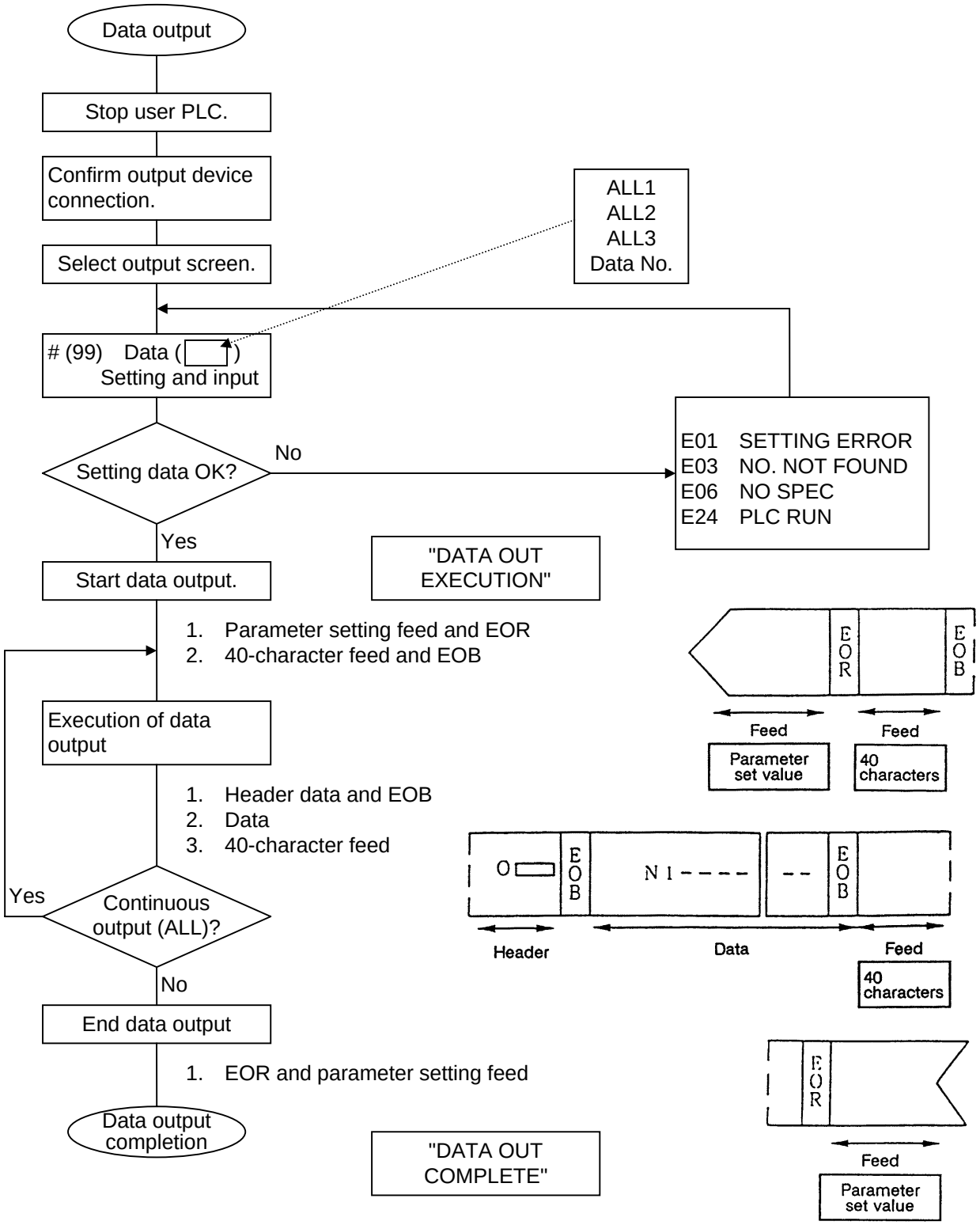
Single data output function

Setting area operation	Output data (No.)	
# (99) DATA ( )	Single data	100 to 149, 150 to 199, 200 to 231, 250 to 299

(Note 1) If a data No. that is not in the specifications is set, the error message "E06 NO SPEC" will display, and the data will not be output.
When group output is executed, the data not in the specifications will not be output.

(Note 2) Refer to "9.1 Data Input/Output Function" for the data type.

(2) Data output operation procedure



(3) Example of data output operation

Confirm that the user PLC is stopped.

Connect the output device.

Call out the data output screen.

DIAGN IN/OUT → MENU → OUTPUT

1) The data output screen will display.

Group output of data

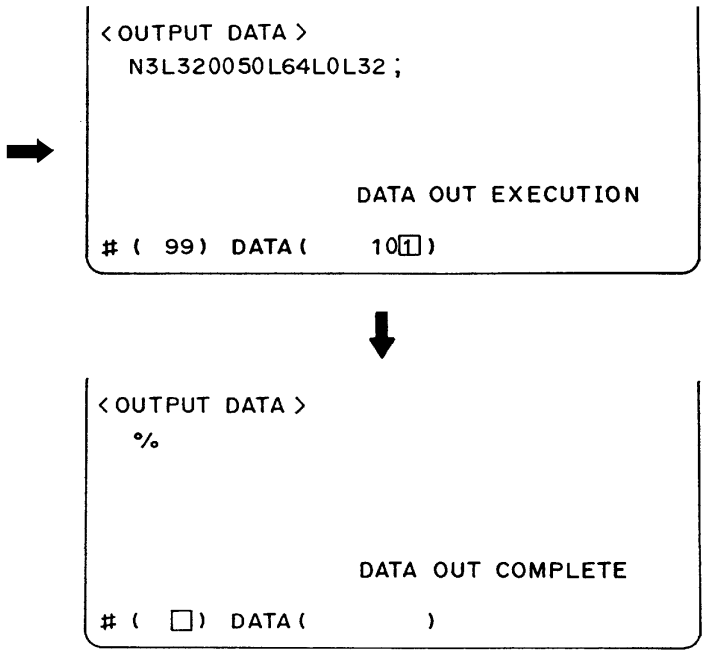
(ex.) ALL1: Plain data (Parameter, R register, workpiece offset, backup data.)

Set 99 in 9 9 setting area, and A L L 1 in the data setting area.

(99) DATA (ALL1)

Press INPUT .

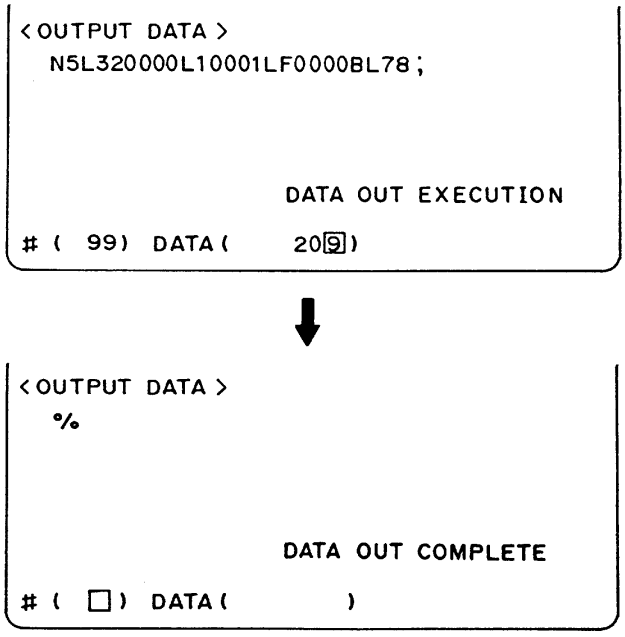
1) The output will start, and the output data No., output data details, and "DATA OUT EXECUTION" message will display.
2) The output operation will end when the end code % (EOR) is output, and the message "DATA OUT COMPLETE" displays.



Output of independent data

Set [9][9] in the # setting area, and the No. of the data to be output in the data setting area.
(Ex. Common variable)
(99) DATA (209)

Press [INPUT] .



1) The output will start, and the output data No., output data details, and "DATA OUT EXECUTION" message will display.
2) The output operation will end when the end code % (EOR) is output, and the message "DATA OUT COMPLETE" displays.

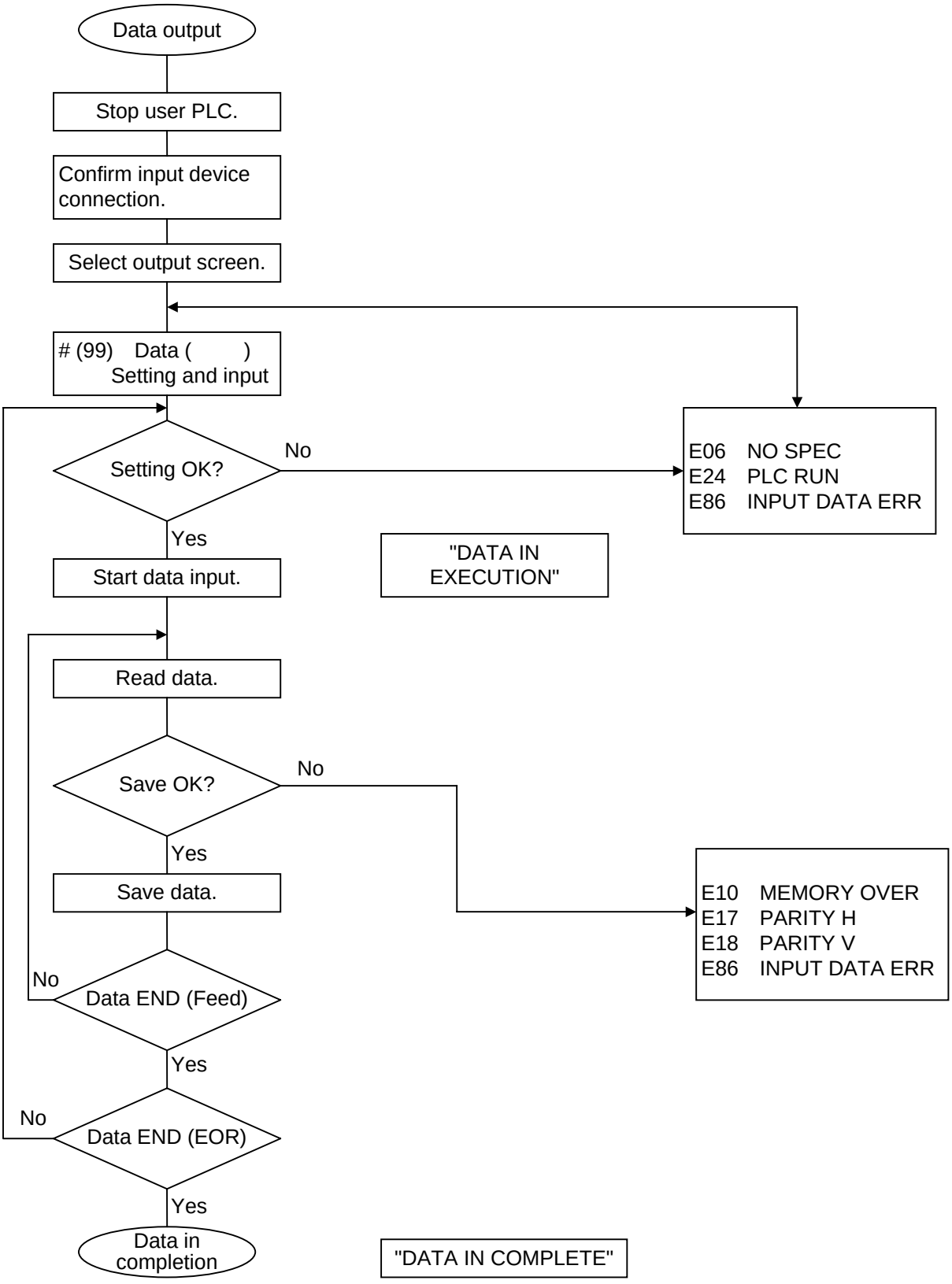
(Note 1) The setting of the data protection key will be ignored.
(Note 2) Data not found in the specifications will not be output.

9.2.3 Data Input and Compare

(1) Explanation of function
When the APLC is stopped, the data output from the input screen can be input and compared.
The target data is the same as for data output.
This function is valid only when the user PLC is stopped.

(Note 1) If data that exceeds the actual memory size is input or compared, the data of the memory size will be read in and then the message "E10 MEMORY OVER" will display. The operation will stop.
(Note 2) The data protection key setting will basically be ignored.
(Note 3) To stop the APLC, enter the emergency stop state, and then set rotary switch NCSYS to "1".

(2) Data input operation procedure



(3) Example of data input operation

Confirm that the user PLC is stopped.

Connect the input device.

Call out the data input screen.

DIAGN IN/OUT → MENU → INPUT

1) The data input screen will display.
2) The input mode is valid when the screen is selected.

Set 99 in # setting area.

(99) DATA ( )

```
< INPUT DATA >

< COMP. DATA >



# ( 99) DATA (          )
```

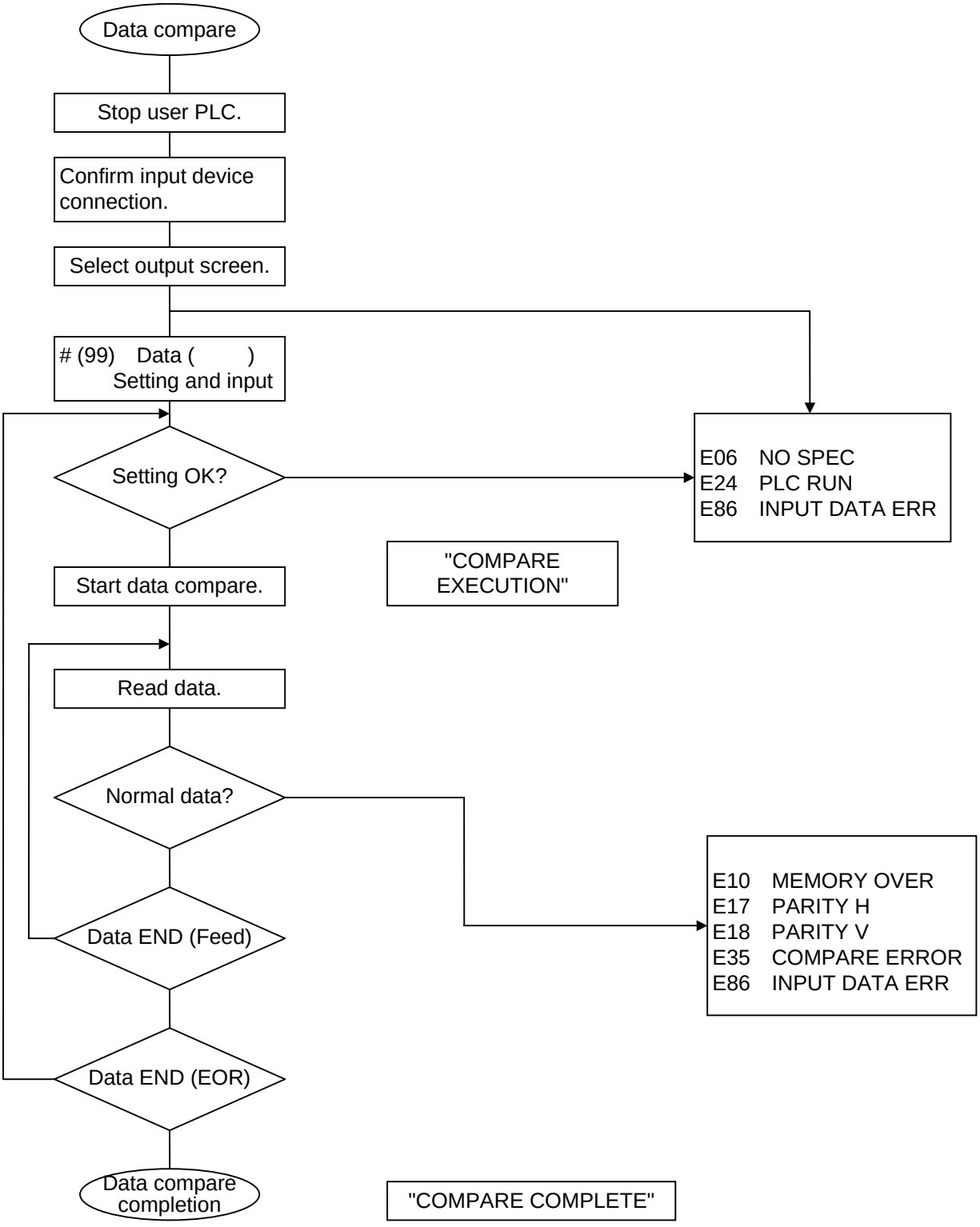
Press INPUT .

1) The input will start, and the input data details and "DATA IN EXECUTION" message will display.
   The No. of the data being input will display in the data setting area.

```
< INPUT DATA >
   N2L0L0L10001L10000;
< COMP. DATA >


                    DATA IN EXECUTION

# ( 99) DATA (      101)
```

```
< INPUT DATA >
  %
< COMP. DATA >


                    DATA IN COMPLETE

# (  □) DATA (          )
```

(4) Data compare operation procedure



(5) Example of data input operation

Confirm that the user PLC is stopped.

Connect the input device.

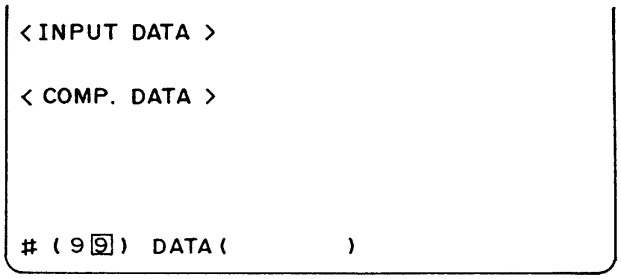
Call out the data input screen, and enter the compare mode.

(10) data (2) [INPUT]

1) The compare mode is canceled when the screen is changed, and the input mode will be validated. Thus, the mode must be set again.

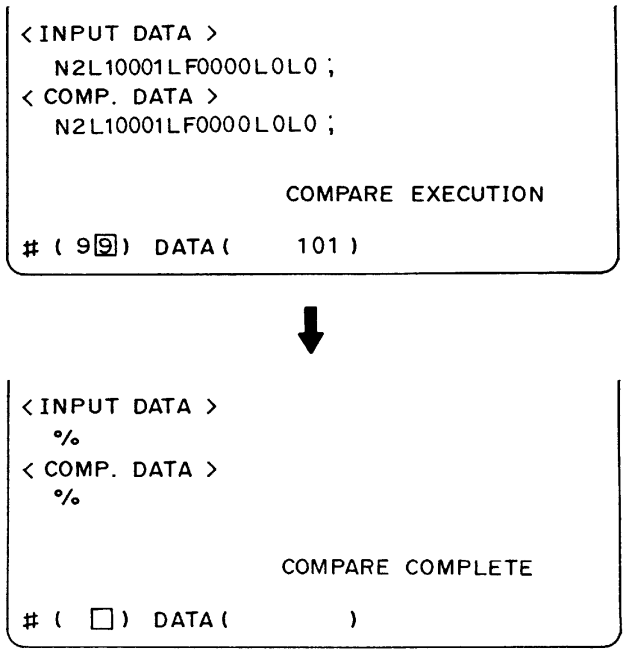
Set [9][9] in # setting area.

(99) DATA ( )



Press [INPUT].

1) The compare will start, and the data being compared and "COMPARE EXECUTION" message will display.
   The No. of the data will display in the data setting area.
2) When the data is compared normally to the end and the end code % (EOR) is read in, the message "COMPARE COMPLETE" will display.
3) If a compare error occurs, the message "E35 COMPARE ERROR" will display, and the operation will be stopped.



9.3 Data Input/Output Using Flash ATA Card

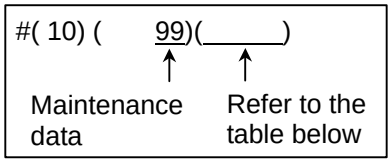
The data can be collectively input/output or compared in ATA CARD IN/OUT screen under the user PLC stop state.

9.3.1 Procedure to Use Flash ATA Card

(1) Stop the user PLC.
To stop the user PLC, enter the emergency stop state by pressing the emergency stop button, and set the rotary switch NCSYS to "1".

(2) Insert a flash ATA card to the slot.

(3) Call out ATA IN/OUT screen and select the maintenance data and its type as below.



Selecting the input/output maintenance data

3rd setting area	Type of input/output maintenance data	Extension
ALL1	System data No.100 to 149	.AL1
ALL2	File data No.200 to 231	.AL2
ALL3	PLC program No.250, No.252 to 299	.AL3
Individual data No.	Individual data (including No.150 to 199 and No.251)	.MNT
Blank	All ALL1 to ALL3	.ALL

(Note 1) For independent data, if the data type No. is set to n, "n.MNT" will be the independent data file name.

(Note 2) Refer to "9.1 Data Input/Output Function" for the data type.

(Note 3) Refer to Chapter 1 "2.7.6 Flash ATA Card I/F" for details of the flash ATA card input/output.

(4) Select the operation (input/output/comparison)

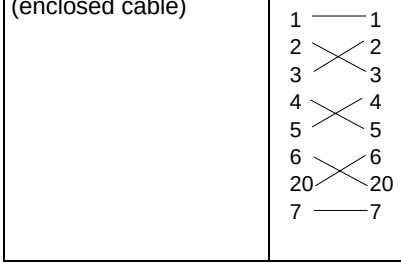
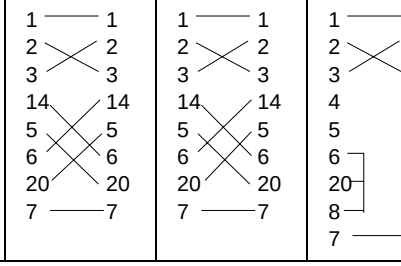
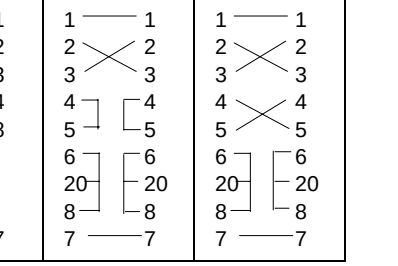
s : Eight or less alphanumeric setting, extension (may not include extension in some cases)
n : Data type No. (100 to 149, 150 to 199 200 to 231, 250 to 299)
k : Character string set in 3rd setting area at "#10 DATA" (ALL1, ALL2, ALL3, data type No.)
(10) (99) (k)

Operation	Setting method		
Input (#1)	**Basic setting**	**When k is "ALLx" : # (1) (File name in ATA card) ( )** **When k is "data type No." : # (1) (File name in ATA card) ( )**	
	Single file designation	#(1) (s) ( )	When k is "ALLx", the files s.ALx in the ATA card are input to the NC memory.
		#(1) (n) ( )	When k is "data type No.", the files n.MNT in the ATA card are input to the NC memory.
		#(1) ( ) ( )	When k is "data type No.", the files k.MNT in the ATA card are input to the NC memory.
Compare (#2)	**Basic setting**	**When k is "ALLx" : # (2) (File name in ATA card) ( )** **When k is "data type No." : # (2) (File name in ATA card) ( )**	
	Single file designation	#(2) (s) ( )	When k is "ALLx", the files s.ALx in the ATA card are compared with the data in the NC memory
		#(2) (n) ( )	When k is "data type No.", the files n.MNT in the ATA card are compared with the data in the NC memory.
		#(2) ( ) ( )	When k is "data type No.", the files k.MNT in the ATA card are compared with the data in the memory.
Output (#3)	**Basic setting**	**When k is "ALLx" : # (3) (File name in ATA card) ( )** **When k is "data type No." : # (3) (File name in ATA card) ( )**	
	Single file designation	#(3) (s) ( )	When k is "ALLx", the maintenance data in the NC memory is output as the file s.Alx in the ATA card.
		#(3) (n) ( )	When k is "data type No.", the maintenance data with the data type is output as the file n.MNT in the ATA card.
		#(3) ( ) ( )	When k is "data type No.", the maintenance data with the data type is output as the file k.MNT in the ATA card.

(Note) An “x” of “ALLx” or “.ALx” indicates one of 1, 2, or 3.

CHAPTER 4 APPENDICES

Appendix 1. Examples of RS-232C I/O Device Parameter Settings and Cable Connections

I/O device / Parameter	Tape reader (Mitsubishi)	Tape puncher (Mitsubishi)	Printer (Mitsubishi)	Floppy disk (Kyoritsu)	Floppy disk (Mitsubishi)	Floppy disk (Tanaka Business)
	PTR-02A	PTP-02A	PRT-02A	D-30	FD-3.5	TBM-F1
DEVICE NAME						
BAUD RATE	2	2	2	2	2	2
STOP BIT	3	3	3	3	3	3
PARITY CHECK	0	0	0	0	0	0
EVEN PARITY	0	0	0	0	0	0
CHR. LENGTH	3	3	3	3	3	3
HAND SHAKE	3	3	2	3	3	3
DC CODE PARITY	1	1	0	1	1	1
DC2/DC4 OUTPUT	0	0	0	1	0	1
CR OUTPUT	0	0	0/1	0	0	0
FEED CHR.	0	No. of characters	0	0	0	0
PARITY V	0	0/1	0	0	0/1	0/1
TIME-OUT (s)	100	100	100	100	100	100
Cable connection (enclosed cable)						

Appendix 2. 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.

2.1 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.

2.2 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.

2.3 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.

2.4 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.

2.5 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

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 0
#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 3. List of Function Codes

Function code ISO	Significance in control unit	Parity V count target	Screen display	Setting and display unit key in	Save in memory	Punch out output ISO	Function in control unit
0 to 9	Significant	Counted	Displayed	Key in enabled	Saved	0~9	Value data
A to Z	Significant	Counted	Displayed	Key in enabled	Saved	A~Z	Address
+	Significant	Counted	Displayed	Key in enabled	Saved	+	Sign, variable operator (+)
–	Significant	Counted	Displayed	Key in enabled	Saved	–	Sign, variable operator (–)
.	Significant	Counted	Displayed	Key in enabled	Saved	.	Decimal point
,	Significant	Counted	Displayed	Key in enabled	Saved	,	
/	Significant	Counted	Displayed	Key in enabled	Saved	/	Block delete (optional block skip) variable operator (÷)
%	Significant	Counted	Displayed (%)	Key in disabled (automatic insertion)	Saved	%	End of record (end of tape save) Rewind start and stop during tape search
LF/NL	Significant	Counted	Displayed (;)	Key in enabled ;/EOB	Saved	LF	End of block
(	Significant	Counted	Displayed	Key in enabled ;/EOB	Saved	(	Control out (comment start)
)	Significant	Counted	Displayed	Key in enabled ;/EOB	Saved	)	Control in (commend end)
:	Significant	Counted	Displayed	Key in disabled	Saved	:	Program No. address (Substitute for 0)
#	Significant	Counted	Displayed	Key in enabled	Saved	#	Variable No.
*	Significant	Counted	Displayed	Key in enabled	Saved	*	Variable operator (×)
=	Significant	Counted	Displayed	Key in enabled	Saved	=	Variable definition
[	Significant	Counted	Displayed	Key in enabled	Saved	[	Variable operator
]	Significant	Counted	Displayed	Key in enabled	Saved	]	Variable operator
BS	Insignificant	Counted	Blank	Key in disabled	Saved		
HT	Insignificant	Counted	Blank	Key in disabled	Saved		
SP	Insignificant	Counted	Blank	Key in enabled	Saved	SP (T-V) Automatic adjustment)	The SP from the EOB to the first character or number card is not subject to the parity V count.
CR	Insignificant	Counted	Blank	Key in disabled	Saved		
DEL	Insignificant	Do not count	Do not display	Key in disabled	Do not save		
NULL	Insignificant	Do not count	Do not display	Key in disabled	Do not save		
Other than the above	Insignificant	Counted	**(Note 2)**	Key in disabled	Saved		

(Note 1) Codes not shown above are saved on the tape, but data other than a comment 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 4. 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

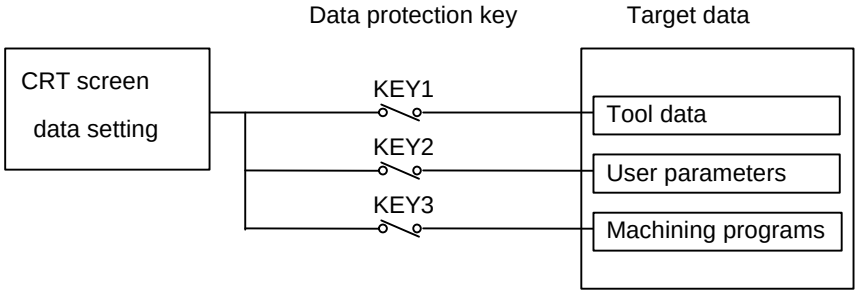
Appendix 5. Data Protection

5.1 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
7	Tape input of tool offset data	IN/OUT INPUT
8	Tape output of tool offset data	IN/OUT OUTPUT

(Note) When key 1 is off, 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".
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.
Manual numeric command also cannot be carried out on the TOOL screen.
When an attempt is made to perform the operation of number 7 or and/or 8, you cannot input or output data through the input key. Instead, the message "DATA PROTECT" will be displayed.

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
6	I/O basic parameter setting	IN/OUT (PARAMETER)
7	MR-J2-CT parameter	DIAGN AUX-PRM
8	Parameter tape input	IN/OUT/INPUT
9	Parameter tape output	IN/OUT/OUTPUT

(Note) When KEY 2 is off, pressing any key other than the MENU key on the screens corresponding to numbers 1 to 7 in Table 2 does not enter any data, but displays message "DATA PROTECT".
When an attempt is made to perform the operation of number 8 or 9, pressing the INPUT key cannot input or output any data, causing message "DATA or PROTECT" to be displayed.

3) Protection of machining program (KEY 3)
When KEY 3 is turned off, the operation items listed in Table 3 are inhibited.

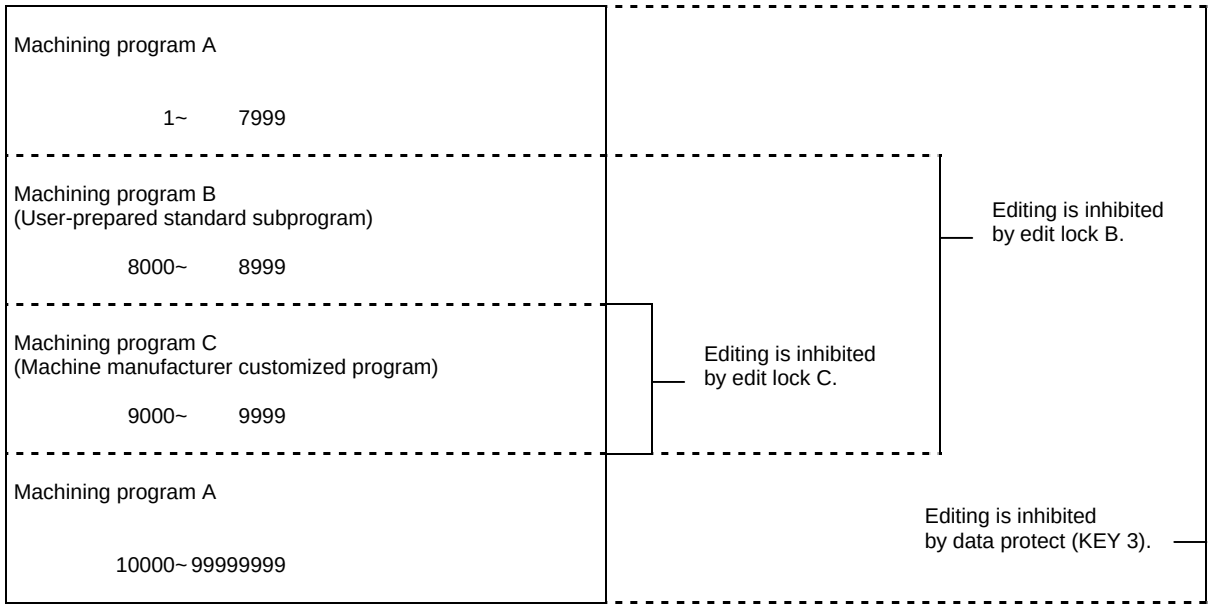
Table 3 Data protection by KEY 3

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	Storing and checking machining program in memory	IN/OUT/INPUT	—
6	Erasing machining program (single, group, all)	IN/OUT/ERASE	—
7	Setting comment of stored program	IN/OUT/FILE	—
8	Copying, condensing, and merging a machining program, and changing its number	IN/OUT/COPY	—
9	Outputting machining program	IN/OUT/OUTPUT	—
10	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 4 and 6 to 8 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 5, 9, or 10 in Table 3, pressing the INPUT key (for 5 and 9) or the cursor key (for 10) causes message "DATA PROTECT" to be displayed without the operation coming into effect.

5.2 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	IN	Input	○	×	×	○	○	×
		Collation	○	×	×	○	○	×
	OUT	Output	○	×	×	○	○	×
	COPY	Copy	○	×	×	○	○	×
		Condense	○	×	×	○	○	×
		Merge	○	×	×	○	○	×
		Program number change	○	×	×	○	○	×
	ERASE	Erase	○	×	×	○	○	×
	COMMENT	Comment setting	○	×	×	○	○	×
MONI- TOR	PROGRAM	Buffer	○	×	×	○	○	×

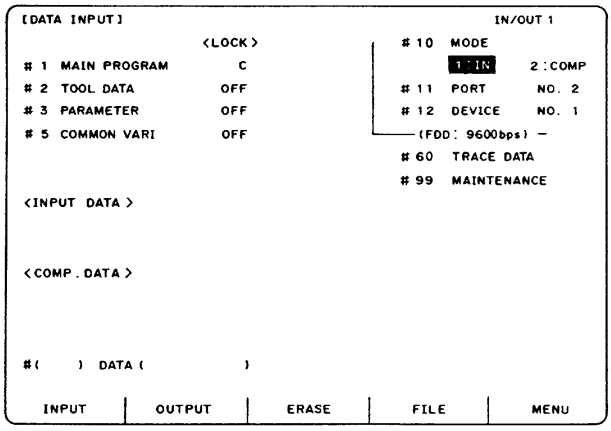
An attempt to perform any of the locked operations causes error message "E15 EDIT LOCK B" or "E16 EDIT LOCK C" to be displayed.

When an edit lock function is effective, the processing by the machining programs is executed except for those I/O functions that are locked. To prevent any negligence of necessary operation, the data protection conditions of machining programs are displayed in the DATA IN/OUT screen. Information displayed and the screen formats are as follows:

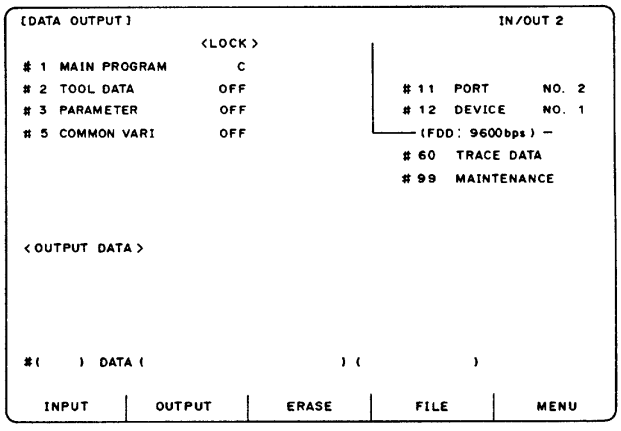
• Information displayed

Data	Protection state <LOCK>	Explanation
#1 Main program	ABC	Programs A, B, and C are protected (Protection KEY 3 off)
	BC	Programs B and C are protected - Edit lock B (Protection KEY 3 on)
	C	Program C is protected - Edit lock C (Protection KEY 3 on, edit lock B off)
#2 Tool data	ON	Tool data is protected (Protection KEY 1 off)
	OFF	(Protection KEY 1 on)
# Parameter	ON	User parameter data is protected (Protection KEY 2 off)
	OFF	(Protection KEY 2 on)

Data input screen



Data output screen



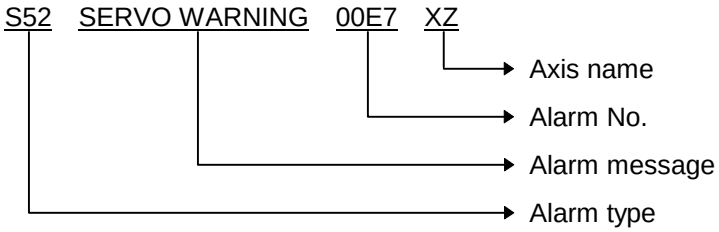
Appendix 6. Table of Conversion Codes for Error Code Output

With this function, the NC alarms and errors which are normally output to NC screen are partially coded and output to PLC I/F devise. Thus, the contents of alarms and errors can be confirmed without NC screen.

6.1 Code Conversion Specifications

The following output will be made when an alarm occurs.
The message displayed in NC screen is not converted.

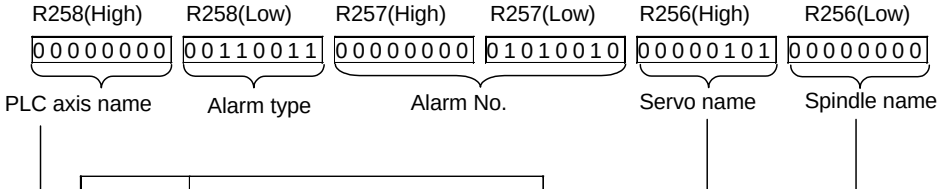
(Example) When servo alarm S52 occurs



The output is as shown above on the NC screen. However, this can be coded and output to PLC I/F devices with this function.

Alarm type	Converted into a 2-digit numeral code and output. (Refer to the table in 6.2.)
Alarm message	Not coded, and not output.
Alarm No.	The No. is output as HEX, but the head digit is the axis name data for some alarms. The head digit of the alarm without axis name will be “0”.
Axis name.....	Error occurrence axis bit is expressed as a numeral and output as the servo or the spindle respectively.

The code output in the above example becomes 00330052, and the output to PLC I/F devices 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, spindle name and PLC axis 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 system are 6, and the max spindle Nos. in the 1st system are 7. The spindle alarm is output to the 1st system.

Axis name is added for the messages such as some of M01, S01 to S52, Y10, Z70 to Z73.
The following are used for the output PLC I/F devices.

1st part system : R256, R257, R258

2nd part system: R356, R357, R358

Refer to the input/output interface signal table.

6.2 Code Table

Alarm types

Alarm		Alarm type	Contents	Axis name	Order of priority
Operation error	M01	11	OPERATION ERROR	Some are added.	8
Stop code	T01	21	CAN'T CYCLE ST	Not added.	9
	T02	22	FEED HOLD		
	T03	23	BLOCK STOP		
	T04	24	COLLATION STOP		
	T10	26	FIN WAIT		
Servo/spindle alarm	S01	31	SERVO ALARM:PR	Added.	2
	S02	32	INIT PARAM ERR		
	S03	33	SERVO ALARM:NR		
	S04	36	SERVO ALARM:AR		
	S51	34	PARAMETER ERROR	Added.	6
	S52	35	SERVO WARNING		
MCP alarm	Y02	41	SYSTEM ALARM	Not added.	3
	Y03	42	AMP. UNEQUIPPED		
	Y10	48	Drv S/W Differnt	Added	
	Y51	45	PARAMETER ERROR		
	Y90	47	SP. NON SIGNAL		
System alarm	Z30		ETHERNET ERROR	Not added.	7
	Z52	52	BATTERY FAULT		
	Z53	53	TEMP. OVER		
	Z55	54	RIO NOT CONNECT		
	Z59	59	TIME CONSTANT		
	Z70	55	ABS. ILLEGAL	Added.	1
	Z71	56	DETECTOR ERROR		
	Z72	57	COMPARE ERROR		
	Z73	58	ABS. WARNING		
	P990	61	PREPRO S/W ERR	Not added.	*5
Program error	P○○○	71	(Program error)	Not added.	5
Auxiliary axis (MR-J2-CT) alarm (Note1)	M00	81	AUX OPER.ALM	Added.	8
	M01	82	AUX OPER.ALM		
	S01	83	AUX SERVO ALM	Added.	2
	S02	84	AUX SERVO ALM		
	S03	85	AUX SERVO ALM		
	S52	86	AUX SERVO WRN		
	Y02	87	AUX SYSTEM ALM	Some are added.	3
	Y03	88	AUX AMP UNEQU.		
	EMG	89	AUX EMERGENCY		
	Z70	8A	AUX POS. ERR	Added.	1
	Z71	8B	AUX DETEC. ERR		
	Z73	8C	AUX SYSTEM WRN		
Emergency stop message	EMG	01	EMERGENC	Not added.	4

(Note1) If multiple alarms occur simultaneously, only the alarm with the highest order of priority in the code table will be output.
Note that the auxiliary axis name is output to the servo axis name area.

The massage at emergency stop is displayed in NC screen shown below.

(Example) EMG EMERGENCY STOP PLC

When the emergency stop occurs, the message is coded as the following table and an alarm No. is output

Emergency stop code list

Error message		Details	Alarm No.
EMG EMERGENCY	EXIN	External emergency stop	0000
	PLC	User PLC emergency stop	0001
	SRV	Servo drive unit not ready	0002
	STOP	User PLC not running	0003
	SPIN	Spindle Amp not ready	0004
	PC_H	PLC high-speed process error	0005
	PARA	Door-open II fixed device setting illegal	0006
	LINK	TOYOPUC communication error	0007
	WAIT	Waiting TOYOPUC connection	0008
	XTEN	External emergency stop	0009
	LAD	User PLC illegal code	0010

The alarm with 4-digit alarm No. and axis name will be coded into 3-digit alarm No. and its axis name will be the head digit.

6.3 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, "0000" will be set in the alarm No. area output to the PLC I/F device.

(3) If an alarm that does not have an axis name occurs, "0" will be set in the axis name area output to the PLC I/F device.

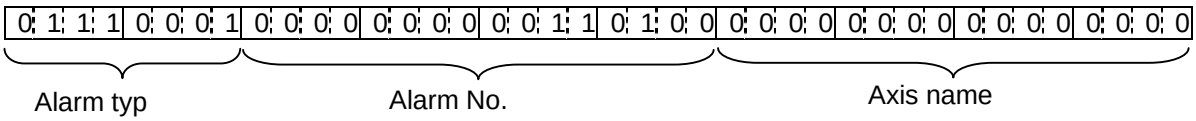
(4) Alarms not shown in the code table are not output to the PLC I/F device.

(5) If multiple alarms occur simultaneously, only the alarm with the highest order of priority in the code table will be output.

(6) The "○○○" section of the program error "P○○○" is output to the alarm No. area.

(Example) When "P34 G-CODE ERROR" occurs, the output will be as follows.

The code to be output is “71003400” and output to the PLC interface as follows.



Appendix 7. List of Alarms

7.1 Operation Alarms (Mxxx)

(The bold characters are the messages displayed on the screen.)

M01 OPERATION ERROR	Alarms occurring due to incorrect operation by the operator during NC operation and those by machine trouble are displayed. The axis name may also appears according to the error No.

Error No.	Details	Remedy
0001	DOG OVERRUN (Dog overrun) When returning to the reference point, the near-point detection limit switch did not stop over the dog, but overran the dog.	• Increase the length of the near-point dog. • Reduce the reference point return speed.
0002	Z-AX NO CRSS One of the axes did not pass the Z-phase during the initial reference point return after the power was turned on.	• Move the detector one rotation or more in the opposite direction of the reference point, and repeat reference point return.
0003	INVALID RET (Invalid return) When manually returning to the reference point, the return direction differs from the axis movement direction selected with the AXIS SELECTION KEY.	• The selection of the AXIS SELECTION key's + or – direction is incorrect. The error is canceled by feeding the axis in the correct direction.
0004	EXT INTRLK (External interlock) The external interlock function has activated (the input signal is "OFF") and one of the axes has entered the interlock state.	• As the interlock function has activated, release it before resuming operation. • Check the sequence on the machine side. • Check for broken wires in the interlock signal line.
0005	INTRL INTRLK (Internal interlock) 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 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.

Error No.	Details	Remedy
0006	H/W STRK END (H/W stroke end) The stroke end function has activated (the input signal is “OFF”) and one of the axes is in the stroke end status.	• As the stroke end limit switch has activated (the stroke end status is established), move the machine manually. • Check for broken wires in the stroke end signal wire. • Check for trouble in the limit switch.
0007	S/W STRK END (S/W stroke end) The stored stroke limit I, II, IIB or IB function has activated.	• As the machine is at the stroke end, move it manually. • If the stored stroke limit in the parameter is incorrectly set, correct it.
0008	Chuck/tail-stock barrier stroke end axis found The chuck/tail-stock barrier function turned on, and an axis entered the stroke end state.	• Reset the alarm with reset, and move the machine in the reverse direction.
0009	Reference point return number illegal Return to the No. 2 reference point was performed before return to the No. 1 reference point was completed.	• Execute No. 1 reference point return.
0020	Reference point retract impossible Retraction to the reference point was made to be performed before the coordinates had not been established.	• Execute reference point return.
0024	Reference point return disabled during absolute position detection alarm A reference point return signal was input during an absolute position detection alarm.	• Reset the absolute position detection alarm, and then perform reference point return.
0025	Reference point return disabled during zero point initialization A reference point return signal was input during zero point initialization of the absolute position detection system.	• Complete zero point initialization, and then perform reference point return.
0051	Synchronization error too large The synchronization error of the master and slave axes exceeded the allowable value under synchronous control.	• Select the correction mode and move one of the axes in the direction in which the errors are reduced. • Increase the allowable value or reset it to 0 (check disabled).
0101	NOT OP MODE (Not operation mode)	• Check for a broken wire in the input mode signal wire. • Check for trouble in the mode selector switch. • Check the sequence program.

Error No.	Details	Remedy
0102	OVERRIDE ZERO (Override zero) The cutting feed override switch on the machine operation panel is set to zero.	• Set the switch to a value other than zero to release the error. • If the switch is set to a value other than zero, check for a short circuit in the signal wire. • Check the sequence program.
0103	EX F SPD ZRO (External feed speed zero) The manual feed speed switch on the machine operation panel is set to zero when the machine is in the jog mode or automatic dry run mode.	• Set the switch to a value other than zero to release the error. • If the switch is set to a value other than zero, check for a short circuit in the signal wire. • Check the sequence program.
0104	F1 SPD ZRO (F1-digit speed zero) The F1-digit feed rate is set to zero when the F1-digit feed command is being executed.	• Set the F1-digit feed rate on the setup parameter screen.
0105	SPINDLE STP (Spindle stop) The spindle stopped during the synchronous feed command.	• Rotate the spindle. • If the workpiece is not being cut, start dry run. • Check for a broken wire in the spindle encoder cable. • Check the connections for the spindle encoder connectors. • Check the spindle encoder pulse.
0106	HNDL FD NOW (Handle feed axis No. illegal) An axis not found in the specifications was designated for handle feed or the handle feed axis was not selected.	• Check for broken wires in the handle feed axis selection signal wire. • Check the sequence program. • Check the No. of axes listed in the specifications.
0107	SPDL RPM EXS (Spindle speed excessive) The spindle speed exceeded the axis clamp speed during the thread cutting command.	• Lower the commanded spindle speed.
0108	Fixed point mode feed axis No. illegal: An axis not found in the specifications was designated for the fixed point mode feed or the fixed point mode feed rate is illegal.	• Check for broken wires in the fixed mode feed axis selection signal wire and fixed point mode feed rate wire. • Check the fixed point mode feed specifications.
0109	BLK ST INTLK (Block start interlock) An interlock signal that locks the start of the block has been input.	• Check the sequence program.
0110	CTBL ST INTLK (Cutting block start interlock) An interlock signal that locks the start of the cutting block has been input.	• Check the sequence program.

Error No.	Details	Remedy
0112	Program Check Mode The automatic start button was pressed during program check or in program check mode.	• Press the reset button to cancel the program check mode.
0115	RESETTING The automatic start pushbutton was pressed during resetting or tape rewinding.	• 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 resetting to end, and then press the automatic start pushbutton.
0117	PLAYBACK NOT POSSIBLE The playback switch was turned on during editing.	• During editing, cancel the function by pressing the input or previous screen key, and then turn on the playback switch.
0120	Synchronization correction mode ON The synchronous correction mode switch was pressed in a mode other than handle or manual feed mode.	• Select the handle or manual feed mode. • Turn off the correction mode switch.
0121	No synchronous control option The synchronous control system (register R932) was set with no synchronous control option.	• Set 0 in register R932.
0124	Simultaneous axis movement prohibited during inclined axis control valid The basic axis corresponding to the inclined axis was started simultaneously in the manual mode while the inclined axis control was valid.	• Turn the inclined axis and basic axis start OFF for both axes. (This also applied for manual/automatic simultaneous start.) • Invalidate the basic axis compensation, or command one axis at a time.
0160	Axis with no maximum speed set for the outside of the soft limit range Returned from the outside of the soft limit range for the axis with no maximum speed set for the outside of the soft limit range.	• Set the maximum speed for the outside of the soft limit range. • Change the soft limit range.
1005	G114.* was executed during G114.*	• Cancel with G113. • Cancel with the spindle synchronization cancel signal (SPSYC).
1106	Spindle synchronous phase calculation illegal The spindle synchronization phase alignment command was issued while the spindle synchronization phase calculation request signal was ON.	• Check the program. • Check the sequence program.

(The bold characters are the messages displayed on the screen.)

M90 PARAM SET MODE	This message appears when the setup parameter lock function is valid.	
Error No.	**Details**	**Remedy**
–	Setup parameter lock unlocked The setup parameter lock is unlocked. The setup parameters can be set, but automatic start is not possible in this state.	• Refer to the instruction manual issued by the machine maker.

7.2 Stop Codes (Txxx)

These codes indicate a status that caused the controller to stop for some reason.
(The bold characters are the messages displayed on the screen.)

T01 CAN'T CYCLE ST	This indicates the state where automatic operation cannot be started when attempting to start it from the stop state.

Error No.	Details	Remedy
0101	AX IN MOTION (axis in motion) Automatic start is not possible as one of the axes is moving.	• Try automatic start again after all axes have stopped.
0102	READY OFF Automatic start is not possible as the NC is not ready.	• Another alarm has occurred. Check the details and remedy.
0103	RESET ON Automatic start is not possible as the reset signal has been input.	• Turn off the reset input signal. • Check that the reset switch is not on constantly due to trouble. • Check the sequence program.
0104	A-OP STP SGL (Automatic operation stop signal "ON") The FEED HOLD switch on the machine operation panel is "ON" (valid).	• Check the FEED HOLD switch. The feed hold switch is the B contact. • Check for broken wires in the feed hold signal wire. • Check the sequence program.
0105	H/W STRK END (H/W stroke end axis) Automatic start is not possible as one of the axes is at the stroke end.	• If one of the axis' ends is at the stroke end, move the axis manually. • Check for broken wire in the stroke end signal wire. • Check for trouble in the stroke end limit switch.
0106	S/W STRK END (S/W stroke end axis) Automatic start is not possible as one of the axes is at the stored stroke limit.	• Move the axis manually. • If an axis is not at the end, check the parameter details.
0107	NO OP MODE (NO operation mode) The operation mode has not been selected.	• Select the automatic operation mode. • Check for broken wires in the automatic operation mode (memory, tape, MDI) signal wire.
0108	OP MODE DUPL (Operation mode duplicated) Two or more automatic operation modes are selected.	• Check for a short circuit in the mode selection signal wire (memory, tape, MDI). • Check for trouble in the switch. • Check the sequence program.
0109	OP MODE SHFT (Operation mode shift) The automatic operation mode changed to another automatic operation mode.	• Return to the original automatic operation mode, and start automatic start.

Error No.	Details	Remedy
0110	Tape search execution Automatic start is not possible as tape search is being executed.	• Begin automatic start after the tape search is completed.
0113	Thermal alarm Automatic start is not possible because a thermal alarm (Z53 TEMP. OVER) has occurred.	• The NC controller temperature has exceeded the specified temperature. Take appropriate measures to cool the unit.
0138	Disabled start during absolute position detection alarm A start signal was input during an absolute position detection alarm. This check is performed on only the NC axes. Thus, even if the alarms related to the absolute position detection occurs on the PLC axis, the start signal is available.	• Reset the absolute position detection alarm, and then input the start signal.
0139	Disabled start during zero point initialization A start signal was input while initializing the absolute position detector's zero point. This check is performed on only the NC axes. Thus, even if the PLC axis is under the zero-point initializing state, the star signal is available.	• Complete zero point initialization before inputting the start signal.
0141	Start during MDI operation at other part system disable When using the multiple part system specifications, the start signal was input in the MDI mode while MDI operation was being carried out with another part system.	• End the other part system's operation before starting.
0190	Automatic start disable Automatic start is not possible because the setup parameter setting enabled state is entered.	• Refer to the instruction manual issued by the machine maker.
0191	Automatic start disable Automatic start was attempted when deleting/reading a file.	• Carry out automatic start after the file deleting/reading process is completed.

T02 FEED HOLD	The feed hold state been entered due to a condition in the automatic operation.

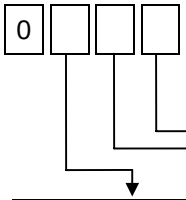
Error No.	Details	Remedy
0201	H/W STRK END (H/W stroke end axis) An axis is at the stroke end.	• Manually move the axis away from the stroke end limit switch. • The machining program must be corrected.
0202	S/W STRK END (S/W stroke end axis) An axis is at the stored stroke limit.	• Manually move the axis. • The machining program must be corrected.
0203	RSET SGNL ON (Reset signal on) The reset signal has been input.	• The program execution position has returned to the start of the program. Execute automatic operation from the start of the machining program.
0204	AUTO OP STOP (Automatic operation stop) The FEED HOLD switch is “ON”.	• Resume automatic operation by pressing the “CYCLE START” switch.
0205	AUTO MD CHING (Automatic mode change) The operation mode changed to another mode during automatic operation.	• Return to the original automatic operation mode, and resume automatic operation by pressing the “CYCLE START” switch.
0206	Acceleration and deceleration time constants too large The acceleration and deceleration time constants are too large. (This problem occurs at the same time as system alarm Z59.)	• Increase the set value of the “#1206 G1bF” parameter. • Decrease the set value of the “#1207 G1btL” parameter. • Lower the cutting speed.
0215	Absolute position detection alarm stop An absolute position detection alarm occurred.	• Reset the absolute position detection alarm.

T03 BLOCK STOP	This indicates that automatic operation stopped after executing one block of the program.

Error No.	Details	Remedy
0301	SNGL BLK ON (Single block on) The SINGLE BLOCK switch on the machine operation panel is ON.	• Automatic operation can be resumed by turning the CYCLE START switch ON.
0302	User macro stop Block stop was commanded in the user macro program.	• The automatic operation can be resumed by turning ON the cycle start switch.
0303	Mode change The automatic mode changed to another automatic mode.	• Return to the original automatic operation mode, and resume automatic operation by turning the CYCLE START switch ON.
0304	MDI completion The last block of MDI was completed.	• Set MDI again, and turn the CYCLE START switch ON to resume MDl operation.
0305	Block cutting start interlock The interlock signal that locks the block cutting start is entered.	• Check the sequence program.
0306	Block cutting start interlock The interlock signal that locks the block cutting start is entered.	• Check the sequence program.
0310	Offset change of inclined Z-axis during program operation Whether to validate the offset of the inclined Z-axis switched during program operation.	• Automatic operation can be restarted by turning on the cycle start switch.

T10 FIN WAIT	This indicates the operation state when an alarm did not occur during automatic operation, and nothing seems to have happened.

Error No.	**Details**
0□□□	The error number is displayed while each of the completion wait modes listed in the table below is on. It disappears when the mode is canceled.



Alarm No.	Waiting for unclamp to complete (Note 2)	Dwelling
0		
1		×
8	×	
9	×	×

Alarm No.	Door open (Note 1)	Spindle position loop wait
0		
1		×
8	×	
9	×	×

Alarm No.	Waiting for spindle orientation to complete	Waiting for cutting speed deceleration	Waiting for rapid traverse deceleration	Waiting for MSTB completion
0				
1				×
2			×	
3			×	×
4		×		
5		×		×
6		×	×	
7		×	×	×
8	×			
9	×			×
A	×		×	
B	×		×	×
C	×	×		
D	×	×		×
E	×	×	×	
F	×	×	×	×

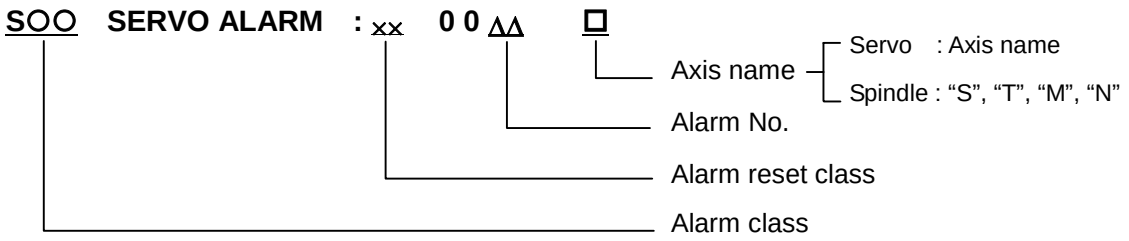
(Note 1) Door open: This mode is enabled by the door interlock function.

(Note 2) Waiting for unclamp to end when "#2076 index_x" = 1.

7.3 Servo · Spindle Alarms

This section describes alarms occurred by the errors in the servo system such as the drive unit, motor and encoder, etc. The alarm message, alarm No. and axis name will display on the alarm message screen. The axis where the alarm occurred and the alarm No. will also display on the servo monitor screen and the spindle monitor screen respectively. If several alarms have occurred, up to two errors per axis will display on the servo monitor screen and the spindle monitor screen respectively.

(The bold characters are the messages displayed on the screen.)



(Note 1) The alarm class and alarm reset class combinations are preset.
(Refer to the separate table for S02, S51 and S52.)

Alarm class	Alarm reset class	Resetting methods
S01	PR	After removing the cause of the alarm, reset the alarm by turning the NC power ON again.
S03	NR	After removing the cause of the alarm, reset the alarm by inputting the NC RESET key.
S04	AR	After removing the cause of the alarm, reset the alarm by turning the drive unit power ON again.

(Note 2) The resetting method may change according to the alarm class.
For example, even if "S03 SERVO ALARM: NR" is displayed, it may be necessary to turn the NC power ON again.

Alarm No.	Name	Meaning
10	Insufficient voltage	Insufficient PN bus voltage was detected in main circuit.
11	Axis selection error	Setting of the axis No. selection switch is incorrect.
12	Memory error 1	A CPU error or an internal memory error was detected during the power ON self-check.
13	Software processing error 1	Software processing has not finished within the specified time.
14	Software processing error 2	Software processing has not finished within the specified time.
15	Memory error 2	A CPU error or an internal memory error was detected during the power ON self-check.
16	Magnetic pole position detection error	Initial magnetic pole for motor control has not been formed yet.
17	A/D converter error	An error was detected in the A/D converter for detecting current FB.
18	Motor side detector: Initial communication error	Initial communication with the motor end detector failed.
19	Detector communication error in synchronous control	Initial communication with the motor end detector on master axis failed when setting closed-loop current synchronous control. Or the communication was interrupted.

Alarm No.	Name	Meaning
1A	Machine side detector: Initial communication error	Initial communication with the linear scale or the ball screw end detector failed.
1B	Machine side detector: CPU error 1	CPU initial error was detected in the linear scale or in the ball screw end detector.
1C	Machine side detector: EEPROM/LED error	An error was detected in the stored data of the linear scale memory. Or the LED deterioration was detected in the ball screw end detector.
1D	Machine side detector: Data error	An error data was detected in the linear scale or in the ball screw end detector.
1E	Machine side detector: Memory error	An internal memory error was detected in the linear scale.
1F	Machine side detector: Communication error	An error was detected in communication data with the linear scale or the ball screw end detector. Or the communication was interrupted.
20	Motor side detector: No signal	No signals were detected in A,B,Z-phase or U,V,W-phase of the pulse motor end detector in a servo system, or in Z-phase of PLG in a spindle system.
21	Machine side detector: No signal	No signals were detected in A,B,Z-phase of the pulse linear scale or the ball screw end detector in a servo system. Or no encoder signals were detected in a spindle system.
22	LSI error	LSI operation error was detected in the drive unit.
23	Excessive speed error 1	A difference between the speed command and speed feedback was continuously exceeding 50 r/min for longer than the setting time.
24	Grounding	The motor power cable is in contact with FG (Frame Ground).
25	Absolute position data lost	The absolute position was lost, as the backup battery voltage dropped in the absolute position detector.
26	Unused axis error	A power module error occurred in the axis whose axis No. selection switch was set to "F"(free axis).
27	Machine side detector: CPU error 2	A CPU error was detected in the linear scale.
28	Machine side detector: Overspeed	The specified max. speed was detected in the linear scale.
29	Machine side detector: Absolute position data error	An error was detected in the absolute position detection circuit of the linear scale.
2A	Machine side detector: Relative position data error	An error was detected in the relative position detection circuit of the linear scale.
2B	Motor side detector: CPU error 1	A CPU initial error was detected in the motor end detector or in the linear scale of a linear servo system.
2C	Motor side detector: EEPROM/LED error	The LED deterioration was detected in the motor end detector. Or an error was detected in the stored data of the linear scale memory of a linear servo system.
2D	Motor side detector: Data error	A data error was detected in the motor end detector or in the linear scale of a linear servo system.

Alarm No.	Name	Meaning
2E	Motor side detector: Memory error	An internal memory error was detected in the linear scale of a linear servo system.
2F	Motor side detector: Communication error	An error was detected in communication data with the motor end detector or with the linear scale of a linear servo system. Or the communication was interrupted.
30	Over regeneration	Over-regeneration detection level became over 100%. The regenerative resistor is overloaded.
31	Overspeed	The motor was detected to rotate at a speed exceeding the allowable speed.
32	Power module overcurrent	Overcurrent protection function in the power module has started its operation.
33	Overvoltage	PN bus voltage in main circuit exceeded the allowable value.
34	NC-DRV communication: CRC error	An error was detected in the data received from the CNC.
35	NC command error	The travel command data that was received from the CNC was excessive.
36	NC-DRV communication: Communication error	The communication with the CNC was interrupted.
37	Initial parameter error	An incorrect parameter was detected among the parameters received from the CNC at the power ON.
38	NC-DRV communication: Protocol error 1	An error was detected in the communication frames received from the CNC.
39	NC-DRV communication: Protocol error 2	An error was detected in the axis information data received from the CNC.
3A	Overcurrent	Excessive current was detected in the motor drive current.
3B	Power module overheat	Thermal protection function in the power module has started its operation.
3C	Regeneration circuit error	An error was detected in the regenerative transistor or in the regenerative resistor.
3D	Spindle speed blocked	The spindle motor failed to rotate faster than 45 r/min, even when the max. torque command was given.
3E	Spindle speed overrun	1. The spindle motor speed feedback was detected to be accelerated exceeding the commanded speed. 2. The spindle motor was detected to be rotated at a speed exceeding the parameter value, while the speed command was "0" (including the case of operation stoppage during the position control).
3F	Excessive speed error 2	A difference between the speed command and speed feedback was detected to exceed the setting amount or setting time in a constant speed operation.
40	Detector selection unit switching error	An error was detected in the motor switching signals that were received from the detector selection unit, while controlling one drive unit and two motors.
41	Detector selection unit communication error	An error was detected in the communication with the detector selection unit, while controlling one drive unit and two motors.

Alarm No.	Name	Meaning
42	Feedback error 1	An error was detected in the feedback signals of the pulse motor end detector in a servo system, or in PLG's feedback signals in a spindle system.
43	Feedback error 2	Excessive difference was detected in position data between the motor end detector and the machine end detector in a servo system. In a spindle system, an error was detected in the encoder feedback signals.
44	Inappropriate coil selected for C axis	When using a coil changeover motor, C-axis was controlled while the high-speed coil was selected.
45	Fan stop	A cooling fan built in the drive unit stopped, and the loads on the unit exceeded the specified value.
46	Motor overheat	Thermal protection function of the motor or in the detector, has started its operation.
47	Regenerative resistor overheat	Thermal protection function of the regenerative resistor, has started its operation.
48	Motor side detector: CPU error 2	A CPU error was detected in the linear scale of a linear servo system.
49	Motor side detector: Overspeed	The specified max. speed was detected in the linear scale of the linear servo system.
4A	Motor side detector: Absolute position data error	An error was detected in the absolute position detection circuit in the linear scale of a linear servo system.
4B	Motor side detector: Relative position data error	An error was detected in the relative position detection circuit in the linear scale of a linear servo system.
4C	Current error at magnetic pole detection	A current error was detected in the IPM spindle motor when the initial magnetic pole was being formed.
4E	NC command mode error	The mode outside the specification was input in spindle control mode selection.
4F	Instantaneous power interruption	The power was momentarily interrupted.
50	Overload 1	Overload detection level became over 100%. The motor or the drive unit is overloaded.
51	Overload 2	Current command of more than 95% of the unit's max. current was being continuously given for longer than 1 second in a servo system. In a spindle system, the load over the continuous rating was being applied for longer than 30 minutes.
52	Excessive error 1	A difference between the actual and theoretical motor positions during servo ON exceeded the setting value in a servo system. In a spindle system, a difference between the position command and position feedback exceeded the setting value.
53	Excessive error 2	A difference between the actual and theoretical motor positions during servo OFF exceeded the setting value.
54	Excessive error 3	When an excessive error 1 occurred, detection of the motor current failed.
55	External emergency stop error	There is no contactor shutoff command, even after 30 seconds has passed since the external emergency stop was input.
57	Option error	An invalid option function was selected.
58	Collision detection 1: G0	When collision detection function was valid, the disturbance torque in rapid traverse (G0) exceeded the collision detection level.

Alarm No.	Name	Meaning
59	Collision detection 1: G1	When collision detection function was valid, the disturbance torque in cutting feed (G1) exceeded the collision detection level.
5A	Collision detection 2	When collision detection function was valid, the command torque reached the max. motor torque.
5C	Orientation feedback error	After orientation was achieved, a difference between the command and feedback exceeded the parameter setting.
5D	Speed monitoring: Input mismatch	As for door state signal of speed monitoring control, a mismatch between the external input signal and the control signal received from the CNC was detected.
5E	Speed monitoring: Feedback speed error	In speed monitoring control, the spindle speed was exceeding the setting speed with the door open.
5F	External contactor error	A contact of the external contactor is welding. Or the contactor fails to be ON during ready ON.
61	Power module overcurrent	Overcurrent protection function in the power module has started its operation.
62	Frequency error	The input power supply frequency increased above the specification range.
63	Supplementary regeneration error	The supplementary regenerative transistor is being ON.
65	Rush relay error	A resistor relay for rush short circuit fails to be ON.
67	Phase interruption	An open-phase condition was detected in input power supply circuit.
68	Watchdog	The system does not operate correctly.
69	Grounding	The motor power cable is in contact with FG (Frame Ground).
6A	External contactor welding	A contact of the external contactor is welding.
6B	Rush relay welding	A resistor relay for rush short circuit fails to be OFF.
6C	Main circuit error	An error was detected in charging operation of the main circuit capacitor.
6D	Parameter error	The capacity of the power supply unit and the regenerative resistor type that was set in the parameter are mismatched.
6E	Memory error	An internal memory error was detected.
6F	Power supply error	A power supply unit is not connected. Or an error was detected in A/D converter of the power supply unit.
71	Instantaneous power interruption	The power was momentarily interrupted.

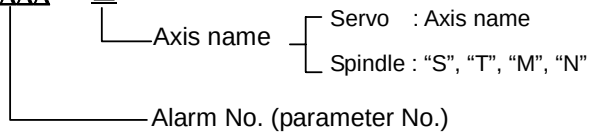
Alarm No.	Name	Meaning
73	Over regeneration	Over-regeneration detection level became over 100%. The regenerative resistor is overloaded.
74	Regenerative resistor overheat	Thermal protection function of the regenerative resistor, has started its operation.
75	Overvoltage	PN bus voltage in main circuit exceeded the allowable value.
76	External emergency stop setting error	As for the external emergency stop settings, the setting on the rotary switch and the parameter setting are mismatched.
77	Power module overheat	Thermal protection function in the power module has started its operation.
7F	Drive unit power supply restart request	A mismatch of program mode selection was detected. Turn the drive unit power ON again.
80	Detector converting unit 1: Connection error	A connection error was detected between the analog output linear scale and the unit MDS-B-HR that is used in a linear servo system.
81	Detector converting unit 1: Communication error	A communication error was detected between the serial output linear scale and the unit MDS-B-HR that is used in a linear servo system.
83	Detector converting unit 1: Judgment error	Judgment of the linear scale analog frequency failed in the unit MDS-B-HR that is used in a linear servo system.
84	Detector converting unit 1: CPU error	A CPU error was detected in the unit MDS-B-HR that is used in a linear servo system.
85	Detector converting unit 1: Data error	A data error was detected in the unit MDS-B-HR that is used in a linear servo system.
86	Detector converting unit 1: Magnetic pole error	An error was detected in the magnetic pole of the unit MDS-B-HR that is used in a linear servo system.
88	Watchdog	The system does not operate correctly.
89	Detector converting unit 2: Connection error	A connection error was detected between the analog output linear scale and the unit MDS-B-HR in a servo system. In a spindle system, the initial communication with MDS-B-PJEX failed.

(Note 1) With alarm “73”, to prevent immediately resumption of operation from the overregeneration state, the alarm cannot be released unless the control power (L11,L12) continuity state has continued for 15 minutes or more after the alarm has occurred. The alarm cannot be released even if the NC power or control power is turned ON immediately after the alarm occurs. If the power is turned ON immediately after the alarm occurred, wait 15 minutes or more in the continuity state, and then turn the power ON again.

(Note 2) Immediately after alarm “75” occurs, the voltage across L+ and L- will be higher than the power voltage, so if the alarm is reset in this state, another alarm could occur.
Wait at least five minutes before resetting alarm “75”.

Alarm No.	Name	Meaning
8A	Detector converting unit 2: Communication error	An error was detected in the communication with the serial output linear scale of the unit MDS-B-HR in a servo system. In a spindle system, an error was detected in the communication with MDS-B-PJEX.
8B	Detector converting unit 2: Automatic tuning error	An abnormal signal was detected from PLG in automatic PLG tuning.
8C	Detector converting unit 2: Judgment error	The detector type outside the specification was designated in MDS-B-PJEX.
8D	Detector converting unit 2: CPU error	A CPU error was detected in the unit MDS-B-HR in a servo system, or in the unit MDS-B-PJEX in a spindle system.
8E	Detector converting unit 2: Data error	A data error was detected in the unit MDS-B-HR.

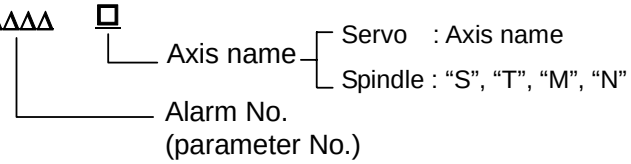
S02 INIT PARAM ERR ΔΔΔΔ □



An error was found in the parameters transmitted from the controller to the drive unit when the power was turned ON.
Remove the cause of the alarm, and then reset the alarm by turning the controller power OFF once.

Alarm No.	Details	Remedy
2201 – **2264**	The servo parameter setting data is illegal. The alarm No. is the No. of the servo parameter where the error occurred.	Check the descriptions for the appropriate servo parameters and correct them.
2301	The number of constants to be used in the following functions is too large: • Electronic gears • Position loop gain • Speed feedback conversion	Check that all the related parameters are specified correctly. sv001:PC1, sv002:PC2, sv003:PGN1 sv018:PIT, sv019:RNG1, sv020:RNG2
2302	High-speed serial incremental detector Parameters for absolute position detection are set to ON during OSE104 and OSE105 connection. Set the parameters for absolute position detection to OFF. To detect an absolute position, replace the incremental specification detector with an absolute position detector.	Check that all the related parameters are specified correctly. sv017:SPEC, sv025:MTYP
2303	No servo option is found. The closed loop (including the ball screw-end detector) or dual feedback control is an optional function.	Check that all the related parameters are specified correctly. sv025:MTYP/pen sv017:SPEC/dfbx
2304	No servo option is found. The SHG control is an optional function.	Check that all the related parameters are specified correctly. sv057:SHGC sv058:SHGCsp
2305	No servo option is found. The adaptive filtering is an optional function.	Check that all the related parameters are specified correctly. sv027:SSF1/aflt
3201-3584	The spindle parameter setting data is illegal. The alarm No. is the No. of the spindle parameter where the error occurred.	Check the descriptions for the appropriate spindle parameters and correct them. Refer to Spindle Drive Maintenance Manual.

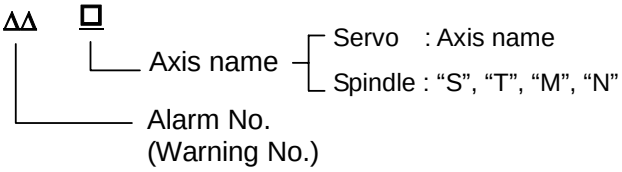
S51 PARAMETER ERROR ΔΔΔΔ □



A warning appears if a parameter set outside the tolerable range is set.
Illegal settings will be ignored.
This alarm will be reset when the correct value is set.

Alarm No.	Details	Remedy
2201 – 2264	Servo parameter setting data is illegal. The alarm No. is the No. of the servo parameter where the warning occurred.	Check the descriptions for the appropriate servo parameters and correct them.
3201-3584	Spindle parameter setting data is illegal. The alarm No. is the No. of the spindle parameter where the warning occurred.	Check the descriptions for the appropriate spindle parameters and correct them. Refer to Spindle Drive Maintenance Manual.

S52 SERVO WARNING 0 0 ΔΔ □



The drive unit warning is displayed.

Alarm No.	Name	Meaning
90	Detector: Initial communication error	Initial communication with the absolute position linear scale failed.
91	Detector: Communication error	An error was detected in the communication with the detector in absolute position detection system.
92	Detector: Protocol error	A data error was detected in absolute position detection system.
93	Initial absolute position fluctuation	The position data have fluctuated during the absolute position initializing.
96	Scale feedback error	An excessive deviation was detected between the motor end detector and MP scale feedback data in a MP scale absolute position detection system.
97	Scale offset error	An error was detected in the offset data received from the MP scale in a MP scale absolute position detection system.
9B	Detector converting unit: Magnetic pole shift warning	An error was detected in the shift distance of the magnetic pole in a linear servo system.
9C	Detector converting unit: Magnetic pole warning	A data error was detected in the magnetic pole of MDS-B-HR after passing Z-phase in a linear servo system.

Alarm No.	Name	Meaning
9E	Absolute position detector: Revolution counter error	An error was detected in the revolution counter of the absolute position detector. The absolute position data cannot be compensated.
9F	Battery voltage drop	The battery voltage that is supplied to the absolute position detector dropped. The absolute position data is retained.
A6	Fan stop warning	A cooling fan built in the drive unit stopped.
A8	Turret indexing warning	The designated position shift amount of turret indexing is outside the setting range.
A9	Orientation feedback warning	As an orientation feedback error occurred, the retrial has been conducted.
E0	Over regeneration warning	Over-regeneration detection level exceeded 80%.
E1	Overload warning	Overload detection level exceeded 80%.
E2	Continuous high-speed revolution warning	The motor was continuously rotated at a speed exceeding the rated speed.
E3	Absolute position counter warning	Deviation between the absolute and relative position data was detected.
E4	Set parameter warning	A parameter setting was outside the setting range.
E6	Control axis detachment warning	Control axis detachment was commanded.
E7	In NC emergency stop state	Emergency stop was input from the CNC.
E8	Excessive supplementary regeneration frequency	Regeneration that are beyond the power supply limitation has frequently occurred.
E9	Instantaneous power interruption warning	The power was momentarily interrupted.
EA	In external emergency stop state	External emergency stop signal was input.
EB	Over regeneration warning	Over-regeneration detection level exceeded 80%.

7.4 MCP Alarm (Yxxx)

An error has occurred in the drive unit other interfaces.
(The bold characters are the messages displayed on the screen.)

Y02 SYSTEM ALARM		An error occurred in the data transmitted between the MCP and amplifier after the power was turned on.
Error No.	**Details**	**Remedy**
0050	Background error	The software or hardware may be damaged. Contact the service center.
0051	0000 CRC error (10 times/910.2ms)	A communication error has occurred between the controller and drive unit. • Take measures against noise. • Check that the communication cable connector between the controller and amplifier and one between the drive unit are tight. • Check whether the communication cable between the controller and drive unit and one between the drive units are disconnected. • A driving drive unit may be faulty. Take a note of the 7-segment LED contents of each driving unit and report to the Service Center.
	0001 CRC error (2 continuous times)	
	0002 Reception timing error (2 continuous times)	
	××03 Data ID error (2 continuous times) ××: Axis No.	
	××04 No. of reception frames error (2 continuous times) ××: Axis No.	

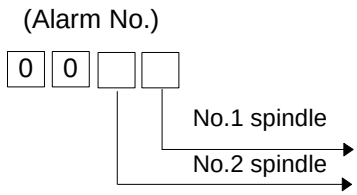
Y03 AMP. UNEQUIPPED	The drive unit is not correctly connected Check the drive unit mounting state. • Check the end of the cable wiring. • Check the cable for broken wires. • Check the connector insertion. The drive unit input power is not being input. The drive unit axis No. switch is illegal.
Error No.	**Details**
Alphabet (axis name)	Servo axis drive unit not mounted.
1 to 4	PLC axis drive unit not mounted.
S	No.1 spindle axis not mounted.
T	No.2 spindle axis not mounted.

Y10 Drv SW Differnt □□□□ **(Axis name)**	The software version of the drive unit connected with CNC differs from the version stored in the CNC as the drive unit software version. • Download the drive unit software whose version agrees with the version stored in CNC.

Y51 PARAMETER ERROR	An error occurred in a parameter that causes an alarm while the control axis was operating.

Error No.	Details	Remedy
1	LN FEED ABNL (Linear feed abnormal) The time constant has not been set or the setting exceeded the setting range.	• Check "#2004 G0tL".
2	CT FEED ABNL (Cutting feed abnormal) The time constant has not been set or the setting exceeded the setting range.	• Check "#2007 G1tL".
3	DLY F-F ABNL (Delayed fast feed abnormal) The time constant has not been set or the setting exceeded the setting range.	• Check "#2005 G0t1".
4	DLT CUTG ABNL (Delayed cutting feed abnormal) The time constant has not been set or the setting exceeded the setting range.	• Check "#2008 G1t1".
9	GRID SPACE ERROR	• Check "#2029 grspc".
12	SYNCHRONOUS TAP CYCLE ACCELERATION/DECELERATION TIME CONSTANT ERROR The time constant has not been set or the setting exceeded the setting range.	• Check spindle parameters "#3017 stapt1" to "#3020 stapt4".
14	SYNCHRONOUS CONTROL SLAVE AXIS NO. SETTING ILLEGAL The master axis and slave axis part systems differ.	• Check "#1068 slavno".
101	ROTARY AXIS GEAR RATIO EXCESSIVE (ABSOLUTE POSITION DETECTION)	• Check "#2201 PC1" and "#2202 PC2".

Y90 SP. NON SIGNAL



Alarm No.	Z open phase	B open phase	A open phase
1			×
2		×	
3		×	×
4	×		
5	×		×
6	×	×	
7	×	×	×

Alarm No.	Details	Remedy
0001 to 0007	There is an error in the spindle encoder signal. The data transmission to the servo drive unit and spindle drive unit is stopped when this error occurs.	• Check the spindle encoder's feedback cable and the encoder.

7.5 System Alarms (Zxxx)

The following messages are displayed with the register at the time when the error occurred if the system stops due to a system error.

Message	Details	Remedy
Parity error	RAM error	• Write down the displayed register, and contact the service center.
Bus error	A non-existing memory was accessed.	
Zero divide	The division with a 0 denominator was attempted.	
Watch dog error	The software process is not functioning correctly.	
Illegal exception	The alarm was caused by an illegal software function not listed above.	
Address error	An illegal memory was accessed.	
Illegal instruction	The software process is not functioning correctly.	
Stack overflow		

The following message appears when a communication error occurs. This is only a warning and has no effects on NC operations.

This message is erased by NC reset or turning NC power OFF to ON.

The communication error is not displayed if the “Communication typ” is “13” (Remote monitoring tool, Setup/Maintenance tool, CNC monitor by GOT).

The communication error of the “Communication typ:0” (API) is not stored as a log information.

Z30 ETHERNET ERROR □□□□ □□□□ (Error No.) (Communi-cation type)		A warning issued when a communication error occurs while Ethernet communication.
Error No.	**Error type**	**Details**
1	Socket open error	An error occurred on the connection to Ethernet I/F card.
2	Bind error	An error occurred at address assignments.
3	Listen error	An error occurred during transition of reception state for the connection request.
4	Accept error	An error occurred at response to connection request (server side).
5	Data receive error	An error occurred while receiving the data.
6	Data receive count error	The length of the received data is illegal.
7	Data send error	An error occurred while sending the data.
8	Connect error	An error occurred at connection request (serer side).
9	PC3JM time-out	There was no response from PC3JM for 5 or more seconds.

Following numbers indicate which communication the error occurred at/on.

Communication No.	**Connection type**
0	API
1	Port for LdCE (Port No.: 4096)
2	Port for MoCE (Port No.: 4097)
3	Port for operation board (Port No.: 4098)
4	Spare (Port No.: 4099)
5	Spare (Port No.: 4100)
6	Spare (Port No.: 4101)
7	Spare (Port No.: 4102)
8	Spare (Port No.: 4103)
9	MC protocol (TCP)
10	MC protocol (UDP)
11	At the communication by OPEN, BUFSND, BUFRCV commands. (TCP)
12	At the communication by OPEN, BUFSND, BUFRCV commands. (UDP)
13	Remote monitor tool, Setup/Maintenance tool, GOT (CNC monitor)

The bold characters are the messages displayed on the screen.

Message	Details	Remedy
Z52 BATTERY FAULT	The voltage of the battery inserted in the NC control unit has dropped. (The battery used to save the internal data)	• Replace the battery of the NC control unit. • After treating the battery, check the machining program,
Z53 TEMP. OVER	The controller or operation board temperature has risen above the designated value. **(Note 1)**	• Cooling measures are required. Turn off the controller power, or lower the temperature with a cooler, etc.
Z55 RIO NOT CONNECT	This occurs when an error occurs in the communication between the controller and remote I/O unit. • Cable breakage • Remote I/O unit fault • Power supply to remote I/O unit fault **(Note 2)**	• Check and replace the cables. • Replace the remote I/O unit. • Check the power supply. (existence of supply, voltage)
Z59 TIME CONSTANT	Acceleration and deceleration time constants are too large. (This alarm is output at the same time as "T02 FEED HOLD 0206.")	• Increase the value specified as the "#1206 G1bF" parameter. • Decrease the value specified as the "#1207 G1btL" parameter. • Lower the cutting speed.

⚠ CAUTION

- ❗ If the battery low warning is issued, save the machining programs, tool data and parameters in an input/output device, and then replace the battery. When the battery alarm is issued, the machining programs, tool data and parameters may be destroyed. Reload the data after replacing the battery.
- ⚠ Do not replace the battery while the power is ON.
- ⚠ Do not short circuit, charge, heat, incinerate or disassemble the battery.
- ⚠ Dispose of the spent battery following local laws.

(Note 1) Temperature warning
If the alarm is displayed when an overheat alarm is detected, the overheat signal will be output simultaneously. If the machine is in automatic operation, the operation will be continued, but restarting will not be possible after resetting or stopping with M02/M30. (Starting will be possible after block stop or feed hold.) The alarm will be reset and the overheat signal will turn off when the temperature drops below the specified temperature.

Z53 Overheat 0 0 0 □

- 1 : The temperature in the controller is high.
- 2 : The temperature around the communication terminal is high.
- 3 : Both temperatures in the controller and that around the communication terminal are high.

The ambient temperature must be lowered immediately when an “Overheat” alarm occurs, but if machining must be continued, the alarm can be invalidated by turning the following parameter off.

PLC parameter bit selection #6449

7 6 5 4 3 2 1 0
□□□□□□□□

- Communication terminal
- Controller

{ 0: Detection valid
1: Detection invalid

(Note 2) RIO communication interrupt

If communication between the control unit and remote I/O unit fails, the alarm and remote I/O unit number are displayed.

(Control unit side connection) (Board side connection)

Z55 RIO NOT CONNECT 0 0 □ □ 0 0 □ □

Hexadecimal notation

Alarm number	RIO (seventh station)	RIO (sixth station)	RIO (fifth station)	RIO (fourth station)
0				
1				×
2			×	
3			×	×
4		×		
5		×		×
6		×	×	
7		×	×	×
8	×			
9	×			×
A	×		×	
B	×		×	×
C	×	×		
D	×	×		×
E	×	×	×	
F	×	×	×	×

Alarm number	RIO (third station)	RIO (second station)	RIO (first station)	RIO (0th station)
0				
1				×
2			×	
3			×	×
4		×		
5		×		×
6		×	×	
7		×	×	×
8	×			
9	×			×
A	×		×	
B	×		×	×
C	×	×		
D	×	×		×
E	×	×	×	
F	×	×	×	×

7.6 Absolute Position Detection System Alarms (Z7x)

Z70 ABS. ILLEGAL □□□□ □□□□ (Error No.) (Axis name)	This error is displayed if the absolute position data is lost in the absolute position detection system.				
Error No.	**Details**	**Remedy**	**Zero point initializa-tion**	**Alarm reset when power is turned off**	**Servo alarm No.**
0001	Zero point initialization is incomplete. Otherwise, the spindle was removed.	Complete zero point initialization.	Required	–	–
0002	The absolute position reference point data saved in the NC has been destroyed.	Input the parameters. If the reference point data cannot be restored, perform zero point initialization.	(Required)	–	–
0003	The parameters used to detect the absolute position have been changed. #1003 iunit #2201 PC1 #1016 iout #2202 PC2 #1017 rot #2218 PIT #1018 ccw #2219 RNG1 #1040 M_inch #2220 RNG2 #2049 type #2225 MTYP	Correctly set the parameters. Turn the power on again, and perform zero point initialization.	Required	–	–
0004	The zero point initialization point is not at the grid position.	Reperform zero point initialization.	Required	–	–
0005	Restoration was possible with parameter input in the above No.0002 state.	Turn the power on again, and operation will be possible.	Not required	–	–
0101	The power was turned on again after the servo alarm No.25 displayed.	Reperform zero point initialization.	Required	–	(25)
0106	The power was turned on again after the servo alarm No.E3 displayed.	Reperform zero point initialization.	Required	–	(E3)

(Note) To release alarm "Z70 ABS. ILLEGAL," enter the parameter data output when establishing the absolute position and turn on the power again. For the rotation axis, however, the alarm cannot be released by entering the parameter data.

Z71 DETECTOR ERROR □□□□ □□□□ (Error No.) (Axis name)	This alarm is displayed if an error is found in the detector for the absolute position detection system.

Error No.	Details	Remedy	Zero point initialization	Alarm reset when power is turned off	Servo alarm No.
0001	The backup voltage in the absolute position detector dropped.	Replace the battery, check the cable connections, and check the detector. Turn the power on again, and perform zero point initialization.	Required	– (Z70-0101 displays after power is turned on again.)	25
0003	Communication with the absolute position detector was not possible during operation.	Check and replace the cables, card or detector. Turn the power on again, and perform zero point initialization.	(Required) Only when detector is replaced.	Reset	91
0004	The absolute position data fluctuated when establishing the absolute position.	Check and replace the cables, card or detector. Turn the power on again, and perform zero point initialization.	(Required) Only when detector is replaced.	Reset	93
0005	An error was found in the serial data from the absolute position detector.	Check and replace the cables, card or detector. Turn the power on again, and perform zero point initialization.	(Required) Only when detector is replaced.	Reset	92
0006	Servo alarm E3 Absolute position counter warning	Operation is possible until the power is turned off.	(Required) When power is turned on again.	Reset (Z70-0106 displays after power is turned on again.)	E3
0007	Initial communication with the absolute position detector was not possible.	Check and replace the cables, card or detector. Turn the power on again, and perform zero point initialization.	(Required) Only when detector is replaced.	Reset	18

Z72 COMPARE ERROR □□□□ □□□□ (Alarm No.) (Axis name)		This alarm is displayed if an error is detected when comparing the detector's absolute position and controller coordinate values in the absolute position system.
Alarm No.	**Details**	**Remedy**
0001	An error was found when confirming the position during execution of G28 or G30.	• The alarm will be reset when the parameter "absg28" or "absm02" is increased or set to 0. Set a value larger than the in-position width (double the value set in SV014) for each parameter. Turn the power ON again after this alarm occurs.
0002	An error was found when confirming the position during execution of M02 or M30.	

Z73 ABS. WARNING □□□□ □□□□ (Warning No.) (Axis name)		This displays a warning in the absolute position detection system.
Alarm No.	**Details**	**Remedy**
0001	Servo alarm 9F Battery voltage drop	The battery voltage dropped or a cable is broken. Absolute position initialization is not required.

7.7 Messages During Emergency Stop (EMG)

Error Items	Details	Remedy
PLC	The user PLC has entered the emergency stop state during the sequence process.	• Investigate and remove the cause of the user PLC emergency stop.
EXIN	The emergency stop input signal is significant (open).	• Cancel the emergency stop input signal. • Check the wiring to see if any wiring is missing.
SRV	An alarm occurred in the servo system causing an emergency stop.	• Investigate and remove the cause of the servo alarm.
STOP	The user PLC (ladder sequence) is not running.	• Check if the rotary switch:CS1 (left switch) on the top of the controller front panel is set to 1. • Check if the PLC edit file save screen (onboard function) [4RUN/SP] (run/stop) switch is turned ON.
SPIN	Spindle drive unit not mounted. The spindle drive unit is not mounted.	• Cancel the causes of the other emergency stop. • Check emergency stop signal and ready signal input in the spindle drive unit.
PC_H	High-speed PC processing abnormal	• Check the sequence program. (To stop monitoring the high-speed PC processing temporarily, set "1" in #1219 aux03 bit1. Disable the monitoring function only as a temporary measure.)
PARA	Setting of the door open II fixed device is illegal.	• Specify the "#1155 DOOR_m" and "#1156 DOOR_s" parameters correctly. (When the door open II fixed device is not used, set "#1155 DOOR_m" and "#1156 DOOR_s" to "100".)
	The dog signal random assignment parameter setting is illegal.	• Correctly set the "#2073 zrn_dog", "#2074 h/w_ot+", "#2075 h/w_ot-" and "#1226 aux10 bit 5" parameters.

Error Items	Details	Remedy
XTEN	External PLC H/W is not found. The external PLC card is not mounted.	• Check that the external PLC card is mounted. • Change the external PLC card.
WAIT	External PLC is not ready. The External PLC Ready signal is not turned ON.	• Reset and start the external PLC. • Turn the NC power OFF and then ON. • Check that the external PLC card is mounted. • Change the external PLC card. • Change the NC card.
LINK	External PLC communication error occurred. A communication error occurred between the external PLC and NC.	
LAD	The user PLC (ladder sequence) has an illegal code.	• Check the user PLC (ladder sequence) to see if it uses illegal device numbers or constants.

*When the “Error Items” is “LINK”, the cause of the error will be saved into the R register as follows.

R register	Details	
R80	The contents of the latest errors. (Cleared when the error is resolved.) Bit 0 : Emergency B Bit 1 : Emergency C Bit 2 : Emergency D Bit 3 : Emergency E Bit 4 : Emergency F	• The external PLC is not ready. • Data is not input from the external PLC. • Data is not read out by the external PLC. • Data is not written into the NC correctly. • 2-port RAM is not cleared when the PC_OK signal is turned ON
R81	The contents of the latest errors. (Retained until the NC power is turned ON next.)	• This is cleared when the NC power is turned ON again.
R82	The control signal. External PLC -> NC	• Writing from PC to NC completion flag
R83	The control signal. NC -> External PLC	• Writing from NC to PC completion flag.
R84	The contents of the latest errors.	• This is not cleared even if the NC power is turned ON again.

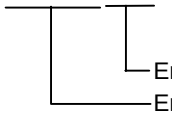
7.8 Communication Error

L10 DN INIT. ERR. 1	Error detected when initializing DeviceNet unit (Type 1: Configuring with SyCon2)

Error No.	Details	Remedy
0035	The baud rate setting is not within the valid range.	• Correctly set the baud rate.
0036	The local station No. (MAC ID) value is not within the valid range.	• Set the local station No. between 0 and 63.
0039	There are two or more stations with the same station No. (MAC ID) in the network.	• Set the station numbers so that they are not duplicated.
00D2	The parameters are not set in the communication PCB's flash ROM.	• This is not a particular problem when using the EEPROM parameters.

L11 DN INIT. ERR. 2	Error detected when initializing DeviceNet unit (Type 2: Configuring with PLC program)

Error No.	Details	Remedy
0001	The local station No. (MAC ID) value is not within the range.	• Set the local station No. within 0000H to 003FH, or to FFFFH.
0002	The baud rate is not within the valid range.	• Set a value between 1 and 3.
0003	The slave station No.'s low-order byte is not within the valid range.	• Set a value between 0 and 63.
0004	The slave station No.'s high-order byte is not within the valid range.	• Set 01H or 80H.
0005	The connection type is not within the valid range.	• Set 0001H, 0002H, 0004H or 0008H.
0006	A slave station with the same station No. as the local station No. is already set.	• Set the slave station Nos. so that they are not duplicated within all stations.
0007	No slave station is set.	• Set at least one slave station.
0008	The total input data length for all slave stations is too long.	• The length must be 256 bytes or less for all slave stations.
0009	The total output data length for all slave stations is too long.	• The length must be 256 bytes or less for all slave stations.
000A	The parameter watchdog timeout action value is illegal.	• Set 0000H, 0001H, 0002H or 0003H.
000B	The expected packet rate value is smaller than the production inhibit time value.	• Set so that the expected packet rate value is greater than or equal to the production inhibit time value.
000C	EEPROM check sum error	• Write the parameters again. • Do not turn the power OFF or reset the system while writing the parameters.

L12 DN LINK ERROR □□□□ □□  	Error detected during DeviceNet communication process

Error No.	Details	Remedy
0001	Network trouble was detected after communication started.	• Check that the cable is connected correctly.
001E	The slave did not respond.	• Generally check the state of the network and slave, and check that the MAC ID and baud rate are correct, that the slave is not down, and that the terminator is not disconnected, etc.
0020	The slave responded with an unspecified error.	• Read the communication error information, read the error information, and take appropriate measures for that error.
0023	The slave responded with an error when establishing the connection.	• Read the communication error information, read the error information, and take appropriate measures for that error.
0024	The parameter input data size and actual slave size do not match.	• Check the slave manual and set the correct input data size.
0025	The parameter output data size and actual slave size do not match.	• Check the slave manual and set the correct output data size.
0026	Response data for a function not supported by the HR871 card was received.	• Check the slave manual, and set so that functions not supported by HR871 are not sent. • Generally check the state of the network and slave, and confirm that the terminator is not disconnected, etc.
0027	The connection is already in the designated mode.	• Generally check the state of the network and slave, and confirm that the terminator is not disconnected, etc.
0028	Unpredicted illegal data was received when establishing the connection.	• Generally check the state of the network and slave, and confirm that the terminator is not disconnected, etc.
0029	A connection is already established with that slave.	• Check the state for a while, and if the connection cannot be established, reset the slave.
002A	The poling response data length differs from the data length read from the slave when the connection was established.	• Generally check the state of the network and slave, and confirm that the terminator is not disconnected, etc.
002B	When receiving a split poling response, the first split data was received twice.	• Generally check the state of the network and slave, and confirm that the terminator is not disconnected, etc.

Error No.	Details	Remedy
002C	When receiving a split poling response, the received split data No. differed from the one available.	• Generally check the state of the network and slave, and confirm that the terminator is not disconnected, etc.
002D	When receiving a split poling response, the middle data or final data was received before receiving the first split data.	• Generally check the state of the network and slave, and confirm that the terminator is not disconnected, etc.
003B	The same station No. (MAC ID) was detected two or more times in the parameters.	• There are two or more slaves with the same station No. in the parameters. Correct the station numbers. • A slave with the same station No. as the local station No. was found in the parameters.
0045	O.Addr in the parameters exceeds 255.	• O.Addr to 255 or less.
0046	I.Addr in the parameters exceeds 255.	• I.Addr to 255 or less.
0047	An illegal connection type was designated.	• Confirm that the connection type value is correct.
0049	The expected packet rate value is smaller than the production inhibit time value.	• Set the expected packet rate value higher than the production inhibit time value.

L13 DN MESSAGE ERR. □□□□ (Error No.)	Error detected while executing message communication

Error No.	Details	Remedy
0002	The resources required for executing the required service could not be used.	• Referring to the slave manual, check the conditions for the slave to notify this error, and remedy accordingly.
0008	The requested service is not mounted or is not defined for this object class or instance.	• Check that the designated MAC ID, class ID, instance ID and attribute ID are correct. • Referring to the slave manual, check the conditions for the slave to notify this error, and remedy accordingly.
0009	Invalid attribute data was detected.	• Check that the designated MAC ID, class ID, instance ID and attribute ID are correct. • Referring to the slave manual, check the conditions for the slave to notify this error, and remedy accordingly.

Error No.	Details	Remedy
000B	The object is already in the mode or state requested by the service.	• Check that the designated MAC ID, class ID, instance ID and attribute ID are correct. • Check the current status using attribute read. • Referring to the slave manual, check the conditions for the slave to notify this error, and remedy accordingly.
000C	The object cannot execute the requested service in the current mode or state.	• Check that the designated MAC ID, class ID, instance ID and attribute ID are correct. • Check the current status using attribute read. • Referring to the slave manual, check the conditions for the slave to notify this error, and remedy accordingly.
000E	A request to change a change prohibited attribute was received.	• Check that the designated MAC ID, class ID, instance ID and attribute ID are correct. • Referring to the slave manual, check the conditions for the slave to notify this error, and remedy accordingly.
000F	The enable/special rights check failed.	• Check that the designated MAC ID, class ID, instance ID and attribute ID are correct. • Referring to the slave manual, check the conditions for the slave to notify this error, and remedy accordingly.
0010	The requested service cannot be executed in the current device state.	• Check that the designated MAC ID, class ID, instance ID and attribute ID are correct. • Referring to the slave manual, check the conditions for the slave to notify this error, and remedy accordingly.
0011	The slave did not respond.	• Generally check the state of the network and slave, and check that the slave is not down, and that the terminator is not disconnected, etc.
0013	Sufficient data to execute the designated operation has not been provided.	• Check that the designated MAC ID, class ID, instance ID and attribute ID are correct. • When executing attribute write, check that the designated data is not insufficient, and that the data length is correct. • Referring to the slave manual, check the conditions for the slave to notify this error, and remedy accordingly.

Error No.	Details	Remedy
0014	The designated attributes are not supported.	• Check that the designated MAC ID, class ID, instance ID and attribute ID are correct. • Referring to the slave manual, check the conditions for the slave to notify this error, and remedy accordingly.
0015	The service provided an unexpected volume of data.	• The data returned by the slave must be 240 bytes or less.
0016	The designated object does not exist in the slave.	• Check that the designated MAC ID, class ID, instance ID and attribute ID are correct. • Referring to the slave manual, check the conditions for the slave to notify this error, and remedy accordingly.
0032	The response data format is illegal.	• Generally check the state of the network and slave, and confirm that the terminator is not disconnected, etc.
0037	The designated slave station No. is not within 0 to 63.	• Designate a value between 0 and 63.
0039	The split response order is illegal.	• Generally check the state of the network and slave, and confirm that the terminator is not disconnected, etc.
00C8	The parameters are not set for the designated slave.	• Designate a slave for which the parameters have been set.
0101	The set data length exceeds 241.	• The data length must be 240 or less.
0102	An illegal value was set for the command No. in the message communication command area.	• Set 0001H, 0101H, 0102H or 0201H for the command No.

7.9 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.

7.9.1 Operation Errors (Exxx)

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 [↓] or [↑] 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. (When special relay E71 is valid by the PLC.) • 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 PLC was not stopped. • Analog output adjustment was attempted when the PLC was not stopped. • An attempt was made to input or output language data during PLC execution. (Measures) • Stop the PLC. • Set the control unit rotary switch to 1. • Set the onboard file screen RUN/STOP setting 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.
E165	**AUX RUNNING**		• The keys other than Function/Menu/Previous page/Next page were pressed in Auxiliary monitor screen during auxiliary axis operation.
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.

7.9.2 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) S-analog output adjustment related

Message	Message details
ADJUST EXECUTION	• Analog output adjustment is being executed without error.
ADJUST COMPLETE	• Analog output adjustment has completed without error.

(5) Auxiliary axis related

Message	Message details
CONTINUE Y/N	• Type Y or N to specify whether to perform operation.
BACKUP EXECUTION	• The auxiliary axis parameters are being backed up in SRAM.
BACKUP COMPLETE	• The backup of the auxiliary axis parameters in SRAM has been completed.
AUX. WRITING EXEC.	• The auxiliary axis parameters in SRAM is being written to MR-J2-CT.
WRITE COMPLETE	• The writing of the auxiliary axis parameters in SRAM to MR-J2-CT has been completed.
ABS POS RESTORED	• The absolute position in SRAM has been restored in MR-J2-CT.

(6) 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.

7.9.3 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 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.
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 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.

7.10 Program Error (Pxxx)

(The bold characters are the message displayed on the screen.)
These alarms occur during automatic operation, and the causes of these alarms are mainly program errors which occur, for instance, when mistakes have been made in the preparation of the machining programs or when programs which conform to the specification have not been prepared.

Error No.	Details	Remedy
P 10	**EXCS. AXIS. No.** The number of axis addresses commanded in the same block exceeds the specifications.	• Divide the alarm block command into two. • Check the specifications.
P 11	**AXIS ADR. ERROR** The axis address commanded by the program and the axis address set by the parameter do not match.	• Revise the axis names in the program.
P 20	**DIVISION ERROR** An axis command which cannot be divided by the command unit has been issued.	• Review the program.
P 30	**PARITY H** The number of holes per character on the paper tape is odd.	• Check the paper tape. • Check the tape puncher and tape reader.
P 31	**PARITY V** The number of characters per block on the paper tape is odd.	• Make the number of characters per block on the paper tape even. • Set the parameter parity V selection off.
P 32	**ADDRESS. ERROR** An address not listed in the specifications has been used.	• Check and revise the program address. • Check the specifications.
P 33	**FORMAT ERROR** The command format in the program is not correct.	• Review the program.
P 34	**G-CODE ERROR** A G code not listed in the specifications has been used.	• Check and correct the G code address in the program.
P 35	**CMD-VALUE OVER** The setting range for the addresses has been exceeded.	• Review the program.
P 36	**PROGRAM END ERR** "EOR" has been read during tape and memory operation.	• Enter the M02 and M30 command at the end of the program. • Enter the M99 command at the end of the subprogram.
P 37	**PROG. No. ZERO** A zero has been specified for program and sequence numbers.	• The program numbers are designated across a range from 1 to 99999999. • The sequence numbers are designated across a range from 1 to 99999.
P 39	**NO SPEC ERR** A non-specified G code was specified.	• Check the specifications.

Error No.	Details	Remedy
P 40	**PREREAD BL. ERR** When cutter compensation is executed, there is an error in the pre-read block and so the interference check is disabled.	• Review the program.
P 60	**OVER CMP. LENG.** The commanded movement distance is excessive. (Over 2^{31})	• Review the axis address command.
P 62	**F-CMD. NOTHING** No feed rate command has been issued.	• 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.
P 65	**NO G05P3 SPEC**	• Check the high-speed mode III specifications.
P 70	**ARC ERROR** There is an error in the arc start and end points as well as in the arc center.	• Check the numerical values of the addresses that specify the start and end points as well as the arc center in the program. • Check the "+" and "–" directions of the address numerical values.
P 71	**ARC CENTER** The arc center is not sought during R-specified circular interpolation.	• Check the numerical values of the addresses in the program.
P 72	**NO HELICAL SPEC** A helical command has been issued though it is not included in the specifications.	• Check the helical specifications. • An Axis 3 command was issued by the circular interpolation command. If there is no helical specification, the linear axis is moved to the next block.
P 90	**NO THREAD SPEC** A thread cutting command has been issued though it is not included in the specifications.	• Check the specifications.
P 93	**SCREW PITCH ERR** The screw pitch has not been set correctly when the thread cutting command is issued.	• Issue the thread cutting command and then set the screw pitch command properly.
P111	**PLANE CHG (CR)** Plane selection commands (G17, G18, and G19) were issued when a coordinate rotation command (G68) was issued.	• Before issuing the plane selection commands, issue G68 and then G69 (coordinate rotation cancel).

Error No.	Details	Remedy
P112	**PLANE CHG (CC)** • A plane selection command (G17, G18, G19) has been issued when the cutter compensation command (G41, G42) or nose R compensation command (G41, G42, G46) is issued. • The plane selection command was issued when nose radius compensation is completed, there is no axial movement command after the G40 command, and the compensation has not been canceled.	• Issue the plane selection command after the cutter compensation command or nose radius compensation command has been canceled (issue axial movement command after the G40 cancel command).
P113	**ILLEGAL PLANE** The arc command axis is not on the selected plane.	• Issue arc command on the correctly selected plane.
P122	**NO AUTO C-OVR** An automatic corner override command (G62) has been issued though it is not included in the specifications.	• Check the specifications. • Delete the G62 command from the program.
P130	**2nd AUX. ADDR** The 2nd miscellaneous function address specified in the program does not match that set by the parameter.	• Check and correct the 2nd miscellaneous function address in the program.
P131	**NO G96 SPEC** (No constant peripheral speed) The constant peripheral speed command (G96) was issued despite the fact that such a command does not exist in the specifications.	• Check the specifications. • Change from the constant peripheral speed command (G96) to the rotation speed command (G97).
P132	**SPINDLE S = 0** No spindle speed command has been specified.	• Review the program.
P133	**G96 P-No. ERR** An invalid constant peripheral speed control axis has been specified.	• Review the parameter specified for the constant peripheral speed control axis.
P150	**NO C-CMP SPEC** • Even though there were no tool radius compensation specifications, tool radius compensation commands (G41 and G42) were issued. • Even though there were no nose R compensation specifications, nose R compensation commands (G41, G42, and G46) were issued.	• Check the specifications.
P151	**G2, 3 CMP. ERR** A compensation command (G40, G41, G42, G43, G44, G46) has been issued in the arc mode (G02, G03).	• Issue the linear command (G01) or rapid traverse command (G00) in the compensation command block or cancel block. (Set the modal to linear interpolation.)

Error No.	Details	Remedy
P152	**I.S.P NOTHING** In interference block processing during execution of a tool radius compensation (G41 or G42) or nose radius compensation (G41, G42, or G46) command, the intersection point after one block is skipped cannot be determined.	• Review the program.
P153	**I.F ERROR** An interference error has arisen while the tool radius compensation command (G41, G42) or nose R compensation command (G41, G42, G46) was being executed.	• Review the program.
P155	**F-CYC ERR (CC)** A canned cycle command has been issued in the tool radius compensation mode.	• The tool radius compensation mode is established when a canned cycle command is executed and so the tool radius compensation cancel command (G40) should be issued.
P156	**BOUND DIRECT** At the start of G46 nose radius compensation, the compensation direction is undefined if this shift vector is used.	• Change the vector to that with which the compensation direction is defined. • Exchange with a tool having a different tip point number.
P157	**SIDE REVERSED** During G46 nose R compensation, the compensation direction is inverted.	• Change the G command to that which allows inversion of the compensation direction (G00, G28, G30, G33, or G53). • Exchange with a tool having a different tip point number. • Turn on the G46 inversion error avoidance parameter.
P158	**ILLEGAL TIP P.** During G46 nose R compensation, the tip point is illegal (other than 1 to 8).	• Change the tip point number to a legal one.
P170	**NO CORR. NO.** The compensation number (DOO, TOO, HOO) command was not given when the tool radius compensation (G41, G42, G43, G46) command was issued. Alternatively, the compensation number is larger than the number of sets in the specifications.	• Add the compensation number command to the compensation command block. • Check the number of compensation number sets and correct it to a compensation number command within the permitted number of compensation sets.
P172	**G10 L-No. ERR** (G10 L-number error) The L address command is not correct when the G10 command is issued.	• Check the address L-Number of the G10 command and correct the number.
P173	**G10 P-No. ERR** (G10 compensation error) When the G10 command is issued, a compensation number outside the permitted number of sets in the specifications has been commanded for the compensation number command.	• First check the number of compensation sets and then set the address P designation to within the permitted number of sets.

Error No.	Details	Remedy
P177	**LIFE COUNT ACT** Registration of tool life management data with G10 was attempted when the used data count valid signal was ON.	• The tool life management data cannot be registered when counting the used data. Turn the used data count valid signal OFF.
P178	**LIFE DATA OVER** The No. of registration groups, total No. of registered tools or the No. of registrations per group exceeded the specifications range.	• Review the No. of registrations.
P179	**GROUP NO. ILL.** • When registering the tool life management data with G10, the group No. was commanded in duplicate. • A group No. that was not registered was designated during the T□□□□99 command. • An M code command must be issued as a single command but 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.	• The group No. cannot be commanded in duplicate. When registering the group data, register it in group units. • Correct to the correct group No.
P180	**NO BORING CYC.** A canned cycle command was issued though there are not canned cycle (G72 – G89) specifications.	• Check the specifications. • Correct the program.
P181	**NO S-CMD (TAP)** The spindle speed command has not been issued when the tapping canned cycle command is given.	• Issue the spindle speed command (S) when the tapping canned cycle command G84, G74 (G84, G88) is given.
P182	**SYN TAP ERROR** Connection to the main spindle unit was not established.	• Check connection to the main spindle. • Check that the main spindle encoder exists.
P183	**PTC/THD No.** The pitch or thread number command has not been issued in the tap cycle of a boring canned cycle command.	• Specify the pitch data and the number of threads by F or E command.
P184	**NO PTC/THD CMD** The pitch or the number of threads per inch is illegal in the tap cycle of the drilling fixed cycle command.	• Check the pitch or the number of threads per inch.
P190	**NO CUTTING CYC** A lathe cutting cycle command was input although the lathe cutting cycle was undefined in the specification.	• Check the specification. • Delete the lathe cutting cycle command.

Error No.	Details	Remedy
P191	**TAPER LENG ERR** In the lathe cutting cycle, the specified length of taper section is illegal.	• The radius set value in the lathe cycle command must be smaller than the axis shift amount.
P192	**CHAMFERING ERR** Chamfering in the thread cutting cycle is illegal.	• Set a chamfering amount not exceeding the cycle.
P200	**NO MRC CYC SPC** A multiple fixed-cycle I command (G70 to G73) was input although this cycle was undefined in the specification.	• Check the specification.
P201	**PROG. ERR (MRC)** • When called with a multiple fixed-cycle I command, the subprogram contained at least one of the following commands: Reference point return command (G27, G28, G30) Thread cutting (G33) Fixed-cycle skip-function (G31) • The first move block of the finish shape program in multiple fixed cycle I contains an arc command.	• Delete the following G codes from this subprogram that is called with the multiple fixed cycle I commands (G70 to G73): G27, G28, G30, G31, G33, fixed-cycle G-code. • Remove G2 and G3 from the first move block of the finish shape program in multiple fixed cycle I.
P202	**BLOCK OVR (MRC)** The number of blocks in the shape program of the multiple fixed cycle I is over 50.	• Specify 50 or a less value. The number of blocks in the shape program called by the fixed cycle I commands (G70 to G73) must be decreased below 50.
P203	**CONF. ERR (MRC)** The multiple fixed-cycle I (G70 to G73) shape program could not cut the work normally because it defined an abnormal shape.	• Review the multiple fixed-cycle I (G70 to G73) shape program.
P204	**VALUE ERR (MRC)** A command value of the multiple fixed-cycle I (G70 to G76) is illegal.	• Review the multiple fixed-cycle (G70 to G76) command value.
P210	**NO PAT CYC SPC** A multiple fixed-cycle II (G74 to G76) command was input although it was undefined in the specification.	• Check the specification.
P220	**NO SPECIAL CYC** No special fixed cycle specifications are available.	• Check the specifications.
P221	**NO HOLE (S-CYC)** A 0 has been specified for the number of holes in special fixed cycle mode.	• Review the program.
P222	**G36 ANGLE ERR** A G36 command specifies 0 for angle intervals.	• Review the program.

Error No.	Details	Remedy
P223	**G12 G13 R ERR** The radius value specified with a G12 or G13 command is below the compensation amount.	• Review the program.
P224	**NO G12, G13 SPC** There are no circular cutting specifications.	• Check the specifications.
P230	**NESTING OVER** A subprogram has been called 8 or more times in succession from the subprogram.	• Check the number of subprogram calls and correct the program so that it does not exceed 8 times.
P231	**NO N-NUMBER** At subprogram call time, the sequence number set at return from the subprogram or specified by GOTO, was not set.	• Specify the sequence numbers in the call block of the subprogram.
P232	**NO PROGRAM No.** The subprogram has not been set when the subprogram is called.	• Enter the subprogram.
P241	**NO VARI NUMBER** The variable number commanded is higher than the numbers in the specifications.	• Check the specifications. • Check the program variable number.
P242	**EQL. SYM. MSSG.** The “=” sign has not been commanded when a variable is defined.	• Designate the “=” sign in the variable definition of the program.
P243	**VARIABLE ERR.** An invalid variable has been specified in the left or right side of an operation expression.	• Correct the program.
P260	**NO COOD-RT SPC** Even though there were no coordinate rotation specifications, a coordinate rotation command was issued.	• Check the specifications.
P270	**NO MACRO SPEC** A macro specification was commanded though there are no such command specifications.	• Check the specifications.
P271	**NO MACRO INT.** A macro interrupt command has been issued though it is not included in the specifications.	• Check the specifications.
P272	**MACRO ILL.** A statement and a macro statement exist together in the same block.	• Review the program and place the executable statement and macro statement in separate blocks.
P273	**MACRO OVERCALL** The frequency of the macro call has exceeded the limit.	• Review the program and correct it so that the macro calls do not exceed the limit imposed by the specification.

Error No.	Details	Remedy
P275	**MACRO ARG. EX.** The number of macro call argument type II sets has exceeded the limit.	• Review the program.
P276	**CALL CANCEL** A G67 command was issued though it was not during the G66 command modal.	• Review the program. • The G67 command is the call cancel command and so the G66 command must be designated first before it is issued.
P277	**MACRO ALM MESG** An alarm command has been issued in #3000.	• Refer to the operator messages on the DIAG screen. • Refer to the instruction manual issued by the machine manufacturer.
P280	**EXC. [,]** The number of parentheses [,] which can be commanded in a single block has exceeded five.	• Review the program and correct it so the number of "[" or "]" does not exceed five.
P281	**[,] ILLEGAL** The number of "[" and "]" parentheses commanded in a single block does not match.	• Review the program and correct it so that "[" and "]" parentheses are paired up properly.
P282	**CALC. IMPOSS.** The arithmetic formula is incorrect.	• Review the program and correct the formula.
P283	**DIVIDE BY ZERO** The denominator of the division is zero.	• Review the program and correct it so that the denominator for division in the formula is not zero.
P290	**IF SNT. ERROR** There is an error in the IF conditional GOTO□ statement.	• Review the program.
P291	**WHILE SNT. ERR** There is an error in the WHILE conditional DO□-END□ statement.	• Review the program.
P292	**SETVN SNT. ERR** There is an error in the SETVN □ statement when the variable name setting was made.	• Review the program. • The number of characters in the variable name of the SETVN statement must be 7 or less.
P293	**DO-END EXCESS** The number of □'s for DO-END□ in the WHILE conditional DO□ – END□ statement has exceed 27.	• Review the program and correct it so that the number of 's in the DO – END statement does not exceed 27.
P294	**DO-END MMC.** The DO's and END's are not paired off properly.	• Review the program and correct it so that the DO's and END's are paired off properly.
P295	**WHILE/GOTO TPE** There is a WHILE or GOTO statement on the tape during tape operation.	• During tape operation, a program which includes a WHILE or GOTO statement cannot be executed and so the memory operation mode is established instead.
P296	**NO ADR (MACRO)** A required address has not been specified in the user macro.	• Review the program.

Error No.	Details	Remedy
P297	**ADR-A ERR.** The user macro does not use address A as a variable.	• Review the program.
P298	**PTR OP (MACRO)** User macro G200, G201, or G202 was specified during tape or MDI operation.	• Review the program.
P300	**VAR. NAME ERROR** The variable names have not been commanded properly.	• Review the variable names in the program and correct them.
P301	**VAR. NAME DUPLI** The name of the variable has been duplicated.	• Correct the program so that the name is not duplicate.
P360	**NO PROG.MIRR.** A mirror image (G50.1 or G51.1) command has been issued though the programmable mirror image specifications are not provided.	• Check the specifications.
P380	**NO CORNER R/C** A command was issued for corner rounding or corner chamfering though there are no such specifications.	• Check the specifications. • Remove the corner rounding or chamfering command from the program.
P381	**NO ARC R/C SPC** Corner rounding or chamfering was specified in the arc interpolation block although corner chamfering/corner rounding II is unsupported.	• Check the specifications.
P382	**CORNER NO MOVE** The block next to corner rounding/ chamfering is not a movement command.	• Replace the block succeeding the corner rounding/chamfering command by G01 command.
P383	**CORNER SHORT** In the corner rounding or chamfering command, the movement distance was shorter than the value in the corresponding command.	• Make the corner rounding or chamfering less than the movement distance since this distance is shorter than the corner rounding or chamfering.
P384	**CORNER SHORT** When the corner rounding or chamfering command was input, the movement distance in the following block was shorter than the length of the corner rounding or chamfering.	• Make the corner rounding or chamfering less than the movement distance since this distance in the following block is shorter than the corner rounding or chamfering.
P385	**G0 G33 IN CONR** A block with corner rounding/chamfering was given during G00 or G33 modal.	• Review the program.
P390	**NO GEOMETRIC** A geometric command was issued though there are no geometric specifications.	• Check the specifications.

Error No.	Details	Remedy
P391	**NO GEOMETRIC 2** There are no geometric IB specifications.	• Check the specifications.
P392	**LES AGL (GEOMT)** The angular difference between the geometric line and line is 1° or less.	• Correct the geometric angle.
P393	**INC ERR (GEOMT)** The second geometric block was specified by an incremental value.	• Specify this block by an absolute value.
P394	**NO G01 (GEOMT)** The second geometric block contains no linear command.	• Specify the G01 command.
P395	**NO ADRS (GEOMT)** The geometric format is invalid.	• Review the program.
P396	**PL CHG. (GEOMT)** A plane switching command was executed during geometric command processing.	• Execute the plane switching command before geometric command processing.
P397	**ARC ERR (GEOMT)** In geometric IB, the circular arc end point does not contact or cross the next block start point.	• Review the geometric circular arc command and the preceding and following commands.
P398	**NO GEOMETRIC1B** Although the geometric IB specifications are not included, a geometric command is given.	• Check the specifications.
P420	**NO PARAM IN** Although the programmable parameter input specifications are not included, the command was given.	• Check the specifications.
P421	**PRAM. IN ERROR** • The specified parameter number or set data is illegal. • An illegal G command address was input in parameter input mode. • A parameter input command was input during fixed-cycle modal or nose R compensation.	• Review the program.
P430	**AXIS NOT RET.** • A command was issued to move an axis' which has not returned to the reference point' away from that reference point. • A command was issued to an axis removal axis.	• Execute reference point return manually. • The command was issued to an axis for which axis removal is validated so invalidate axis removal.

Error No.	Details	Remedy
P431	**NO 2ndREF. SPC** A command for second' third or fourth reference point return was issued though there are no such command specifications.	• Check the specifications.
P434	**COLLATION ERR** One of the axes did not return to the start position when the origin point collate command (G27) was executed.	• Review the program.
P435	**G27/M ERROR** An M command was issued simultaneously in the G27 command block.	• An M code command cannot be issued in a G27 command block and so the G27 command and M code command must be placed in separate blocks.
P436	**G29/M ERROR** An M command was issued simultaneously in the G29 command block.	• An M code command cannot be issued in a G29 command block and so the G29 command and M code command must be placed in separate blocks.
P438	**NOT USE (G52)** The local coordinate system was commanded during the G54.1 command.	• Review the program.
P450	**NO CHUCK BARR.** The chuck barrier on command (G22) was specified although the chuck barrier was undefined in the specification.	• Check the specification.
P460	**TAPE I/O ERROR** An error has arisen in the tape reader or' alternatively' in the printer during macro printing.	• Check the power and cable of the connected devices. • Check the I/O device parameters.
P461	**FILE I/O ERROR** A file of the machining program cannot be read.	• In memory mode, the programs stored in memory may have been destroyed. Output all of the programs and tool data once and format them.
P600	**NO AUTO TLM.** An automatic tool length measurement command (G37) was execute though there are no such command specifications.	• Check the specifications.
P601	**NO SKIP SPEC.** A skip command (G31) was issued though there are no such command specifications.	• Check the specifications.
P602	**NO MULTI SKIP** A multiple skipping command (G31.1' G31.2 or G31.3) was issued though there are no such command specifications.	• Check the specifications.

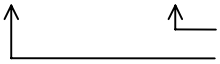
Error No.	Details	Remedy
P603	**SKIP SPEED 0** The skip speed is 0.	• Specify the skip speed.
P604	**TLM ILL. AXIS command** No axis or two or more axes were specified in the automatic tool length measurement block.	• Specify only one axis.
P605	**T-CMD IN BLOCK** The T code is in the same block as the automatic tool length measurement block.	• Specify this T code before the block.
P606	**NO T-CMD BEFOR** The T code was not yet specified in automatic tool length measurement.	• Specify this T code before the block.
P607	**TLM ILL. SIGNL** Before the area specified by the D command or decelerating area parameter d' the measurement position arrival signal went on. The signal remains off to the end.	• Review the program.
P608	**SKIP ERROR (CC)** A skip command was specified during radius compensation processing.	• Specify a diameter cancel (G40) command' or remove the skip command.
P610	**ILLEGAL PARA.** • G114.1 was commanded when the spindle synchronization with PLC I/F command was selected. • Spindle synchronization was commanded to a spindle that is not connected serially.	• Check the program. • Check the argument of G114.1 command. • Check the state of spindle connection.
P990	**PREPRO S/W ERR** Combining commands that required pre-reading (nose R offset, corner chamfering corner R, geometric I, geometric IB, and multiple fixed cycle commands) resulted in eight or more pre-read blocks.	• Reduce the number of commands that require pre-reading or delete such commands.

7.11 Auxiliary Axis Alarms

7.11.1 Auxiliary Axis Alarms (Sxxx, Zxxx, Mxxx)

Display example

S01 AUX SERVO ALM □□□□ □



Axis No. 1 to the number of MR-J2-CT axes connected
Alarm information
(Follows MR-J2-CT alarm information)

(1) S01 AUX SERVO ALM

Alarm information	Details		Remedy
0011	PCB error 1	An error occurred in the drive unit's internal PCB.	• Replace servo drive unit.
0013	Software processing timeout, clock error	An error occurred in the drive unit's internal reference clock.	• Replace servo drive unit.
0016	Motor type, detector type error	Motor type error	• Use a correct drive unit and motor combination.
		Detector initial communication error.	• Connect correctly. • Replace the motor. • Replace or repair cable.
		Detector CPU error	• Replace the motor (detector).
0017	PCB error (A/D conversion initial error)	An error occurred in the drive unit's internal A/D converter.	• Replace servo drive unit.
0025	Absolute position lost	An error occurred in the detector's internal absolute position data.	• Turn the power ON for 2 to 3 minutes while the alarm is occurring, and then turn the power ON again. • Replace the battery, and initialize the absolute position again.
0034	CRC error	An error occurred in the communication with the NC.	• Take countermeasures against noise.
0036	Communication timeout, NC down	Communication with the NC was cut off.	• Connect correctly. • Turn the NC power ON. • Replace the drive unit or NC.
0037	Parameter error (Regenerative resistance error)	The parameter setting value is incorrect.	• Set the parameter correctly.
0038	Frame error	An error occurred in the communication with the NC.	• Take countermeasures against noise.
0039	INFO error	Undefined data was transferred from the NC.	• Change the NC software version to a compatible version.

(2) S02 AUX SERVO ALM

Alarm information	Details		Remedy
0011	PCB error 1 (drive circuit error)	An error occurred in the drive unit's internal PCB.	• Replace servo drive unit.
0013	Software processing timeout, clock error	An error occurred in the drive unit's internal reference clock.	• Replace servo drive unit.
0015	EEROM error	A write error occurred to the EEROM in the drive unit.	• Replace servo drive unit.
0017	PCB error (A/D conversion error)	An error occurred in the drive unit's internal A/D converter.	• Replace servo drive unit.
0018	PCB error (LSI error)	An error occurred in the drive unit's internal LSI.	• Replace servo drive unit.
0020	Detector error	An error occurred in the communication between the servo drive unit and detector.	• Connect correctly. • Replace or repair cable.
0024	Ground fault detection	A ground fault of the output was detected when the power was turned ON.	• Repair the ground fault section. • Replace the cable or motor.

(3) S03 AUX SERVO ALM

Alarm information	Details		Remedy
0010	Undervoltage	The power voltage is 160V or less.	• Review the power supply. • Replace the servo drive unit.
0030	Regeneration error	The tolerable regeneration power of the internal regenerative resistor or external regenerative option was exceeded.	• Set the parameter #002 correctly. • Connect correctly. • Lower the positioning frequency. • Change the regenerative option to a larger capacity. • Lower the load. • Review the power supply.
		Regenerative transistor error	• Replace the servo drive unit.
0031	Overspeed	The motor's speed exceeded the tolerable momentary speed.	• Increase the acceleration/ deceleration time constant. • Review the gear ratio. • Replace the detector.
0032	Overcurrent	A current exceeding the servo drive unit's tolerable current flowed.	• Repair the wiring. • Replace the servo drive unit. • Take countermeasures against noise.

Alarm information	Details		Remedy
0033	Overvoltage	The voltage of the converter in the servo drive unit was 400V or more.	• Wire correctly. • Replace the servo drive unit. • For the internal regenerative resistor, replace the drive unit. • For the external regenerative option, replace the regenerative option.
0046	Motor overheating	An operation state causing the motor to overheat continued.	• Reduce the motor load. • Review the operation pattern.
0050	Overload 1	The servo drive unit or servomotor overload protection function activated.	• Reduce the motor load. • Review the operation pattern. • Change to a motor or drive unit with large output. • Change the setting of the automatic tuning response characteristics. • Correct the connection. • Replace the servomotor.
0051	Overload 2	The max. output current flowed for several seconds due to a machine collision or overload.	• Review the operation pattern. • Change the setting of the automatic tuning response characteristics. • Correct the connection. • Replace the servomotor.
0052	Excessive error	A position deflection exceeding the excessive error detection setting value occurred.	• Increase the acceleration/ deceleration time constant. • Increase the torque limit value. • Review the power facility capacity. • Review the operation pattern. • Replace the servomotor. • Connect correctly. • Repair or replace the cable.

(4) S52 AUX SERVO WRN

Alarm information	Details		Remedy
0092	Battery voltage drop	The absolute position detection battery voltage dropped.	• Mount a battery. • Replace the battery and initialize the absolute position.
00E0	Over-regeneration warning	The regeneration power may have exceeded the tolerable range of the built-in regenerative resistor or external regenerative option.	• Lower the positioning frequency. • Change the regenerative option to a larger one. • Lower the load.
00E1	Overload warning	The overload alarm 1 could occur.	• Refer to the items for S03 0050.

Alarm information	Details	Cause	Remedy
00E3	Absolute position counter warning	There is an error in the absolute position detector internal data.	• Take countermeasures against noise. • Replace the servomotor.
00E9	Main circuit OFF warning	The servo ON signal was input while the main circuit power was OFF. The contactor operation is faulty.	• Turn ON the main circuit power.

(5) Z70 AUX POS. ERR

Alarm information	Details	Cause	Remedy
0001	Zero point initialization incomplete	The zero point (reference point) has not been initialized in the absolute position system.	• Initialize the zero point (basic point).
0002	Absolute position data lost	The absolute position coordinate data in the drive unit has been lost.	• Initialize the zero point (basic point).
0003	Absolute position system related parameter error	The absolute position system related parameters have been changed or lost.	• Correctly set the parameters and then initialize the zero point (basic point).

(6) Z71 AUX DETEC. ERR

Alarm information	Details	Cause	Remedy
0001	Absolute position memory battery voltage drop	The data in the detector has been lost. Battery voltage drop. Detector cable wire breakage or looseness.	• Check the battery and detector cable and then initialize the zero point (basic point).

(7) Z73 AUX SYSTEM WRN

Alarm information	Details	Cause	Remedy
0001	Absolute position memory battery voltage warning	Battery voltage drop. Detector cable wire breakage or looseness.	• Check the battery and detector cable. The zero point does not need to be initialized.
0003	Absolute position counter warning	An error occurred in the detector's absolute position counter.	• Replace the detector.

(8) M00 AUX OPER. ALM

Alarm information	Details	Cause	Remedy
0001	Near-point dog length insufficient	When executing dog-type reference point, the zero point return speed is too fast or the dog length is too short.	• Lower the zero point return speed or increase the dog length.
0003	Reference point return direction illegal	When executing reference point return, the axis was moved in the opposite of the designated direction.	• Move the axis in the correct direction.

(9) M01 AUX OPER. ALM

Alarm information	Details	Cause	Remedy
0101	No operation mode	The operation mode is not designated, or the operation mode was changed during axis movement.	• Correctly designate the operation mode.
0103	Feedrate 0	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.	• Set a value other than zero in the feedrate setting or override value.
0160	Station No. designation illegal. Starting not possible.	A station No. exceeding the No. of indexed divisions was designated.	• Correctly designate the station No.
0161	Reference point return incomplete. Starting not possible.	Automatic/manual operation was started before reference point return was executed with the incremental system.	• Execute the reference point return.
0162	In initializing reference point. Starting not possible.	The start signal was input while initializing the absolute position basic point.	• Complete the absolute position basic point initialization.
0163	In absolute position alarm. Starting not possible.	The start signal was input during an absolute position alarm.	• Initialize the absolute position basic point and then fix the absolute position coordinates.
0164	In random positioning mode. Manual operation not possible.	The manual operation mode was started during the random positioning mode.	• Turn the random positioning mode OFF before switching to the manual operation mode.
0165	Uneven indexing station No. illegal. Starting not possible.	The commanded station No. was higher than 9 or the number of indexing stations during uneven indexing.	• Check the commanded station No. and the parameter “#100 station” setting.

7.11.2 Auxiliary Axis MCP Alarms (Yxxx)

Y02 AUX SYSTEM ALM			An error occurred in the data transmitted between the MCP and drive unit after the power was turned on.
Error No.	**Details**		**Remedy**
0050	Background error		The software or hardware may be damaged. Contact the service center.
0051	0000	CRC error (10 times/910.2ms)	A communication error has occurred between the controller and drive unit. • Take measures against noise. • Check that the communication cable connector between the controller and drive unit and one between the drive units are tight. • Check whether the communication cable between the controller and drive unit and one between the drive units are disconnected. • A driving amplifier may be faulty. Take a note of the 7-segment LED contents of each drive unit and report to the Service Center.
	0001	CRC error (2 continuous times)	
	0002	Reception timing error (2 continuous times)	
	××03	Data ID error (2 continuous times) ××: Axis No.	
	××04	No. of reception frames error (2 continuous times) ××: Axis No.	

Y03 AUX AMP UNEQU.	The drive unit is not correctly connected. Check the auxiliary axis drive unit mounting state. • Check the end of the cable wiring. • Check the cable for broken wires. • Check the connector insertion. The auxiliary axis drive unit input power is not being input. The auxiliary axis drive unit axis No. switch is illegal.
Error No.	**Details**
Axis No.	bit correspondence (bit 0 : 1st axis, bit 1: 2nd axis, bit 2: 3rd axis, …bit6: 7th axis) **(Note)** As for C6 system, up to 5 axes of MR-J2-CT can be connected. Thus, the setting range is bit 0 to 4

Revision History

Date of revision	Manual No.	Revision details
Nov. 2000	BNP-B2259*	First edition created.
Sept. 2003	BNP-B2259A	(1) The cover (front, spine, back covers) design was changed. (2) "Type", "Independent Product Code" and "Reference No." were added to the back cover. (3) The following changes were made to comply with the Version C software. • The following changes were made in "Chapter I Screen Operations". • "2. NC MONITOR" screen (Details were changed due to changes in screen specifications.) • "3. 2-SYSTEM MONITOR" screen (Details were changed due to changes in screen specifications.) • "4. DATA I/O" screen (Target file types and device selection, etc., were changed.) • "5. ALARM DIAGNOSIS" screen (Added) • "6. SETUP/MAINTENANCE" screen (Details were changed due to changes in screen specifications.) 1. "6.6.6 ATA Flash Card I/F" (Added) 2. "6.7.7 Operation History" (Completely revised) 3. Other mistakes were corrected, and unnecessary sections were revised or deleted. • Mistakes were corrected and figures were added in "Chapter 2 Machine Operations". • Mistakes in "Chapter 3 Maintenance" were corrected. • Error messages, etc., were added/revised, and mistakes were corrected in "Chapter 4 Appendices". • Other parameters and error messages, etc., were added or corrected, and other mistakes were corrected.
Mar. 2004	BNP-B2259C	(1) The following changes were made to comply with the Version D software.

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 ELECTRIC CORPORATION
HEAD OFFICE : MITSUBISHI DENKI BLDG., 2-2-3, MARUNOUCHI, CHIYODA-KU, TOKYO 100-8310, JAPAN

MODEL	MC6/C64/C64T
MODEL CODE	008-043
Manual No.	BNP-B2259C(ENG)

(0403) MEE