Changes for the Better

MITSUBISHI ELECTRIC

CNC
MELDAS 60/60S Series

OPERATION MANUAL

BNP-B2180J(ENG)

Introduction

This manual is referred to when using the MELDAS 60/60S Series.
This manual explains how to operate, run and set up this NC unit. Read this manual thoroughly before using the NC unit. To safely use this NC unit, thoroughly study the "Precautions for Safety" on the next page before use.

* The "MELDAS60 Series" includes the M64A, M64, M65, M66 and M65V.
* The "MELDAS60S Series" includes the M64AS, M64S, M65S and M66S.

Details described in this manual

⚠ CAUTION

- ⚠ For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine maker takes precedence over this manual.
- ⚠ Items not described in this manual must be interpreted as "not possible".
- ⚠ This manual is written on the assumption that all option functions are added. Confirm with the specifications issued by the machine maker before starting use.
- ⚠ Refer to the Instruction Manual issued by each machine maker for details on each machine tool.
- ⚠ Some screens and functions may differ depending on the NC system (or its version), and some functions may not be possible. Please confirm the specifications before use.

Refer to the following documents.

MELDAS 60/60S Series	Alarm / Parameter Manual	BNP-B2201
MELDAS 60/60S Series	MELDASMAGIC64 Programming Manual (M TYPE)	BNP-B2182
MELDAS 60/60S Series	MELDASMAGIC64 Programming Manual (L TYPE)	BNP-B2181

< 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 a serial input/output device and save them. Refer to Section "III-8 Maintenance Functions" in this manual for information on how to do this.

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 maker, 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 bodily injury 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

1. Items related to operation

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

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

⚠ CAUTION

1. Items related to product and manual

⚠ For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine maker takes precedence over this manual.

⚠ Items not described in this manual must be interpreted as "not possible".

⚠ This manual is written on the assumption that all option functions are added. Confirm with the specifications issued by the machine maker before starting use.

⚠ Refer to the Instruction Manual issued by each machine maker for details on each machine tool.

⚠ Some screens and functions may differ depending on the NC system (or its version), and some functions may not be possible. Please confirm the specifications before use.

2. Items related to installation and assembly

⏚ Ground the signal cables to ensure stable system operation. Also ground the NC unit main frame, power distribution panel and machine to one point, so they all have the same potential.

⚠ If the control unit's rotary switch is set to "7", all data in the NC will be erased and the system will not start up.

3. Items related to preparation before use

❗ Always set the stored stroke limit. Failure to set this could result in collision with the machine end.

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

4. Items related to screen operation

⚠ If the tool offset and workpiece coordinate system offset are changed during automatic operation (including during single block stop), they are validated from the command of the next block or blocks onwards.

⚠ When forcibly setting (forcibly outputting) data on the I/F diagnosis screen during machine operation, pay careful attention to the sequence operation.

⚠ All of the various data in the NC memory is erased when formatting. Be sure to use the transfer function to transfer all necessary data to another storage device before formatting.

⚠ Even if the tool compensation amount write command, parameter write command or variable data write command is executed with graphic check, the data will be actually written in, and the original data will be overwritten.

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

⃠ Do not change setup parameters without prior approval from the machine maker.

5. Items related to programming

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

⚠ " ; " "EOB" and " % " "EOR" are explanatory notations. The actual codes are "Line feed" and "%" for ISO, and "End of Block" and "End of Record" for EIA.

⃠ Do not change the fixed cycle program without prior approval from the machine maker.

(Continued on next page)

⚠ CAUTION

6. Items related to operation

- ⚠ Stay out of the moveable range of the machine during automatic operation. During rotation, keep hands, feet and face away from the spindle.
- ⚠ Carry out dry operation before actually machining, and confirm the machining program, tool offset and workpiece coordinate system offset.
- ⚠ 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.
- ⃠ Program so the mirror image function is turned ON/OFF at the mirror image center. The mirror image center will deviate if the function is turned ON/OFF at a position other than the mirror image center.

7. Items related to faults and abnormalities

- ❗ If a BATTERY FAULT alarm is issued, save the machining program, tool data and parameters before replacing the battery.
- ❗ If the axis overruns or emits an abnormal noise, immediately press the emergency stop button and stop the axis movement.

8. Items related to maintenance

- ⚠ Incorrect connections may damage the devices, so connect the cables to the specified connectors.
- ⚠ Do not apply voltages other than those indicated in this manual on the connector. Doing so may lead to destruction or damage.
- ⃠ 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.
- ⃠ Do not connect the cable by pulling on the cable wire.
- ⚠ Do not short circuit, charge, overheat, incinerate or disassemble the battery.
- ⚠ Dispose the spent battery according to local laws.
- ⚠ Do not replace the control unit while the power is ON.
- ⚠ Do not replace the base I/O unit while the power is ON.
- ⚠ Do not replace the control section power supply PCB while the power is ON.
- ⚠ Do not replace the expansion PCB while the power is ON.
- ⚠ Do not replace the memory cassette while the power is ON.
- ⚠ Be careful that metal cutting chips, etc., do not come into contact with the connector contacts of the memory cassette.
- ⚠ Do not replace the high-speed program server unit while the power is ON.

Disposal

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

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

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

Contents

I. OPERATION SECTION

II. MACHINE OPERATION MANUAL

III. SETUP

IV. APPENDIXES

I. OPERATION SECTION

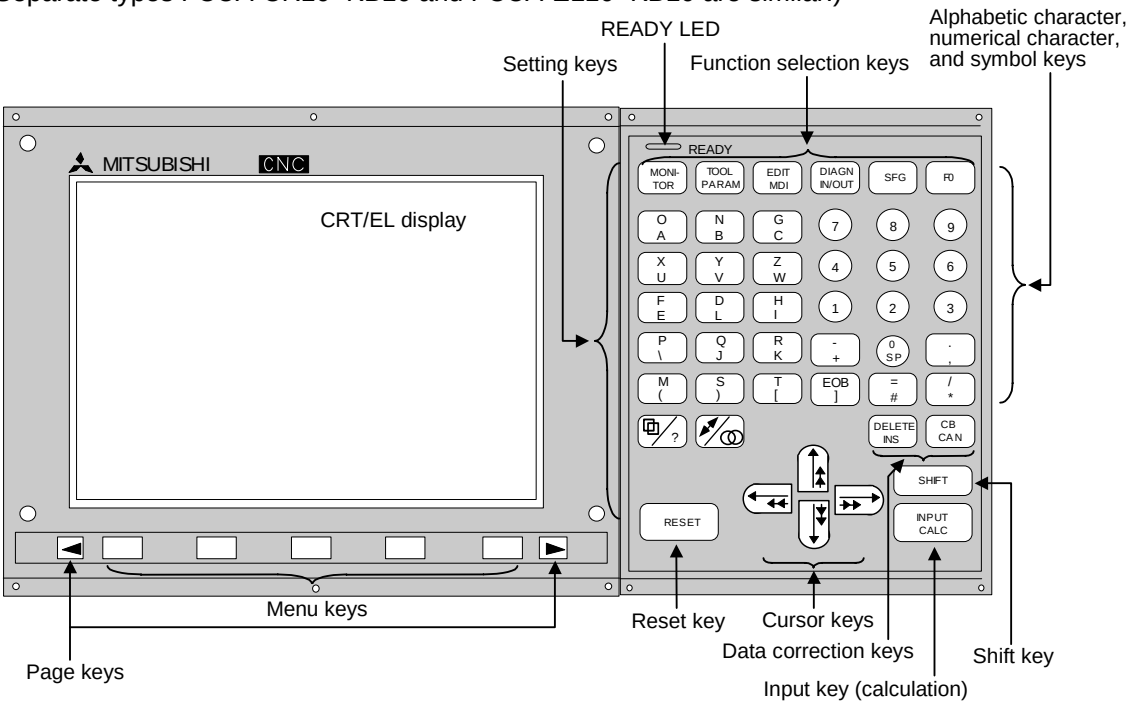
1. Setting and Display Unit Operation

1.1 Appearance of Setting and Display Unit

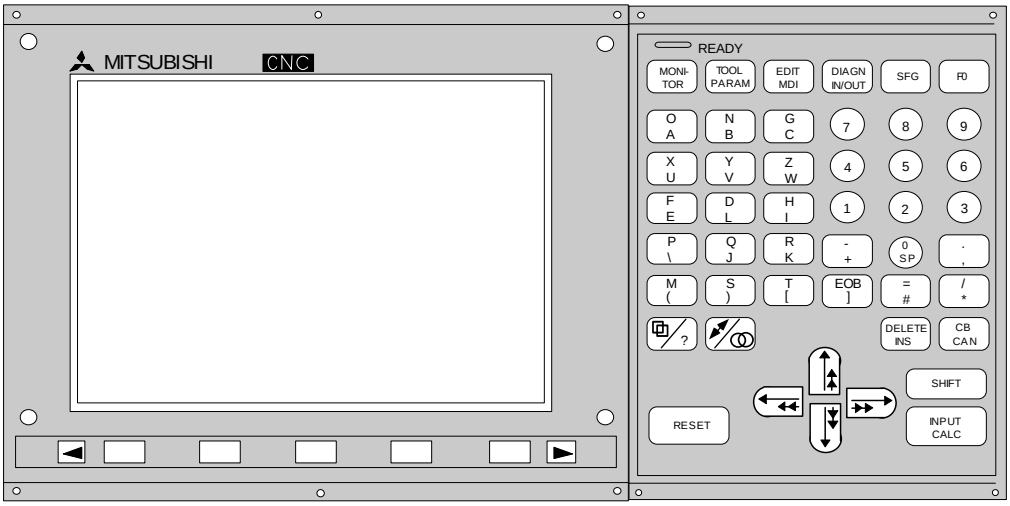
The setting and display unit consists of a display unit (9-inch umber color), keys, and menu keys, as illustrated below:

(1) Appearance of the CT100 Setting and Display Unit ... Example of key layout for machining center system

(Separate types FCUA-CR10+KB10 and FCUA-EL10+KB10 are similar.)



(2) Appearance of the CT120 Setting and Display Unit ... Example of key layout for lathe system

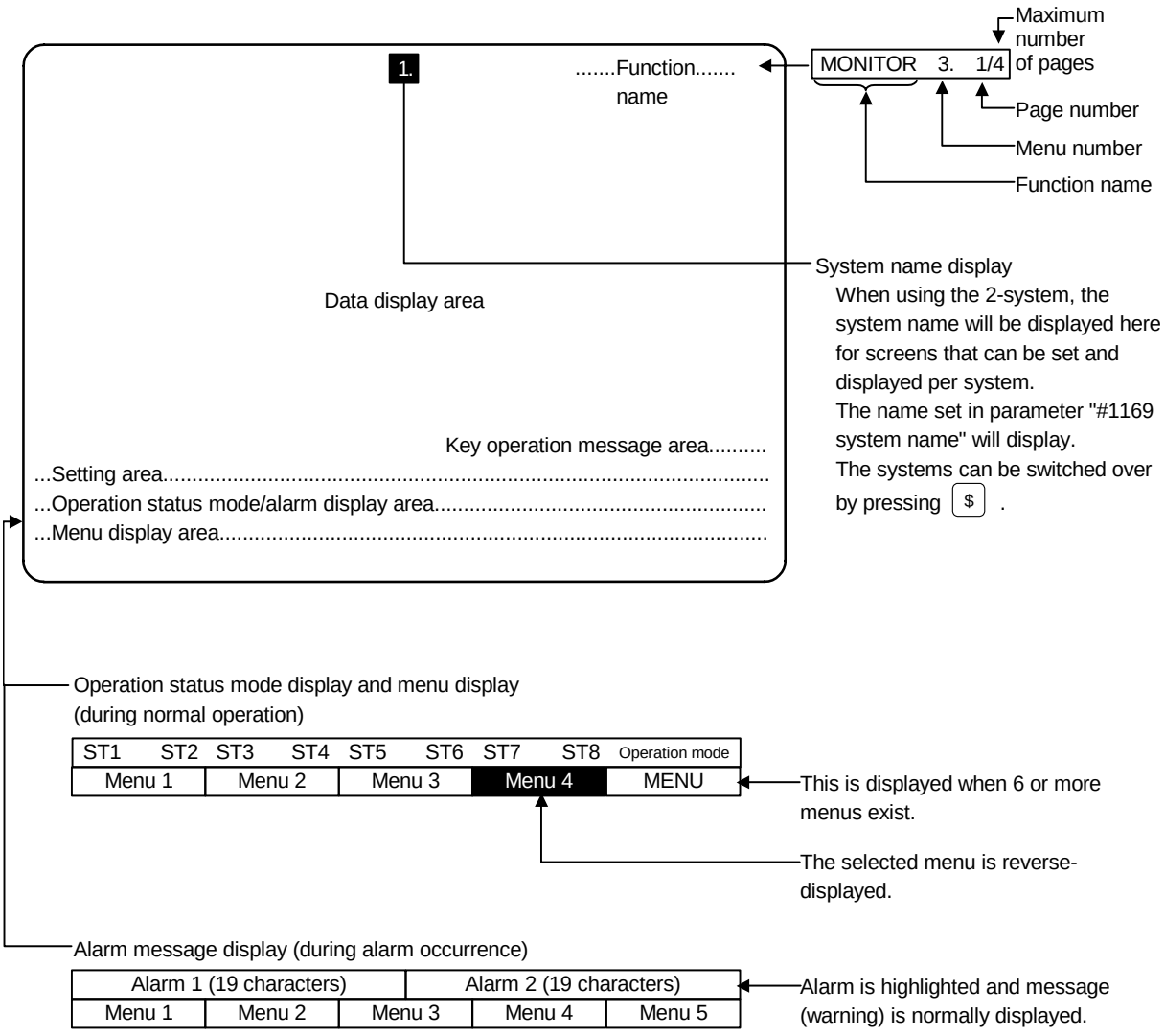


(Note 1) To enter the letter or symbol on the lower right of an alphabetic or symbol key, press the corresponding key while holding down the [SHIFT] key.

(Example) Pressing the [O A] while holding down the [SHIFT] key types letter "A".

1.2 Functions of Display Areas

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



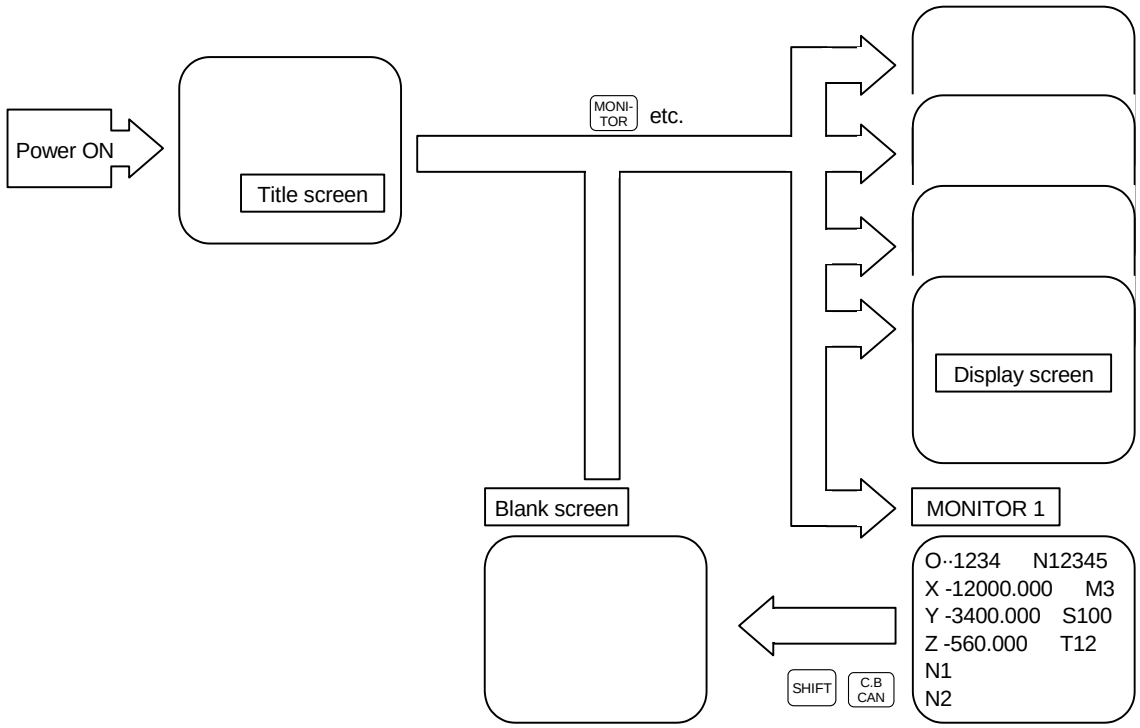
Explanation of operation status display

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	␣␣␣	This indicates that subprogram is not executed.
	SB1 ∿ SB4	Machining program execution is controlled according to subprogram data. Each value of 1 to 4 indicates the subprogram depth.
ST5	G54 ∿ G59	Selection of the workpiece coordinate system is indicated.
ST6	G40	Tool radius compensation cancel state
	G41	During tool R compensation (left)
	G42	During tool R compensation (right)
ST7	fix	Fixed cycle is being executed.
	PR	State in which power must be rebooted to validate set parameter.
	␣␣␣	State other than the above.
ST8		

(Note 1) ␣␣␣ denotes blank display.

1.3 Screen Transition Diagram

1.3.1 Screen Transition when Power Is Turned ON



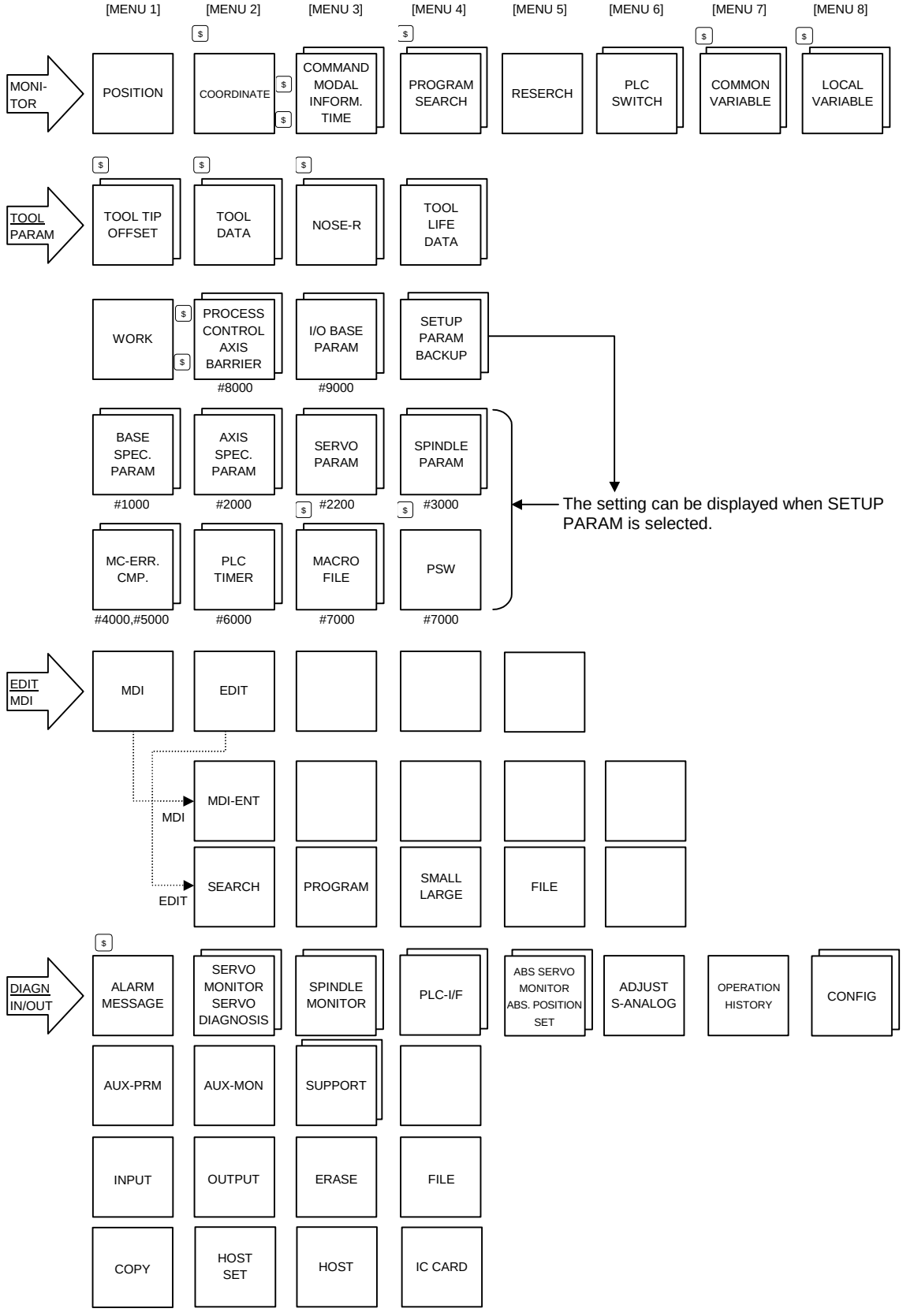
(1) When the power is turned ON, the "Title" screen is displayed. To select a display screen on the "Title" screen, press the corresponding "function selection" key.

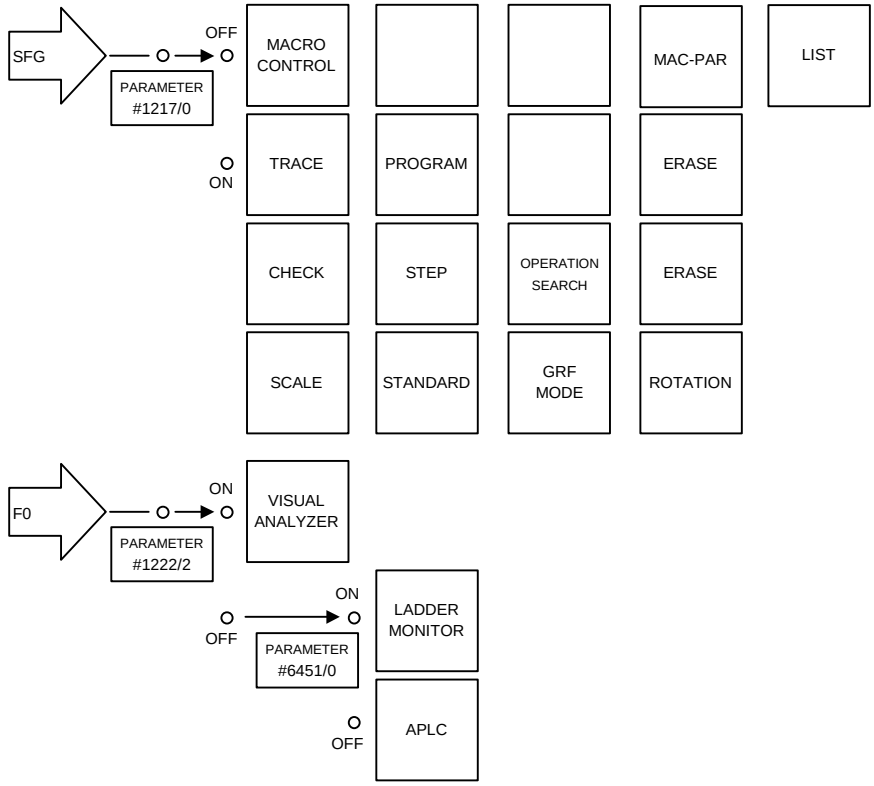
(2) To select a blank screen on a display screen, select the "MONITOR 1" screen and press SHIFT key, then C.B CAN key.

To select a display screen on the "blank screen", press the corresponding "function selection" key.

1.3.2 Screen Transition Diagram (Lathe system)

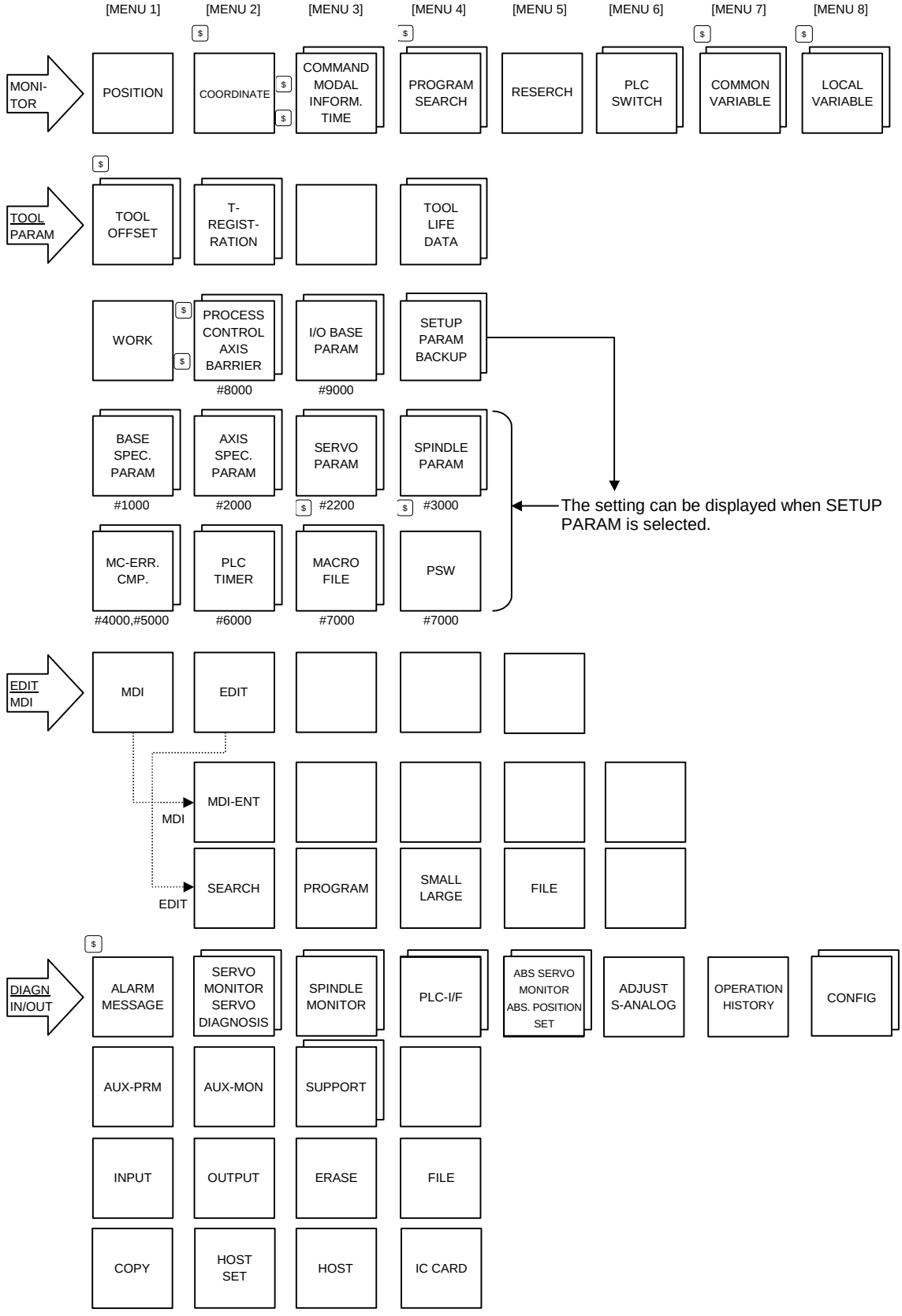
Screens with a [$] mark will change between systems if the [$] key is pressed when using the 2-system.

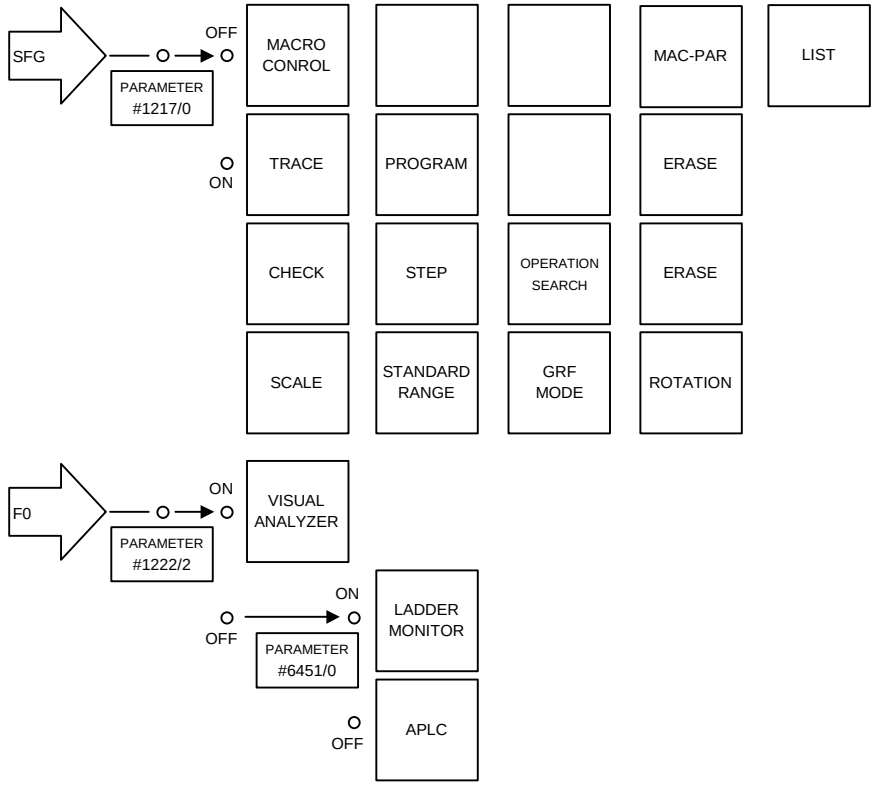



SFG
OFF
ON
PARAMETER
#1217/0
MACRO
CONTROL
MAC-PAR
LIST
TRACE
PROGRAM
ERASE
CHECK
STEP
OPERATION
SEARCH
ERASE
SCALE
STANDARD
GRF
MODE
ROTATION
F0
ON
PARAMETER
#1222/2
VISUAL
ANALYZER
ON
OFF
PARAMETER
#6451/0
LADDER
MONITOR
OFF
APLC

1.3.3 Screen Transition Diagram (Machining center system)

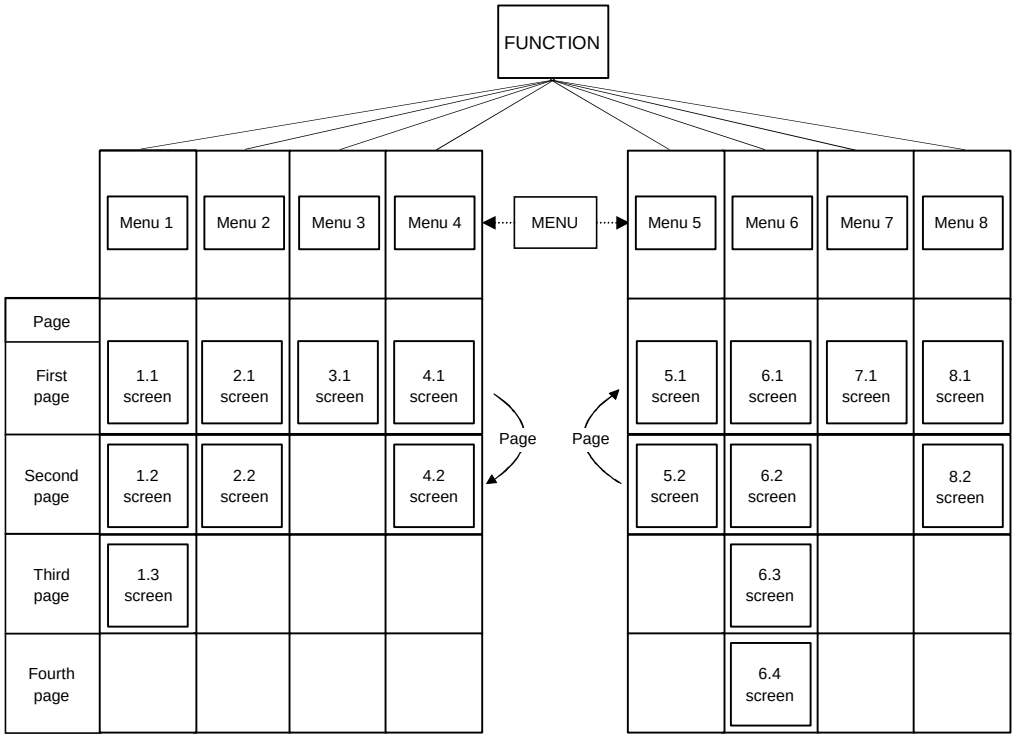
Screens with a [$] mark will change between systems if the [$] key is pressed when using the 2-system.




SFG
OFF
PARAMETER
#1217/0
ON
MACRO
CONROL
MAC-PAR
LIST
TRACE
PROGRAM
ERASE
CHECK
STEP
OPERATION
SEARCH
ERASE
SCALE
STANDARD
RANGE
GRF
MODE
ROTATION
F0
ON
PARAMETER
#1222/2
VISUAL
ANALYZER
ON
OFF
PARAMETER
#6451/0
LADDER
MONITOR
OFF
APLC

1.4 Screen Selection Procedure

The following operation methods are based on using the exclusive setting and display unit.
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.



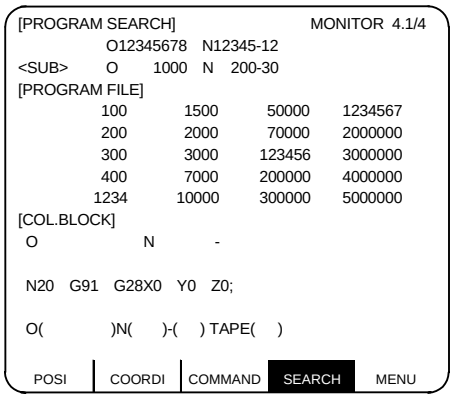
(1) Select a function screen.

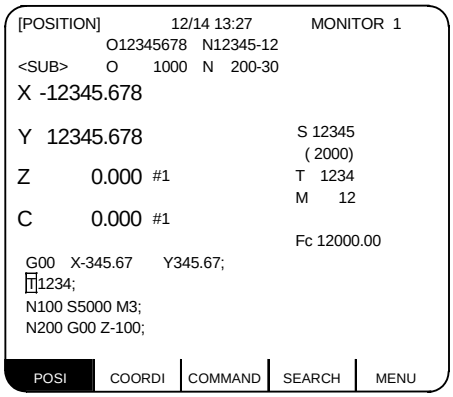
MONI-TOR TOOL PARAM EDIT MDI DIAGN IN/OUT SFG F0

Press the function selection key corresponding to the function screen to be displayed.
(Example) Press the MONI-TOR key.

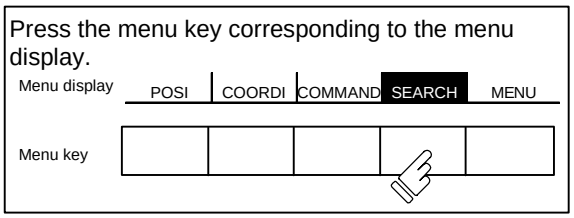
1) The previously displayed menu screen is displayed in the data display area.
2) The first display screen after power is turned ON is the screen on the first menu.

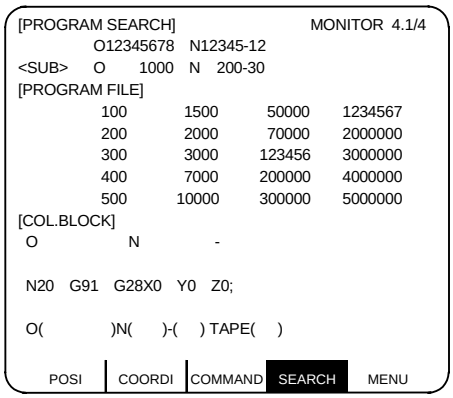
If the same function selection key is again pressed, a return is made to the first page screen of the first menu.
(Example) Again press the MONI-TOR key.





(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.

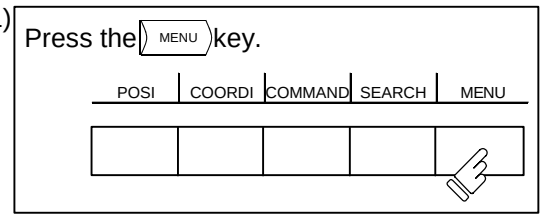


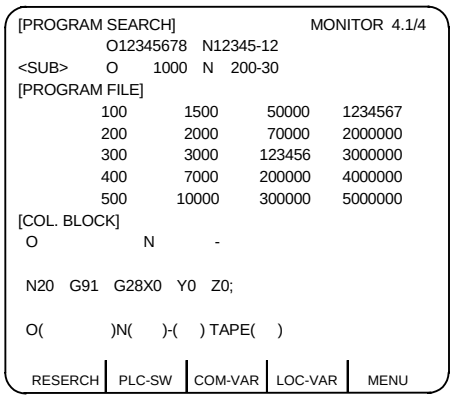


1) The selected menu screen is displayed in the data display area.
2) The selected menu is highlighted in the menu display area.

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.

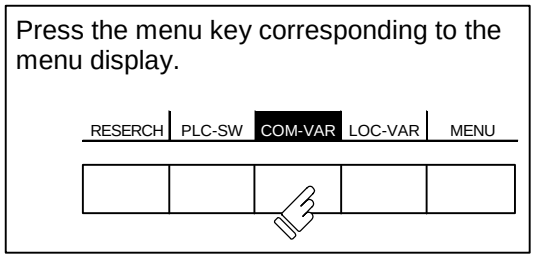
(1)

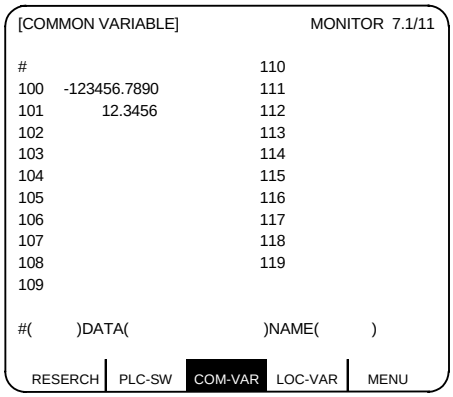




1) Only the menu display area is changed and the remaining menu group is displayed.

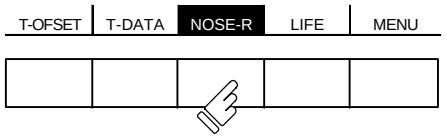
(2)

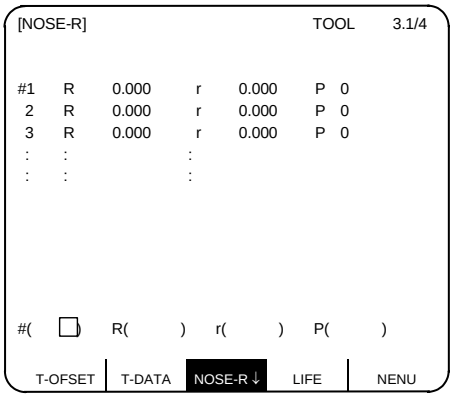




When the screen selection menu is selected, the screen that " ↓ " mark is displayed after the menu means that the operation menu exists.

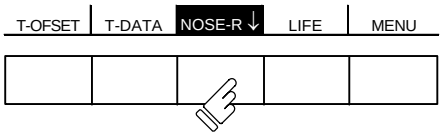
(1) Press the menu key corresponding to the menu display.

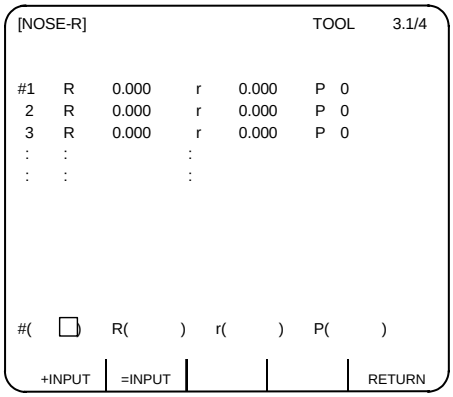




1) The selected menu screen is displayed in the data display area.
2) The selected menu is highlighted and " ↓ " mark is displayed after the menu.

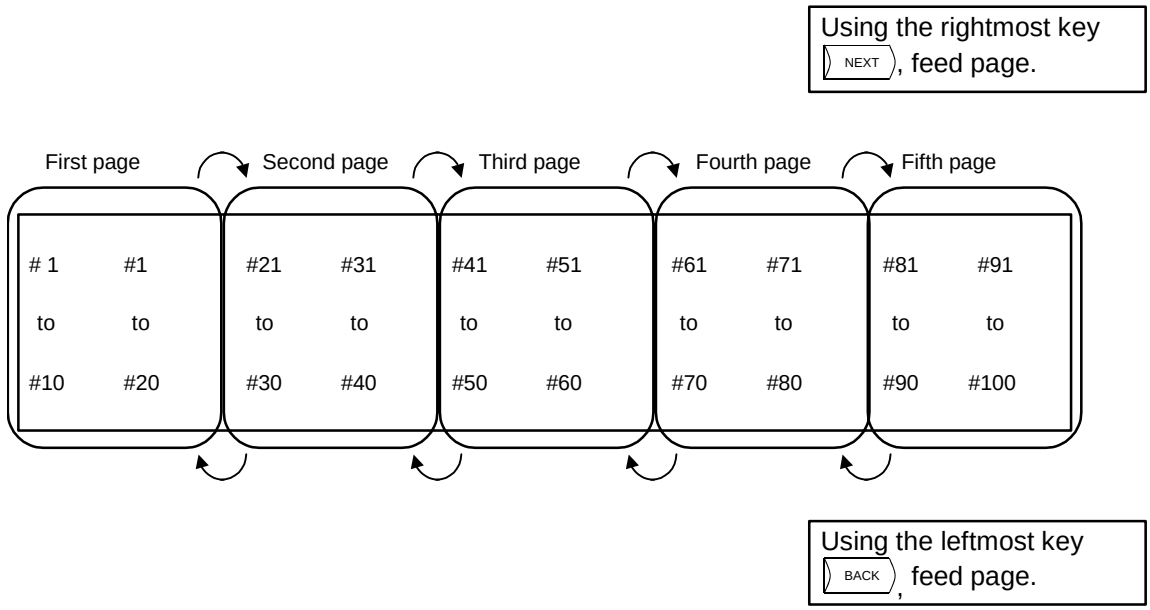
(2) Press the menu key again.





1) The operation menu is displayed in the menu display area.

(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) is the "next page" screen selection key. The leftmost page key (BACK) is the "previous page" screen selection key.



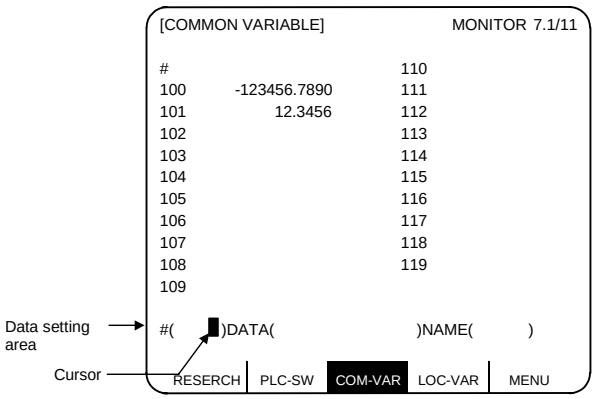
1.5 Data Setting Method

(1) Outline of data setting

The data setting method consists mainly of the following steps:

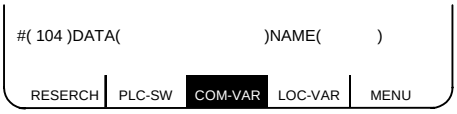
(1) Enter the data number.
(2) Move the cursor.
(3) Press data keys.
(4) Press the INPUT key.

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].

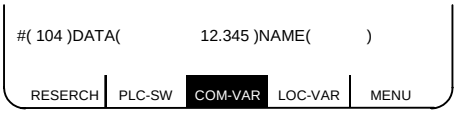


(2) Move the cursor.

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

(3) Press data keys.

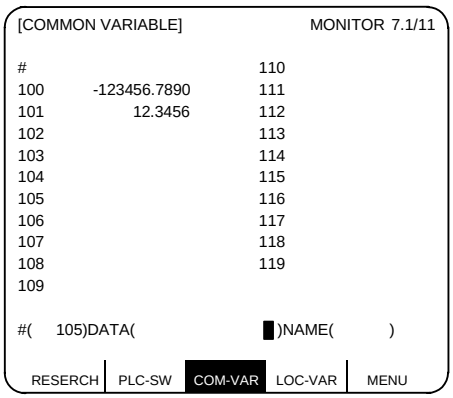
Seeing the data display area contents, enter new data by using the keys.
(Example) To change to 12.345, press [1] [2] [•] [3] [4] [5].



(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.

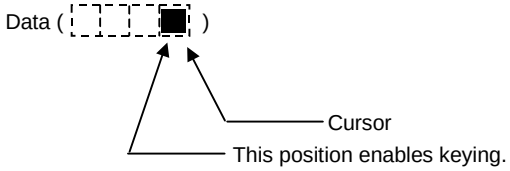


3) To consecutively set data, repeat (3) and (4).
4) To change the data number, press the [INPUT] key. The number is incremented by one. When the [↓] key is pressed, the number is incremented by one. When the [↑] key is pressed, the number is decremented by one. The data number can also be directly changed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

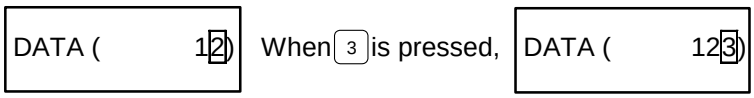
(Note 1) Data in the setting area is only displayed on the screen and is not set in memory until the [INPUT] key is pressed. If the screen is changed before the [INPUT] key is pressed, the data in the setting area becomes invalid.

(2) Cursor control and operation examples

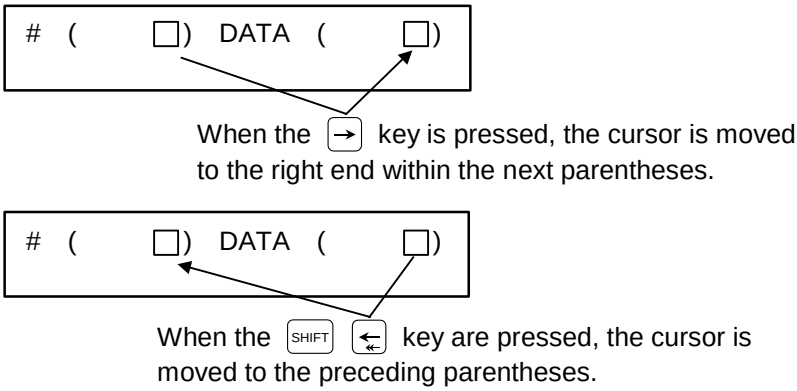
1) Data write into the display screen (by keying) is made at the position indicated by the cursor. When the cursor is not displayed, keying is not effective.



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.



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



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

If you press [3], [3] by mistake,	# (12) DATA (1233)
If you once press the [DELETE INS] key,	# (12) DATA (123)
If you again press the [DELETE INS] key,	# (12) DATA (12)

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

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

Display is made in the setting area as shown in the right.	☞	# (10) DATA (12.345)
If you press the C.B CAN key,	☞	# (10) DATA ( )

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

Display is made in the setting area as shown in the right.	☞	# (10) DATA (12.345)
If you press SHIFT C.B CAN,	☞	# ( ) DATA ( )

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

Display is made in the setting area as shown in the right.	☞	# (10) DATA (12.345)
If you make successive four strokes of the ← key,	☞	# (10) DATA (12.345)
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 (13.345)

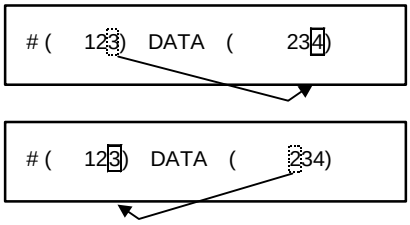
The cursor is only moved one column to the right.

If you press 0 SP 0 SP 0 SP,	☞	# (10) DATA (13.000)

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 parenthesis, 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 parenthesis, the cursor is moved to the right end within the preceding parentheses.

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



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

If you press the SHIFT ← 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.

1.6 Screen Saver / Backlight OFF

The screen saver function protects the display unit by turning OFF the screen after the time set in the parameters has elapsed. The screen can also be turned OFF with key operations on the POSITION screen. The backlight OFF function turns OFF the backlight in order to extend the life of the LCD screen's backlight. The screen can be turned ON by pressing any of the keys on the key operation panel.

(1) Turning the screen OFF

(a) Screen Saver

If there is no key operation or a screen display request signal input from the machine within the time set in the parameter (#8078 Screen Saver Time), the screen will be turned OFF.
If the parameter is set to 0, the screen will not be turned OFF.

The screen can turn OFF by pressing the [SHIFT] and [C.B CAN] keys on the POSITION screen. Even if the parameter is set to 0, the screen can be turned OFF by pressing the [SHIFT] and [C.B CAN] keys.

(Note) The screen will not be turned OFF even if the [SHIFT] and [C.B CAN] keys are pressed on a screen other than the POSITION screen.

(b) Backlight OFF

When the Screen Sever function works, the backlight turns OFF.

(2) Turning the screen ON

If a key is pressed or the screen display request signal is input while the screen is OFF, the screen will turn ON. (If an LCD is used, the backlight will also turn ON.)
When a function select key is pressed, the screen will turn ON (the backlight will also turn ON for an LCD), and each key function will be executed.

When alphanumeric or symbol key is pressed:

[1] → Only screen is turned ON

[Z] → Only screen is turned ON

* The following keys are also included.

([INPUT]/[↑]/[↓]/[←]/[→]/[C.B CAN]/[DELETE INS])

When a function select key is pressed:

[MONI-TOR] → Screen turns ON and screen shifts

(Note) If a key is pressed or the screen display request signal is input while the screen is ON, counting of the time to turn the screen OFF will restart.

(3) Setting the parameters

#	Item	Contents	Setup range (unit)
8078	Screen Saver	Set the time to turn the screen OFF. The screen saver will not turn ON if 0 is set.	0 to 60 (min) 0: Do not turn screen OFF.

(4) Target display units

This screen saver function is valid with the following display units.

(a) 9-type CRT/9.5-type EL
(b) 7.2-type/10.4-type monochrome LCD
(c) 10.4-type color LCD

(Note1) The display unit in (a) is valid only when the screen is turned OFF by the SHIFT and C.B CAN keys on the POSITION screen.

(Note2) The display units in (b) and (c) have a backlight which is turned ON/OFF.

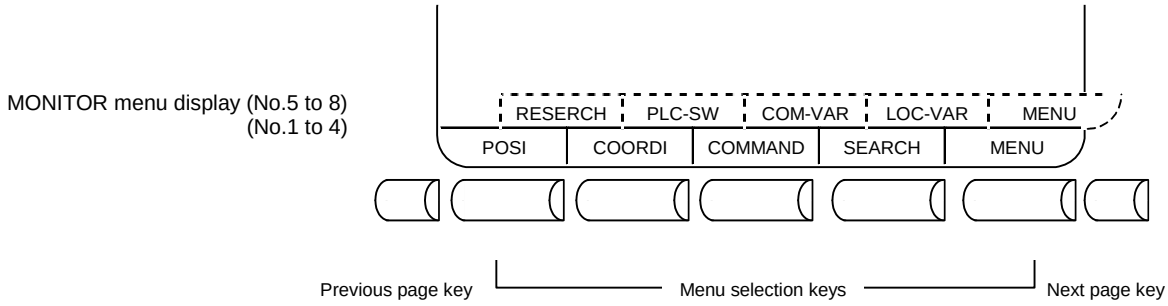
(5) Precautions

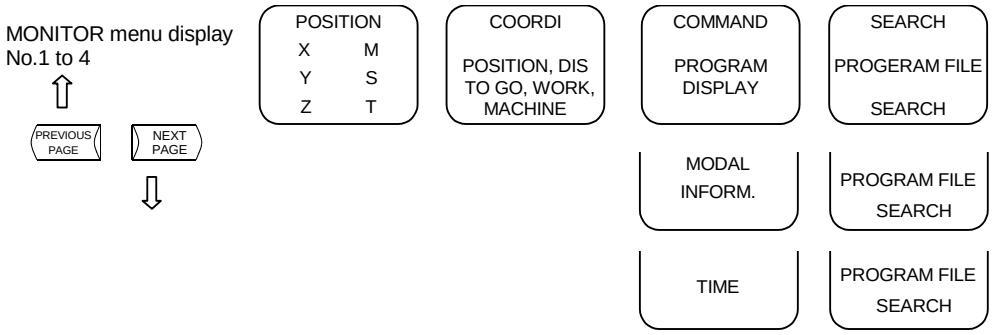
(a) If the screen is turned OFF while keys can be operated on the EDIT screen, etc., and an alphanumeric, symbol or INPUT key is pressed, the first key will be handled as that for turning the screen ON. The key will not be input.
(b) If the function key, menu key, page key or system changeover key is pressed while the screen is OFF, the screen corresponding to the pressed key will turn ON.
(c) The screen will not turn ON even if the reset key is pressed. However, if the screen display request signal is input when the reset key is pressed, the screen will turn ON. Refer to the instruction manual issued by each machine maker for details. Note that whether the screen will turn ON when any of the machine operation board keys (other than the NC operation board keys) is pressed will differ according to the machine specifications. Refer to the instruction manual issued by each machine maker for details.
(d) Correspondence of SHIFT key.

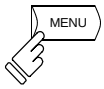
The screen will not turn ON when just the SHIFT key is pressed.

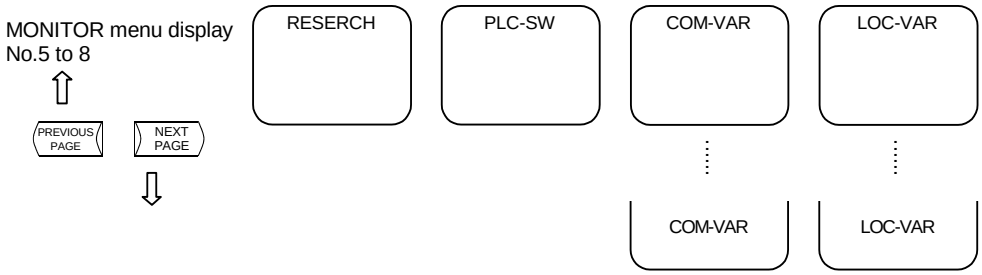
2. Monitor

When the function selection key [MONI-TOR] is pressed, the following menu appears:





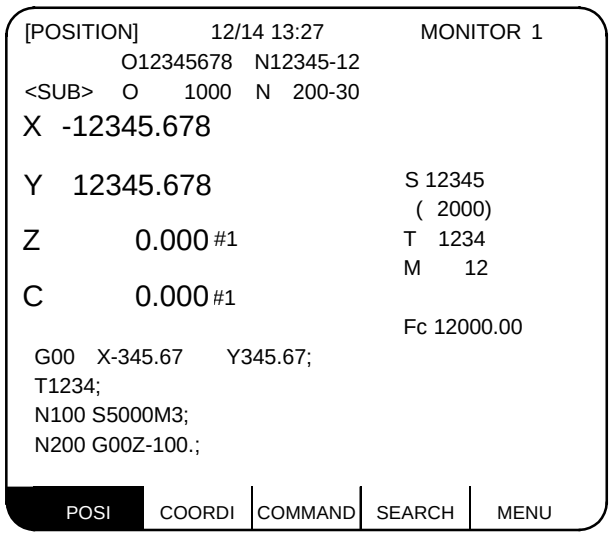




2.1 POSITION

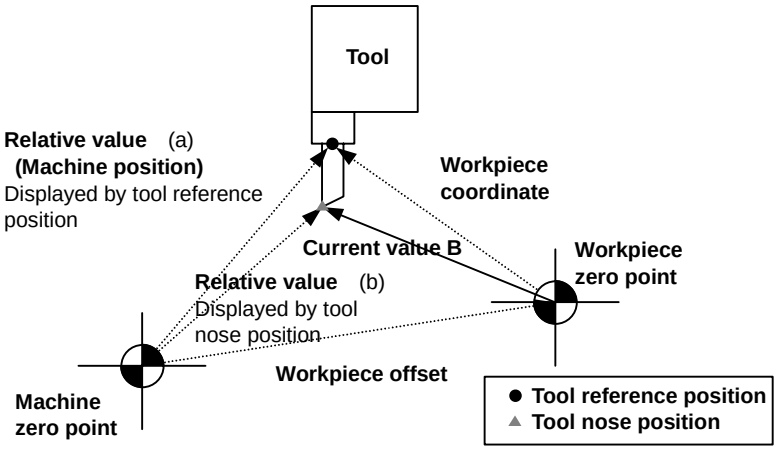
When the menu key [POSI] is pressed, the POSITION screen is displayed.

(4-axis pecifications)



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 relative value is coordinate position using referenced to the machine zero point. 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) are displayed. Whether the tool reference position (figure below (a)) or the current position of the tool nose position (figure below (b)) that considers offset, such as tool length offset amount or tool diameter compensation amount, in the tool reference position is applied to the display of the relative value can be selected with the parameter.  (Continued on next page)

<table>
<tr><th>Display item</th><th>Explanation</th></tr>
<tr><td></td><td>The relative of the relation value counter display contents and the parameters is as shown below.
<M60 series>
<table><tr><th>#1287 ext23 /bit3
#1221 aux 05/bit7</th><th>0</th><th>1</th></tr><tr><td>0</td><td>Relative value (figure below (a))</td><td>Relative value (figure below (b))
The compensation amount is considered according to #1287 ext23/bit4,5 contents.</td></tr><tr><td>1</td><td>Relative value (figure below (a))</td><td>The current value B is displayed in M60S series.
In the M60A/64 lathe system, not only the counter of the POSITION screen but also the relative value of the COORDINATE screen changes to the current value B.
The relative value (figure below (b)) is displayed in the M64A/64(Machining system) /65/66.</td></tr></table>
<M60S series>
<table><tr><td rowspan="2">#1221 aux 05/bit7</td><td>0</td><td>Relative value
The compensation amount is considered according to #1287 ext23/bit4,5 contents.</td></tr><tr><td>1</td><td>Current value B
The counter of the POSITION screen changes to the current value B.</td></tr></table></td></tr>
<tr><td>S 12345 (2000)

T 1234
M 12

Fc 12000.00</td><td>The spindle rotation speed command value is displayed. The actual spindle rotation speed is shown in ( ).
The tool command value is displayed.
The last four digits of the miscellaneous function command value are displayed.
During interpolation feed, the speed in the current vector direction in moving is displayed.
During independent axis feed, the speed of the axis with the highest speed is displayed.</td></tr>
<tr><td>G00 X-345.67 Y345.67;
T1234;
N100 S5000M3;
N200 G00Z-100.;</td><td>Four blocks of the current program being executed are displayed.
The top block is an already executed block.
The subsequent three lines are the subsequent block program.</td></tr>
</table>

2.1.1 Total Clear of Screen

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

(1) Select 1st menu 〉POSI〈 on the MONITOR screen and press the SHIFT and C.B CAN keys to clear total screen.

(2) If you want to display screen after clearing of total screen, press a function select key such as MONI-TOR to display the screen you desire.

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 ;.

(Note1) Origin zero is valid only when #1123 origin is set to 0.
(Note2) Counter zero and origin zero are disabled in the current value B.

In the following operations, the INPUT key has the counter zero function and the C.B CAN key has the origin zero (set zero) function.

```
X -12345.678
Y      1.234
Z     12.345
C    123.456
```

Press the address key X.

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

```
X -12345.678
Y      1.234
Z     12.345
C    123.456
```

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

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

```
X      0.000
Y      1.234
Z     12.345
C    123.456
```

```
X      0.000
Y      0.000
Z     12.345
C    123.456
```

2.1.3 Manual Numeric 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 numeric command

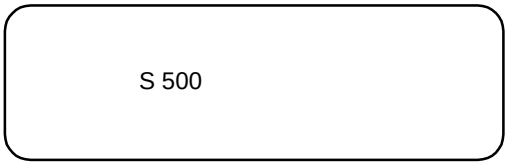
M, S or T command sequence is not under way. Even during automatic start or pause, for example, the manual numeric command is available if above conditions are met.

(2) Operating procedures of manual numeric 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 numeric command. The spindle function key is [S], 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.

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

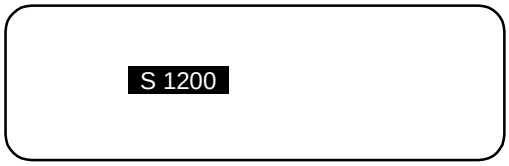
On screen, last executed command value is displayed. ☞



Press the address key [S]. ☞

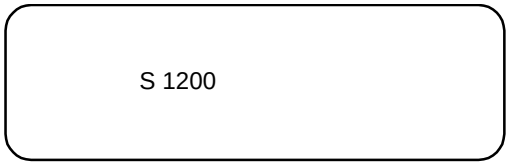
1) The address corresponding to the pressed key and numerical value setting range are highlighted.

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:

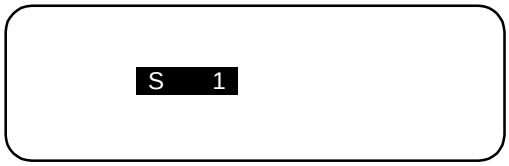
Method (1) While pressing the DELETE INS key, delete the set digits one by one. Then, retry to enter the correct digits.

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

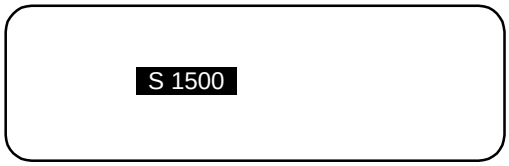
(Example) In this condition, the numeric is desired to be replaced by S1500.

Method (1)

Press DELETE INS DELETE INS DELETE INS 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.

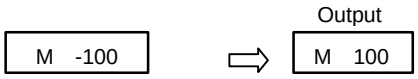
(4) Setting/output range of manual numeric command

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

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

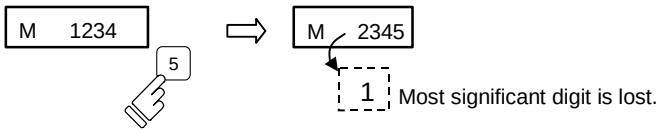
(Note 1) If the type is BCD output and a negative number is set, the positive value converted from it will be output.

(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)

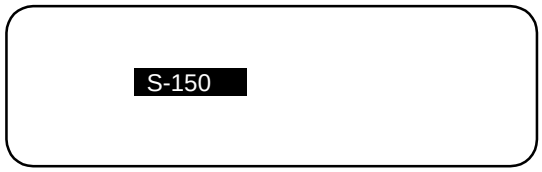


(5) Other notes on operation

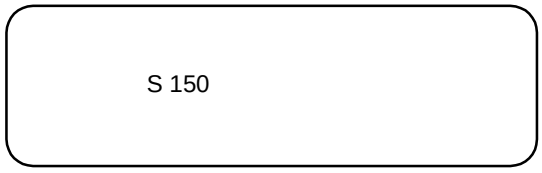
(1) When a minus command is set:
Before setting the numeric, press the [−] key.

(Example) If S-150 is specified:

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



Press the [INPUT] key.



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.
- If a manual numeric command address-key such as M, S, T is pressed, the previous operation will stop. In this case, the next manual numeric command sequentially begins.
- 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.
- If the [SHIFT] [C.B CAN] keys are pressed, the manual numeric command will stop. In this case, the POSITION screen is blanked.
- If one of the following keys is pressed, the operation will not stop:
  1) Position display function key [MONI-TOR]
  2) [−] key pressed before a numeric is set (will be processed as a minus command.)
  3) [DELETE INS] 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 these codes are issued.

(4) No surface speed command is available.
In the constant surface 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 numeric commands are carried out (M, S, T keys) are attempted when the manual numeric command protect function is valid, the error message "E05 NOT ACCEPTABLE" will occur.

2.1.4 Displaying Automatic Operation Program

(1) Displaying the operation program during automatic operation
During memory, tape, 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
The head block of the operation searched program is displayed at the line of the next command.

(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 work program and one execution block

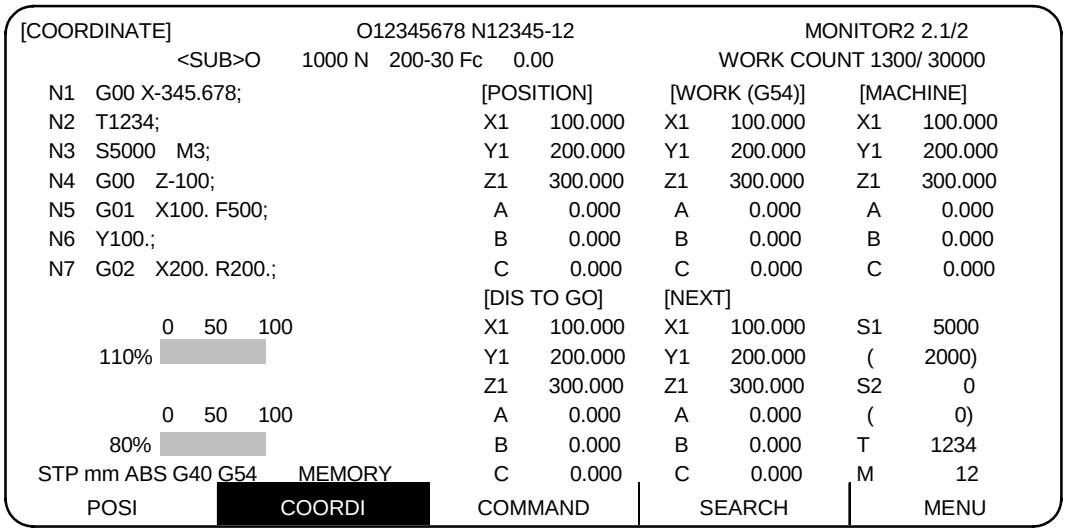
1) A block containing only EOB or only a comment statement is not interpreted as one execution block. Instead it is processed as one block together with the next block.

2) A block that does not contain a movement command or MST command, such as a variable command, is not interpreted as one execution block. Instead, the program up to the block containing a movement command or MST command is handled in the same manner as one block.

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

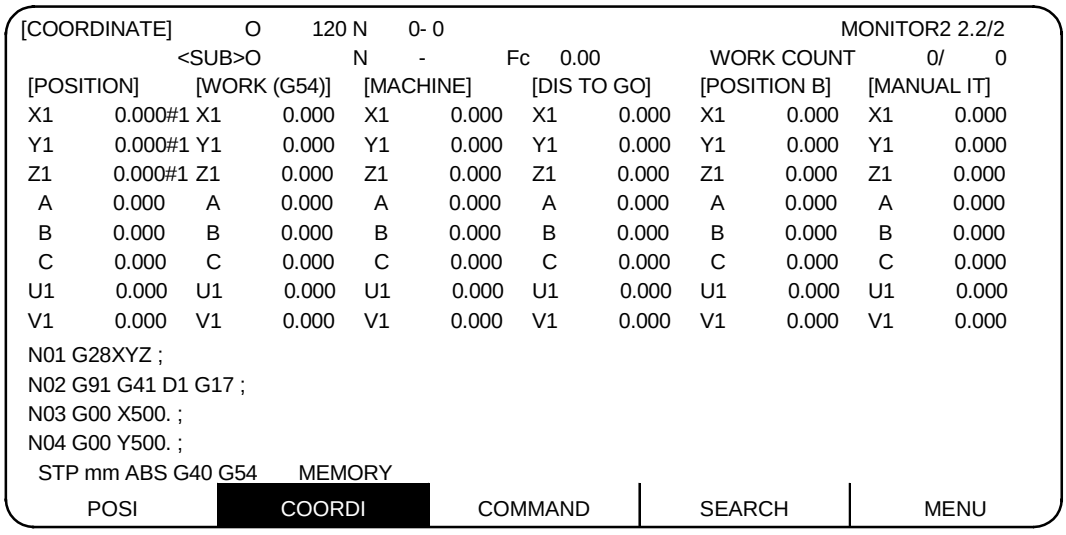
2.2 COORDINATE

When the menu key COORDI is pressed, the COORDINATE screen is displayed.

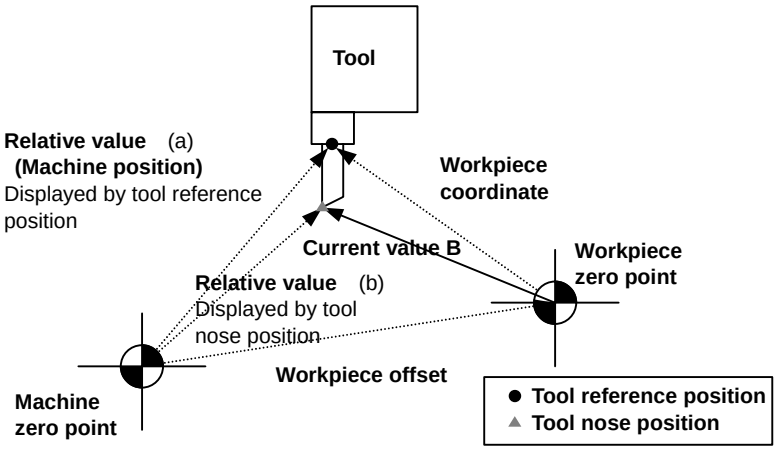


Multiple axis display screen

This screen is displayed on second page of the COORDINATE screen for 4-Spindle specification or more or 7-Servo axis specification or more.



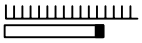
Display item	Explanation
O12345678 N12345-12	The currently executing program number, sequence number, and block number are displayed.
<SUB> O 1000 N 200-30	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.

Display item	Explanation
[POSITION] X -345.678 Y 345.678 Z 0.000#1 C 0.000	The current position during execution and the status abbreviation of the axis are displayed. The status symbol is the same as the display on the POSITION screen. **(Note)** When using the M64A/M64 lathe system, the relative value display can be changed to the current value B (value that does not include tool length offset amount, tool diameter compensation amount, workpiece coordinate offset amount) by setting parameter "#1221 aux05/bit7" and "#1287 ext23/bit3". 

Display item	Explanation
[WORK (G54)] X -345.678 Y 345.678 Z 0.000 C 0.000	G54~G59, P1~P48 workpiece coordinate system modal numbers and the workpiece coordinates in the workpiece coordinate system are displayed. **(Note)** P1 to P48 are options.
[MACHINE] X -345.678 Y 345.678 Z 0.000 C 0.000	The coordinate 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.
[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] X1 0.000 S1 5000 Y1 0.000 (2000) Z1 0.000 S2 0 A 0.000 (0) B 0.000 T 1234 C 0.000 M 12 (Note) On the multi-axis display screen, this display item corresponds to the area in which [POSITION B] and [MANUAL IT] are displayed.	This displays the command contents of the block executed after the block currently in execution during automatic operation. The following display items can be selected according to setting the parameters. **(Note1)** [MST] The spindle rotation speed command value is displayed. The actual spindle rotation speed is shown in ( ). The tool command value is displayed. The last four digits of the miscellaneous function command value are displayed. [POSITION B] Tool nose position coordinate that is considered tool length offset and tool diameter compensation can be displayed in workpiece coordinate. Tool length offset and tool diameter compensation amount that are considered depend on tool (T) designation or the currently selected tool No. that is input from the external source. [MANUAL IT] The amount moved with the manual mode while the manual absolute switch was OFF is displayed. The manual interrupt amount can be selected for the counter value displayed on the coordinate value screen using parameter.

(Note1) The type of position counter to display can be selected with the base specification parameter (#1137 Cntsel).

Parameter #1137 Cntsel	Counter	
	Left	Right
00 or 10	Next command	MST
01 or 11		Next command
02 or 12		Current value B
03 or 13		Manual interrupt amount
20	Current value B	MST
21		Next command
22		Current value B
23		Manual interrupt amount
30	Manual interrupt amount	MST
31		Next command
32		Current value B
33		Manual interrupt amount

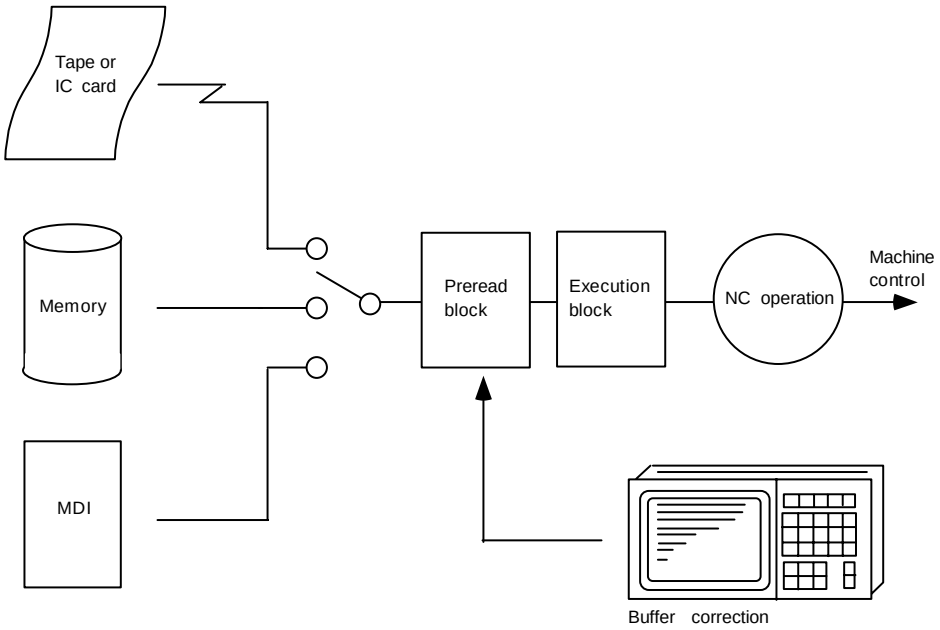
Display item	Explanation
N1 G00 X-345.678 Y345.678; N2 T1234; N3 S5000 M3; N4 G00 Z-100; N5 G01 X100.F500; N6 Y100.; N7 G02 X200.R200.;	The current work program being executed is displayed. This is the same as the POSITION screen display.
SPINDLE  Z-AX 	The spindle load and Z axis load can be displayed as a bar graph, using the user PLC.
WORK COUNT: □/□ Workpiece count Max. workpiece count	Workpiece count : Indicates count data of the number of workpieces. Workpiece count Max. value : The max. workpiece value set in #8003 WRK LIMIT value is displayed. Display range: 0~999999

2.2.1 Correcting the Buffer

(1) Outline

During automatic operation (memory or tape operation) or MDI operation, a block stop can be applied, and the next command can be corrected or changed.
When a program error occurs, the block in which the error occurred can be corrected without resetting the NC, and operation can be continued.

(Note) When running a machining program from the external memory, even if the buffer is corrected, the revisions are not reflected on the original program.



(2) Details

(a) The next command can be corrected in the following two cases.

- When single block stop is applied, and there is a machining program containing a next command to be corrected.
- During automatic operation, there is an error (program error) in the next command's machining program, and the program is stopped.

(b) During memory or MDI operation, not only the displayed buffer data but also the memory and MDI contents are corrected with the buffer corrections.

(c) If an error occurs in the preread block, the block in which the error occurred can be corrected.

(Note) The buffers on the second and third pages of the COORDINATE screen cannot be corrected.

(3) Operation method

During a single block stop or when a program error stop occurs, the buffer can be corrected with the following operations, and operation can be continued.

(a)

Select the first page of the COORDINATE screen.

(b)

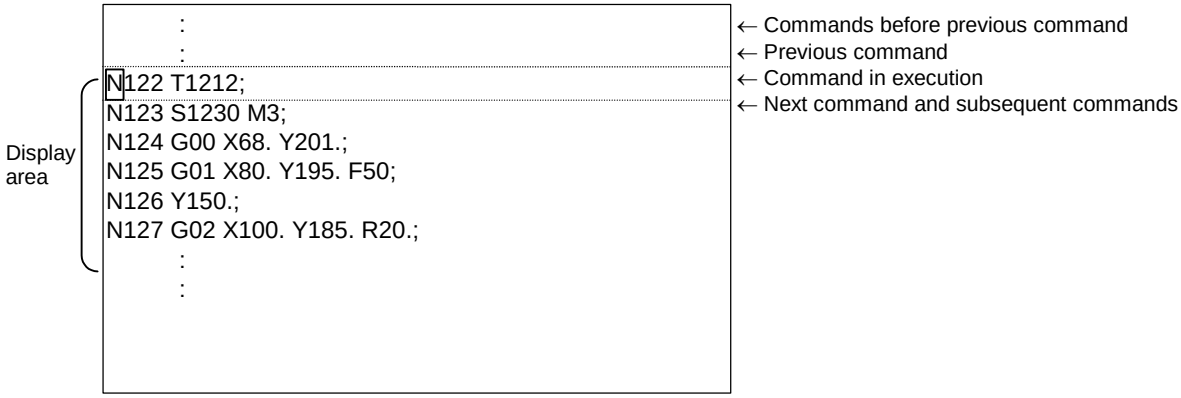
Press the one of the cursor keys (↓,↑,←,→) or tab keys (⇐,⇒).

☞ The buffer correction mode will be entered, and the buffer correction area will be highlighted.

* Buffer correction area (39 characters × 6 lines)

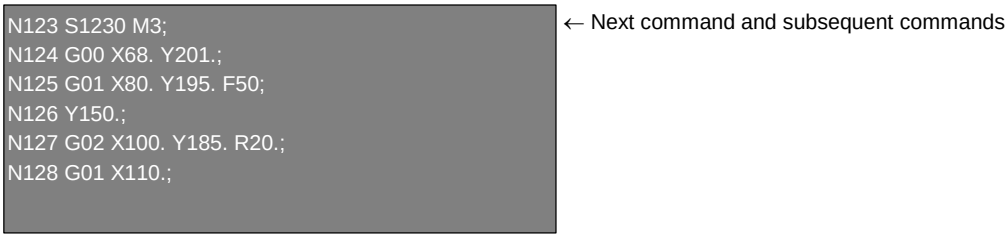
The normally executed program appears in this area. (Up to six lines will be displayed.)

[Buffer correction area in normal state]



When correcting the buffer, the display changes so that the next command is displayed at the head of the area, and the entire buffer correction area is highlighted. The cursor will initially flicker at the head of the next command. The cursor can be moved freely within the buffer correction area using the cursor keys. (6 lines)

[Buffer correction area during buffer correction]



(c)

Correct the program with the same method as editing a normal program.

(d)

Press the INPUT key. ☞ The buffer correction mode ends, and the corrected data is written into to program.
If a program error has occurred, the error display disappears.

(e)

Confirm that the corrected data is correct, and then restart. ☞ The program execution resumes from the currently stopped position.

(Example) An example of creating and executing the following program is given below.
If a program error (P62) occurs in the N125 block, the cause of the error is removed by correcting the buffer.

```
N121 G28 X0 Y0;
N122 T1212;
N123 S1230 M3;
N124 G00 X68. Y201.;
N125 G01 X80. Y195.;
N126 Y150.;
N127 G02 X100. Y185. R20.;
N128 G01 X110.;
N129 G01 X120.;
N130 G01 X130.;
```

(a)

Start automatic operation. ☞

1) A program error (P62) will occur after N124 is executed.

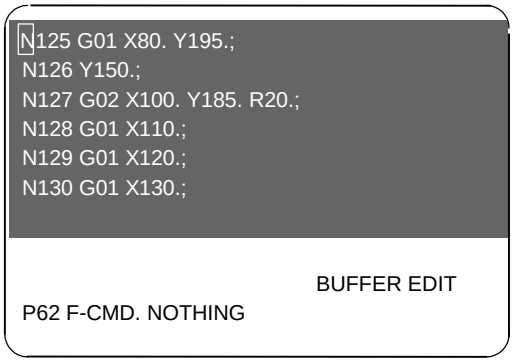
```
N124 G00 X68. Y201.;
N125 G01 X80. Y195.;
N126 Y150.;
N127 G02 X100. Y185. R20.;
N128 G01 X110.;
N129 G01 X120.;


P62 F-CMD. NOTHING
```

(b)

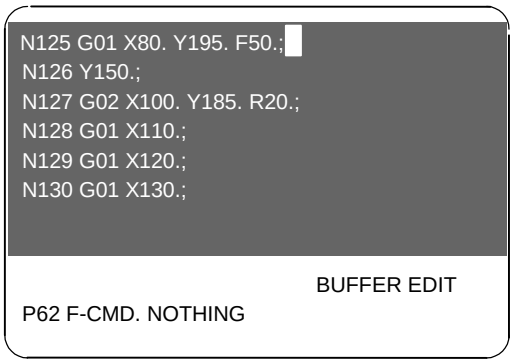
Press the → key. ☞

1) The head area of the program being executed will change to the buffer correction area when the cursor key is pressed.
(The buffer correction mode will be entered.)
2) The message "BUFFER EDIT" will appear.



(c)

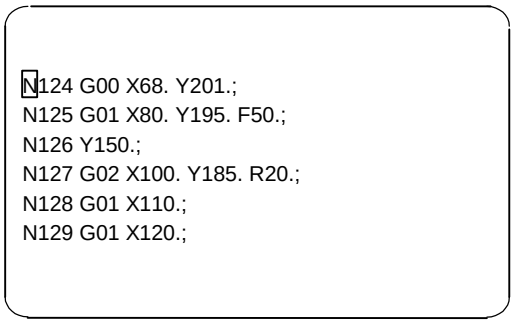
Insert "F50" at the end of the N125 line. ☞



(d)

Press the INPUT key. ☞

1) The buffer correction will end when the INPUT key is pressed, and the program being executed will display.
(The buffer correction mode will be canceled.)
2) The message "BUFFER EDIT" will disappear.
3) The program error (P62) will disappear.



(e)

Start automatic operation. ☞ Execution will resume from the N125 block.

(4) Supplement

(a) If there is no data for the next command during automatic operation start or automatic operation with tape, etc., the buffer correction mode will not be entered even if the cursor key is pressed. % is always inserted at the end of the memory or the next command data will not be lost.

Buffer correction possible
(memory operation)

N128 G01 X110.;
%

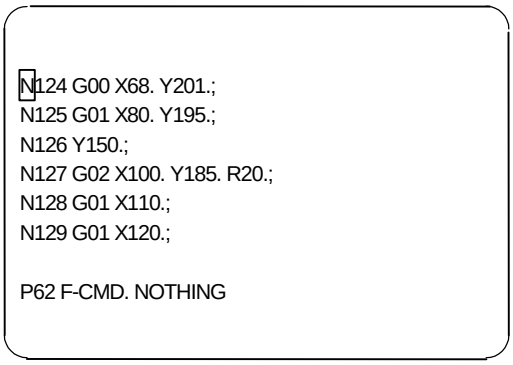
Buffer correction not possible
(tape operation)

N128 G01 X110.;

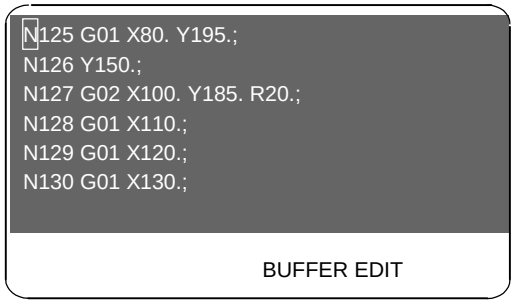
(b) The key operations for making corrections during the buffer correction mode are the same as the operations for editing the program. However, blocks other than those displayed in the buffer correction area cannot be displayed and operated by feeding the page with the NEXT PAGE or PREVIOUS PAGE keys or by scrolling with the cursor keys. The page feed and scrolling operations will be ignored.

Operation	Buffer correction	Program correction
Scroll	Not possible	Possible
Page feed/return	Not possible	Possible
Cursor movement	Possible	Possible
Character replacement	Possible	Possible
Character insertion	Possible	Possible
Character deletion	Possible	Possible
Block deletion	Possible	Possible

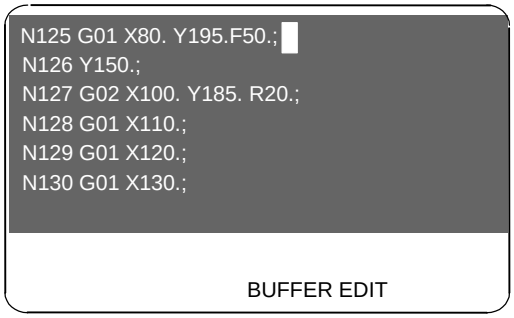
(c) Even if the buffer is corrected, if the [INPUT] key has not been pressed, the corrections can be returned to the original next command by pressing the [NEXT PAGE] or [PREVIOUS PAGE] key. The mode will remain the buffer correction mode.



Press the [→] key. ☞

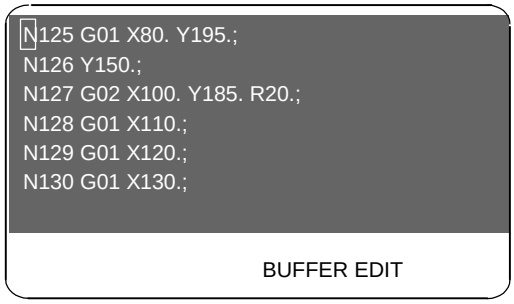


Insert F50. at the end of the N125 line. ☞



Press the [NEXT PAGE] or [PREVIOUS PAGE] key. ☞

1) Return to the state when buffer correction is started.

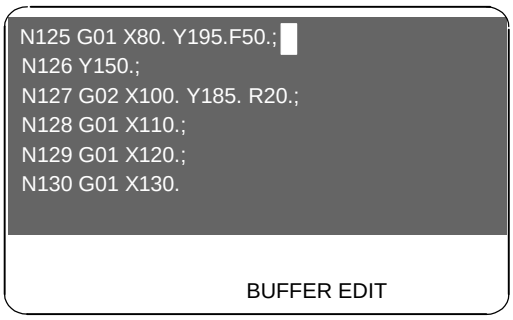


(Note) The [NEXT PAGE] and [PREVIOUS PAGE] keys function to cancel the edited details during buffer correction, so the page cannot be switched to the COORDINATE screen page 2 and following. Quit buffer correction to change the page.

(d) Buffer correction will be canceled if another screen is opened or reset is executed during the correction. The corrected details will not be reflected.

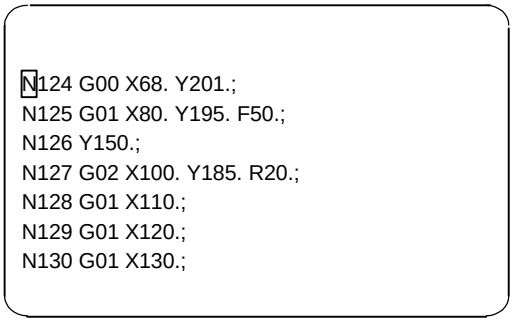
(e) Operation cannot be started during buffer correction. The "M01 Operation alarm 0013" will occur.

(f) If there is no ; (EOB) in the last block edited when INPUT is pressed, it will be added automatically.

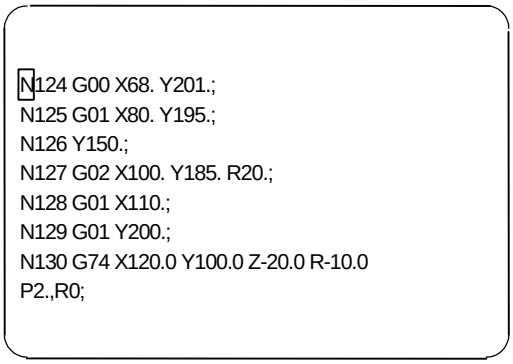


Press the INPUT key. ☞

1) ";" is added to the end of the N130 block.

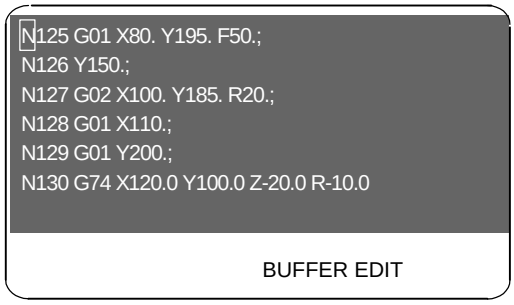


(g) When the buffer correction mode is entered, there may be cases when the program up to ; (EOB) does not fit in and only part of the program is displayed because the last block displayed in the buffer correction area is long.



Press the → key. ☞

1) "P2.,R0" of N130 is not displayed.

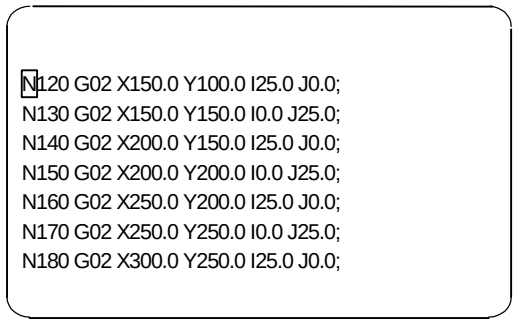


(Note) Handling when entire block is not displayed
The results will differ according to the state when INPUT key was pressed.

State	Results
; (EOB) is not added to the end of displayed data.	The section (P2.,R0;) not displayed will be the continuing part of the displayed section.
; (EOB) is added to the end of the displayed data.	The section (P2.,R0;) not displayed will become separate block.
The N130 block is deleted with the C.B CAN key.	During memory/MDI operation: The section (P2.,R0;) not displayed will also be deleted. During tape operation: Only the displayed section will be deleted, and the section not displayed will be kept as a separate block.

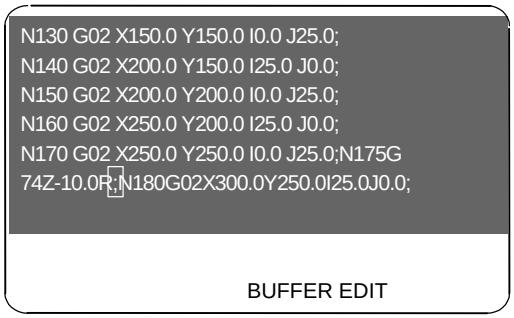
(h) The number of characters that can be input at once will be the total of the number of characters added (or deleted) with buffer correction and the number of characters in the other blocks displayed in the correction area. The maximum number will be 234 characters (39 characters × 6 lines). When adding data, characters exceeding the correction area's margin cannot be added.
In this case, press the [INPUT] key and quit the buffer correction once. Then, correct the buffer again. When buffer correction is started, a line return is added for each block and a space is added for each word, so there will be more space for adding characters.

(Example) Add "N175 G74 Z–10.0 R–5.0 P2.0 ;" after N170.

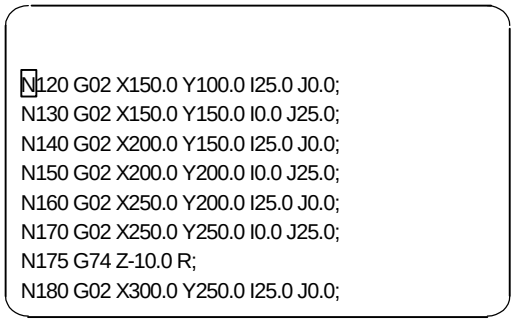


Press the [→] key. ☞

Insert "N175G74Z–10.0R;". ☞

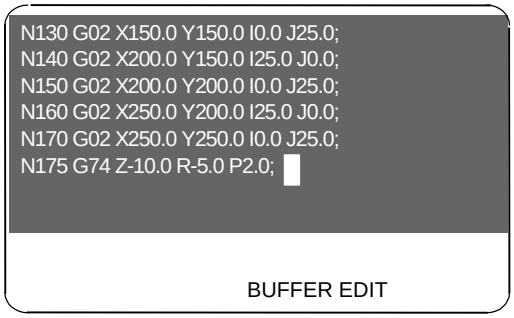


Press the [INPUT] key. ☞

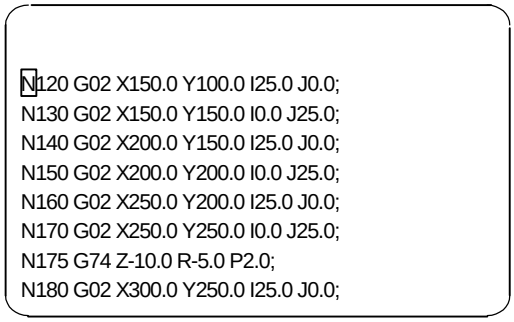


Press the → key.

Insert "–5.0P2.0" at the end of the N175 line.



Press the INPUT key.



(i) If an error occurs in the preread block and the buffer correction mode is entered, the block in which the error occurred will appear at the head of the area.

(j) The error cannot be canceled for a fixed cycle or compound fixed cycle that cannot be displayed. Programs that cannot be displayed include:
- Fixed cycles
- Compound fixed cycles
- Machine maker macros (when base specification parameter #1166 fixpro=0)
- When running program No. 9000 to 9999 while the program display lock is valid (base specification parameter #1122 pglk_c=1).

(k) When the program area selection (base specification parameter #1050 MemPrg) is set to 0 [program system same control specifications] and programs with the same numbers are run for the two systems, the buffer cannot be corrected. The message "CAN'T BUF. EDIT" will appear.

(l) The buffer cannot be corrected during IC card operation with M198. The message "CAN'T BUF. EDIT" will appear.

(m) Cases in which buffer cannot be corrected
The subprogram call command (M98) and return command (M99) will execute the next block to be executed (subprogram head and return designation) with one automatic start. Thus, the buffer for the subprogram's head and return destination blocks cannot be corrected.

(Example 1)

O10	O100
G28 XYZ	N100 G01 Y10. F5000.;
:	N110 G01 Z10.;
N10 G0 X50.;	M99;
M98 P100;	%
N11 G0 X100.;	
N12 G0 X150.;	
M02;	
%	

If automatic start is executed when the block is stopped with N10, the N100 block will be executed and will stop with N110. Thus, the buffer for N100 cannot be corrected.
If automatic start is executed when the block is stopped with N110, the N11 block will be executed and will stop with N12. Thus, the buffer for N11 cannot be corrected. (Note that if the buffer is corrected when the block is stopped before N10, then N11 can be corrected.)

Normal buffer correction of the subprogram's head is not possible, as shown in the above example. Note that if there is a G0/G1 command in the M98 block, the block will stop at that block. (Example: G0M98P100;) If a block containing only the sequence No. is created at the head of the subprogram, the program will stop at that block, so buffer correction of the following blocks is possible.

(Example 2)

```
O100
N1;          ← Stops here
G01Y10.F5000;
M99;
%
```

(6) Precautions

(a) When an error occurs during continuous operation, if the program processes the variables, etc., in one step, the display will start not from the error block but instead from the head of the variables in which there was a preread error block. Six lines of the program will be displayed.

Example) X100.;

```
#100=0;
#101=1;
#102=2;
#103=3;
#104=4;
#105=5;
```

← When the buffer correction mode is entered, these six lines will be displayed.

X-10000000; ← Example when this block has a program error (P35).

If the macro block has seven or more lines, the block in which the program error occurred will not be displayed.

(b) Edit lock C
When edit lock C is valid (base specification parameter #1121 edlk_c=1), buffer correction of program numbers 9000 to 9999 is not possible. The "E16 EDIT LOCK C" error will occur.
When the base specification parameter #1122 pglk_c=1/2, buffer correction of program numbers 9000 to 9999 is not possible. The "E16 EDIT LOCK C" error will occur.

(Note) When either base specification parameter #1121 edlk_c or #1122 pglk_c is set, the power must be rebooted.

(c) Data protection key
Buffer correction is possible when data protection key 1 (*KEY1: Y238) is ON.
Buffer correction is possible when data protection key 2 (*KEY2: Y239) is ON.
Buffer correction is not possible when data protection key 3 (*KEY3: Y23A) is ON. The message "DATA PROTECT" will appear.

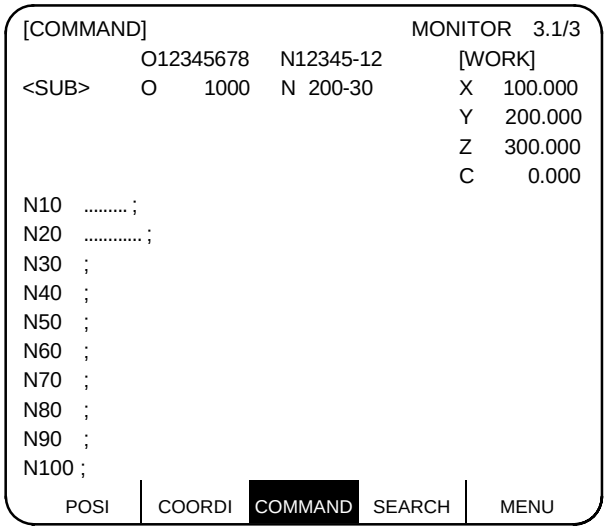
(d) Edit lock B
When edit lock B is valid (control parameter #8105 EDIT LOCK B = 1), buffer correction of program numbers 8000 to 9999 is not possible. The "E15 EDIT LOCK B" error will occur.

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 PAGE] key or [PREVIOUS PAGE] key.

2.3.1 Execution Program Monitor

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

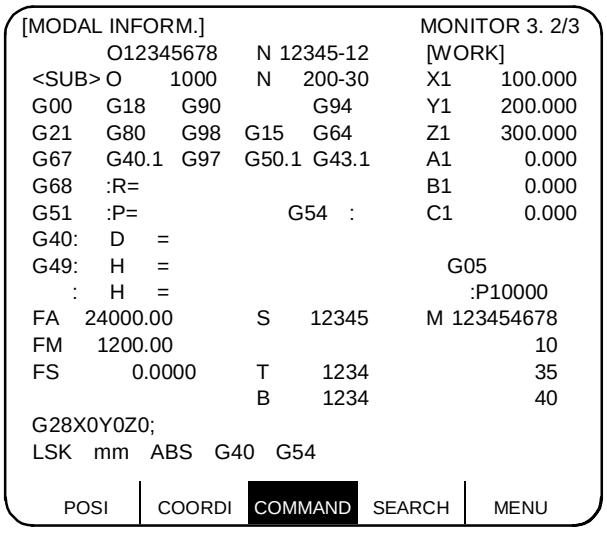


Display item	Explanation
O12345678 N12345-12	The currently executing program number, sequence number, and block number are displayed.
<SUB> O1000 N200-30	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.
N10 ..... ; N20 ..... ; N30 ; ⟆ 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. The read data is displayed also during tape running in the above way.
[WORK] X 100.000 Y 200.000 Z 300.000 C 0.000	The workpiece coordinates in the workpiece coordinate system being currently executed are displayed.

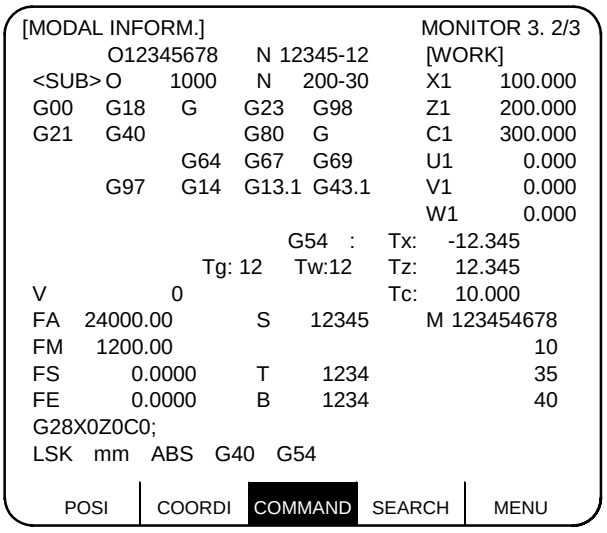
2.3.2 Execution Modal Monitor

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

[M system]



[L system]

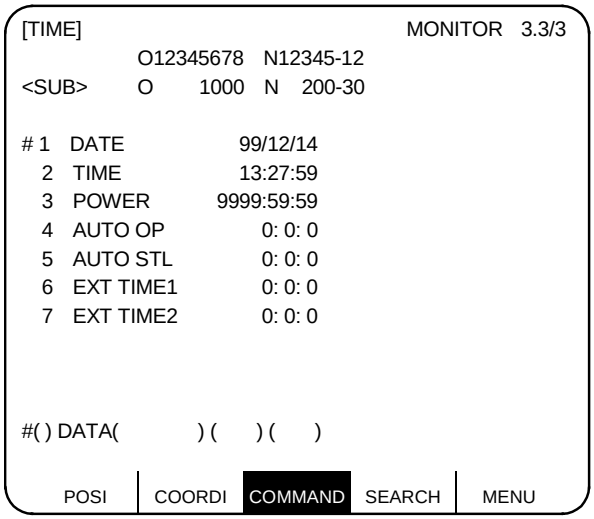


Display item	Explanation
O12345678 N12345-12	The currently executing program number, sequence number, and block number are displayed.
<SUB> O1000 N200-30	When a subprogram is being executed, the program number, sequence number, and block number of the subprogram are displayed.
[WORK] X1 100.000 Y1 200.000 Z1 300.000 :	The workpiece coordinates in the workpiece coordinate system being currently executed are displayed. Same as the POSITION screen.
[MODAL INFORM.] G00.....G94 ((G67.....G43.1	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 program.
G40:D G49:H	The tool radius compensation modal and offset number and wear amount are displayed. The tool length offset modal and offset number and wear amount are displayed.

Display item	Explanation
G68: R= (M system) G51: P= (M system) G05: P10000 (M system)	The rotation angle for the program coordinate rotation command is displayed.
	The scaling magnification is displayed.
	The high-speed machining mode and high-speed high-accuracy mode are displayed. High-speed machining mode 3 (G05 : P3) High-speed high-accuracy control II (G05 : P10000) SSS control valid (G05 : P10000S) High-speed high-accuracy control I (G05.1 : Q1) Spline interpolation (G05.1 : Q2) Mode OFF (G05 : P0)
Tx: –12.345 (L system) Tz: 12.345 (L system) Tc: 10.000 (L system)	The total of the X axis, Z axis and additional axis' tool length and wear compensation amounts for the tool being used is displayed.
Tg: 12 (L system) Tw: 12 (L system)	The tool length offset No. is displayed. The wear compensation No. is displayed.
v (L system)	The constant surface speed spindle rotation speed is displayed.
FA 24000.00	The program command asynchronous feedrate modal value currently being executed is displayed. (mm/min)
FM 1200.00	The manual feedrate is displayed. (mm/min)
FS 0.0000	The program command synchronous feedrate modal value currently being executed is displayed. (mm/rev)
FE 0.0000 (L system)	The thread lead command synchronous feedrate modal value currently being executed is displayed. (mm/rev)
S 12345	The modal value of the current program command S being executed is displayed.
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-100.234~	The current program block being executed is displayed.

2.3.3 Total Integrating Time Display

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



Display item			Explanation
O12345678 N1234-12			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	99/12/14	The date and time are set and displayed. year/month/day hour : min : sec
2	TIME	13:27:59	
3	POWER ON	9999:59:59	The total integrating time in each operation state is displayed.
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	

(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.

#3 POWER ON 0: 0: 0

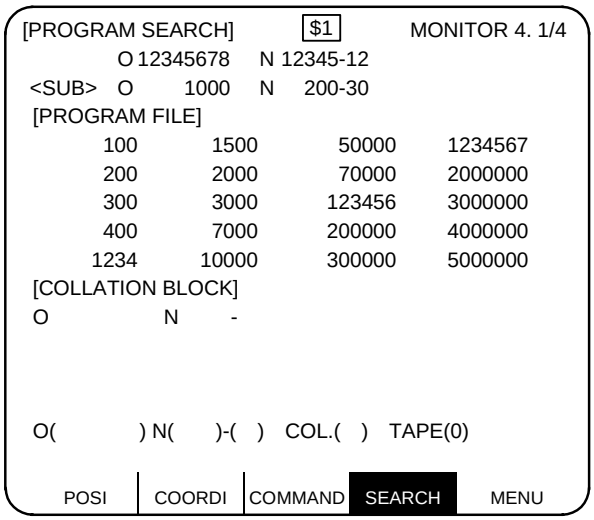
DATE : Set date (set "YEAR" in last 2 digits of Gregorian calendar)
TIME : Set time in 24-hour mode.
POWER ON : Total integrating time of the time from control unit power ON to OFF.
AUTO OP : Total integrating time of the work time from AUTO STL button pressing in the memory (tape) mode to M02/M30 or reset button pressing.
AUTO STL : Total integrating time during automatic starting from AUTO STL button pressing in the memory (tape) mode or MDI to feed hold stop, block stop, or reset button pressing.
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.

2.4 PROGRAM SEARCH

When the menu key [SEARCH] is pressed, the PROGRAM SEARCH screen is displayed.

The SEARCH screen enables you to call the program number, sequence number, and block number for automatic operation from the machining programs registered in memory (or on paper type).



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.
[COLLATION BLOCK] O N -	The program position for compare stop is displayed.

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.

(Note 1) When using the 2-part system, the system name of the currently selected system is displayed as $1 (system 1) and $2 (system 2). This is not displayed when using a 1-part system. (Only L system)

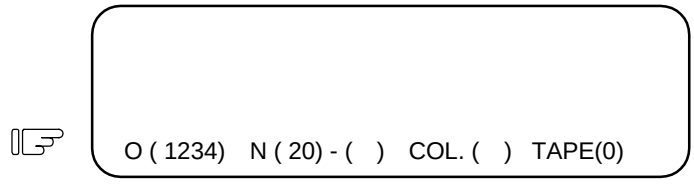
(Note 2) When using the 2-part system, the details displayed in the list of machining program numbers can be switched with the parameters.

#1050 MemPrg	Details
0, 2, 4, 6	The numbers of the machining programs registered in the memory common for the systems are listed.
1, 3, 5, 7	The numbers of the machining programs registered in the memory for the selected system are listed.

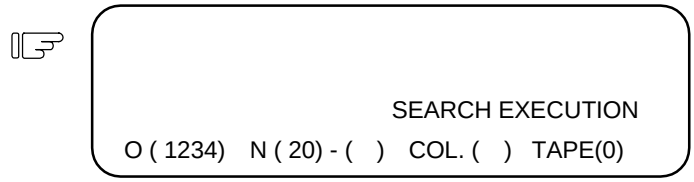
2.4.1 Memory Search

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.

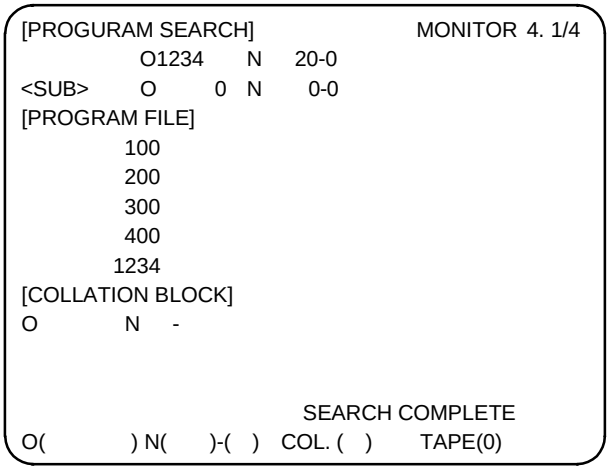
Set the program number to be called. Set the sequence number and block number as required.
(**Example**)
To call O1234 N20 block,
O (1 2 3 4) N (2 0) - ( )
COL. ( ) TAPE (0)



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.

(Note 3) When using the 2-part system, the method for calling the program No., sequence No. and block No. for automatic operation can be switched with the parameters.

#1050 MemPrg	#1285 ext21/bit1	Details
0, 2, 4, 6	-	Selected system: The machining program registered in the memory common for the systems is called with the designated program No., sequence No. and block No.
		System that is not selected: The called machining program is held.
1, 3, 5, 7	OFF	Selected system: The machining program registered in the memory for the selected system is called with the designated program No., sequence No. and block No.
		System that is not selected: The called machining program is held.
	ON	Selected system: The machining program registered in the memory for the selected system is called with the designated program No., sequence No. and block No.
		System that is not selected: The machining program registered in the memory for each system is called with the designated program No.

(Note 4) When using the 2-part system and the same number batch search for all system programs is valid, the presence of machining programs for each system is searched for in the memory in the following manner.

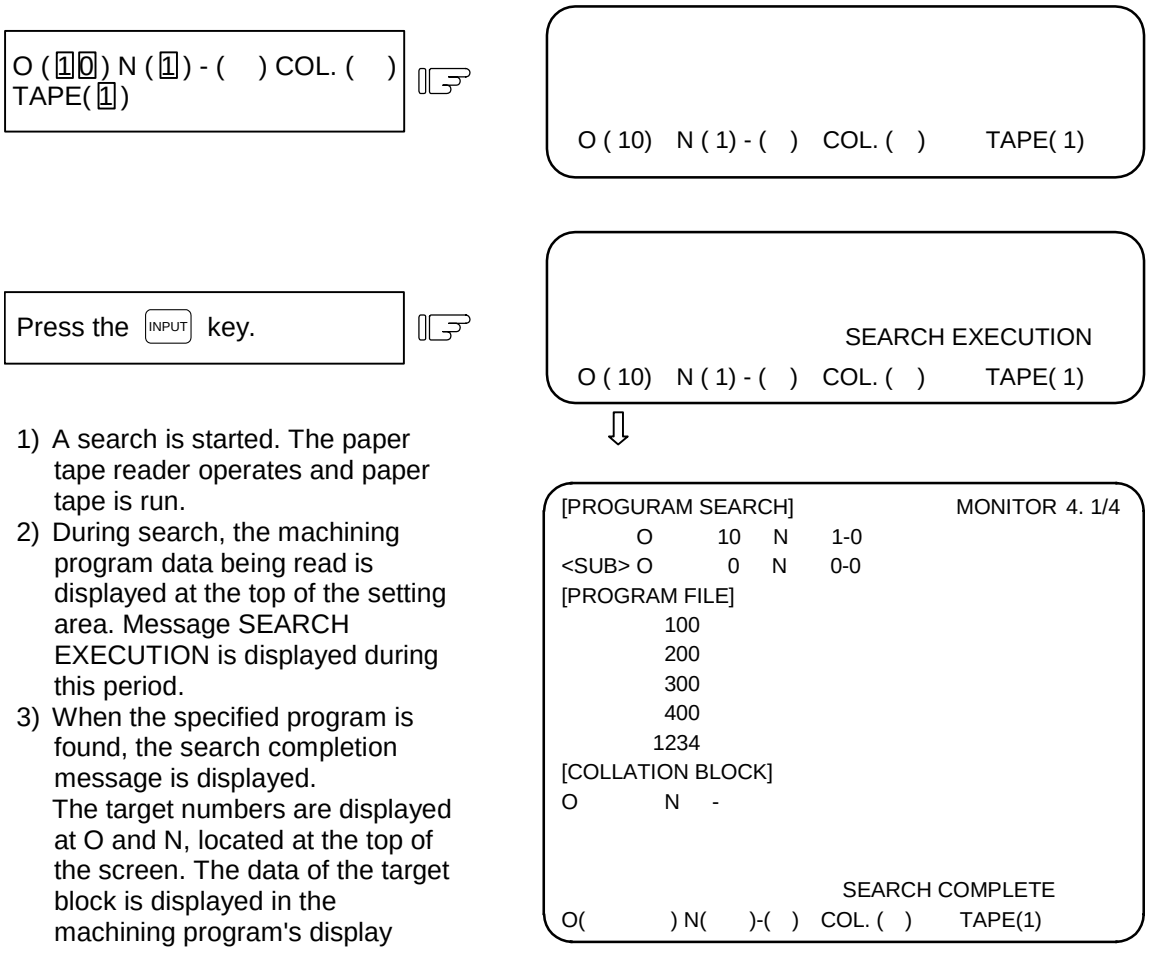
Presence of program		Operation
System 1	System 2	
Yes	Yes	Memory search is carried out simultaneously for the System 1 and System 2 machining programs.
Yes	No	Memory search is carried out for the System 1 machining program, and then an error (E14) occurs.
No	Yes	Memory search is carried out for the System 2 machining program, and then an error (E14) occurs.
No	No	An error (E14) occurs.

2.4.2 Tape Search

If processing is desired to be executed from a halfway position on paper tape when running the machining program using paper tape, the tape can be searched for the sequence number and other information.
Before using the tape reader, match tape reader setting and control unit input/output parameter setting. Set the input/output basic parameters and input/output device parameters on the DATA IN/OUT screen.
Mount the paper tape on the tape reader. Then, select tape operation mode and execute the following search:

(1) Set the target program number in O ( ). Set the target sequence number in N ( ). Set the target block number in - ( ). Set "1" in TAPE ( ).
(2) Press the INPUT key.

(Example)



1) A search is started. The paper tape reader operates and paper tape is run.
2) During search, the machining program data being read is displayed at the top of the setting area. Message SEARCH EXECUTION is displayed during this period.
3) When the specified program is found, the search completion message is displayed.
The target numbers are displayed at O and N, located at the top of the screen. The data of the target block is displayed in the machining program's display area.

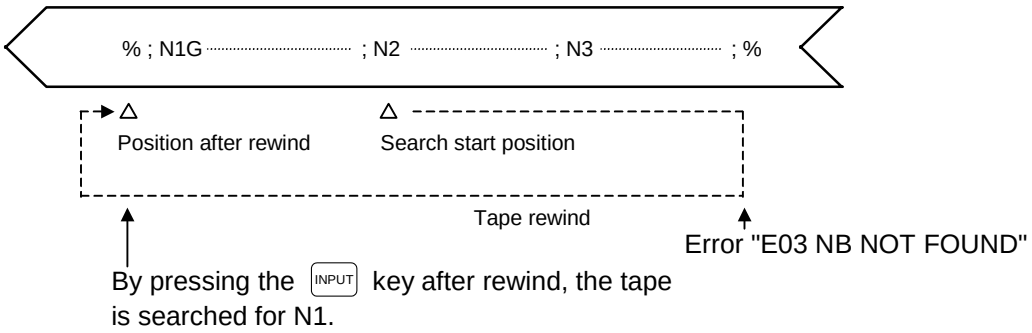
(Note 1) If the tape contains a $ mark during tape operation, a program error (P32) will occur at the $ mark.

(Supplements)

(1) Search starts in the position set in the tape reader. (In the label skip status, control jumps to the first EOB.)
(2) After the search is completed, the searched block is read and the tape reader stops.
(3) If the NC is reset during search, the search stops. If the NC is reset after search is completed, the unsearched status returns.
(4) If the specified block has not been found after the data to the EOR is read, the following message is displayed: "E03 NB NOT FOUND"
If control parameter "% RWD (SEARCH)" is OFF, the tape will stop at the EOR of the program end. If the parameter is ON, the tape will be rewound to the EOR of the program head and will stop there.

(Note) Even if control parameter "% RWD (SEARCH)" is ON, the tape will not be rewound if I/O DEVICE PARAM "REWIND CODE" has not been set correctly.
The rewind code depends on the I/O device used. Refer to the I/O device manual for rewind code details.

(Example) When the tape was searched for N1 from a halfway position, the tape end was reached before N1 was found. (When "% RWD (SEARCH)" is ON)



(5) If the target program number is not specified, the tape will be searched for only N and B. This does not relate to the program numbers in the tape.
(6) If the INPUT key is pressed after normal tape search is completed and other information including another NB is set, search will be executed. If a block stop status is entered after search is completed and the automatic running status is entered by pressing the automatic start button once, tape search will not be executed.
(7) After tape search is completed, "1" is retained in the setting field of TAPE ( ). Thus, the value does not need to be set for each tape search. Only when memory search is desired to be executed, set "0" in the setting field of TAPE ( ).
(8) The "LSK" display is cleared when the first EOB is read. It is displayed at reset or EOR read time.

2.4.3 Compare Stop

The single block stop state can be applied at a random block without turning the "SINGLE BLOCK" switch ON.
By using compare stop, the shape machined up to the designated block can be easily compared and machining can be resumed.

(1) Setting compare stop

Designate the program No., sequence No. and program No., and set 1 in COL. ( ).
(Example) To compare stop at 01234N20-3
O (1234) N (20) - (3)
COL. (1) TAPE ( )

☞

O (1234) N (20) - (3) COL. (1) TAPE( )

To execute compare stop with an MDI program, set 0 (zero) for the program No.

Press the INPUT key.

☞

[COLLATION BLOCK] COLL. EXEC
O 1234 N 20 - 3

O ( ) N ( ) - ( ) COL. ( ) TAPE ( )

The program No. ("MDI" for MDI), sequence No., block No., and the message "COLL. EXEC" will appear at [COLLATION BLOCK]. The setting areas will change to blanks.

Press the "CYCLE START" switch.

☞

[COLLATION BLOCK]
O N -

O ( ) N ( ) - ( ) COL. ( ) TAPE ( )

1) Operation will start.
2) When the designated block is reached, the signal block stop state will be applied after executing that block.
3) When the single block stop mode is established by compare stop, the program number, sequence number and block number appearing in [COLLATE BLOCK] as well as the "COLL. EXEC" message are cleared.

(2) Canceling compare stop

Set 0 in COL. ( ).
O ( ) N ( ) - ( )
COL. (0) TAPE ( )

[COLLATION BLOCK] COLL. EXEC
O 1234 N 20 - 3

O ( ) N ( ) - ( ) COL. (0) TAPE ( )

Press the INPUT key.

The program number, sequence number and block number settings as well as the "COLL. EXEC" message are cleared.

[COLLATION BLOCK]
O N -

O ( ) N ( ) - ( ) COL. (0) TAPE ( )

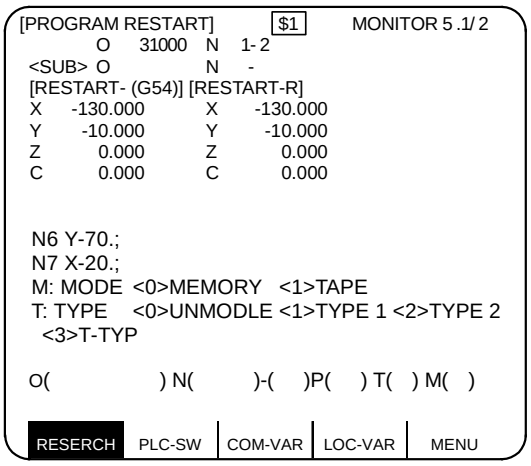
(3) Precautions

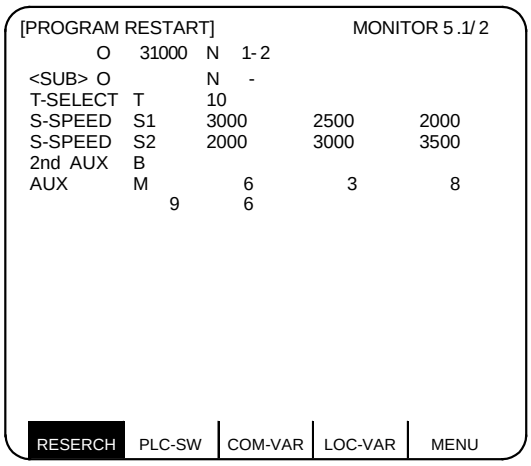
1) When there are several identical sequence numbers and block numbers in a program, compare stop results after the first corresponding block in the sequence of execution has been executed.
2) The compare stop setting is canceled in the following cases.
   - When compare stop has been performed
   - When "0" has been set in the "COL. ( )" on the SEARCH screen
   - When the reset mode has been established
3) If only the program number is set, compare stop will take place at the head of the program only when there is a program number at the first line.
4) When the program and sequence numbers have been set and the block number has not been set, the block number is considered to be "0".
5) Compare stop cannot be performed for blocks being executed or blocks already read into the preread buffer. (Compare stop can be set.)
6) Compare stop cannot be canceled if the block in which compare stop is set is being executed or has already been read into the preread buffer. (Cancellation of compare stop can be set.)
7) Even if a block not included in the execution program is assigned, no check is conducted to verify whether it exists in the program.
8) Compare stop is not performed in the tapping mode.
9) Compare stop is possible in a subprogram, but is not possible in a machine maker macro program.
10) If compare stop is set for a fixed cycle block, compare stop will be executed after the positioning block is completed.
11) Compare stop is possible while the program display is locked (compare stop in an address 9000 program).
12) If compare stop is set for M98 program call, compare stop will be executed at the M98 block.
13) When using the 2-part system, the methods for searching the compare stop block can be switched with the parameters.

#1050 MemPrg	Details
0, 2, 4, 6	The compare stop block in the machining program registered in the memory common for the systems is searched with the designated program No., sequence No. and block No.
1, 3, 5, 7	The compare stop block in the machining program registered in the memory for the selected system is searched with the designated program No., sequence No. and block No.

2.5 Resuming the Program

The PROGRAM RESTART screen will open when the menu key (RESEARCH) is pressed.





The program restart function is used to resume machining after the machining program has been stopped midway. The program and block to be restarted are searched for, and machining is resumed from that block.

The restart types include type 1, type 2 and type 3 (T command restart).

Restart method	Details
Restart type 1	After machining is reset due to a tool breakage, etc., machining is restarted from the designated sequence number and block number.
Restart type 2	After machining program is stopped due to a halt and the power is turned OFF and ON, machining is restarted from the designated sequence number and block number.
Restart type 3 (T command restart)	After the machining program is stopped due to tool breakage, etc., the T command block executed last in the halted program is searched for and machining is restarted from the next block.

(Note) Restart type 3 is valid only with the lathe system.

Restart type 1 and restart type 2 include types A and B.

Restart method	Type	Details
Restart type 1/2	Type A	The search is executed only in the machining program number having the designated sequence number and block number. The machining program number cannot be omitted.
	Type B	The restart search is executed for the currently searched machining program, so the machining program to be restart searched cannot be input. The designated sequence number and block number are searched from all programs within the currently searched program number. (Sequence number and block number in the subprogram are searched when there are subprograms.)

Display item	Details	
O 31000 N 1- 2 <SUB> O N -	This displays the restart searched position (program No., sequence No., block No.). If a subprogram is searched, those numbers also display.	
[RESTART- (G54)] X -130.000 Y -10.000 Z 0.000 C 0.000	This displays the remaining distance when the restart search is completed.	
[RESTART-R] X -130.000 Y -10.000 Z 0.000 C 0.000	This displays the position on the local coordinate system when the restart search is completed.	
N6 Y-70.; N7 X-20.;	This displays two blocks of the restart searched program.	
O( ) N( ) -( )	<Setting range> 1 to 99999999 0 to 99999 0 to 99	These set the program number, sequence number and block number to be searched. For type A, the program No. cannot be omitted. For type B, the program No. cannot be input.
P( )	0 to 9999	This sets the number of times the search block appears. When, for instance, a block in the subprogram is to be searched, the search block will be executed a multiple number of times when that subprogram is called for a similar number of times, and so the number of times the block is to be executed is set here. There is no need to set this when a one-time execution is to be searched or when the search block is to be executed only once. If 0 is set, a one-time execution is searched.
T( )	0	This designates an unmodal search.
	1	This designates a type 1 restart search.
	2	This designates a type 2 restart search.
	3	This designates a type 3 restart search. (This designates T command restart.)
M( )	0	This designates memory search.
	1	This designates tape search.
T-SELECT T	This displays the tool command value.	
S-SPEED S1	This displays the 1st spindle rotation speed command value.	
S-SPEED S2	This displays the 2nd spindle rotation speed command value.	
2nd AUX B	This displays the last four digits of the 2nd miscellaneous function command value.	
AUX M	This displays the last four digits of the miscellaneous function command.	

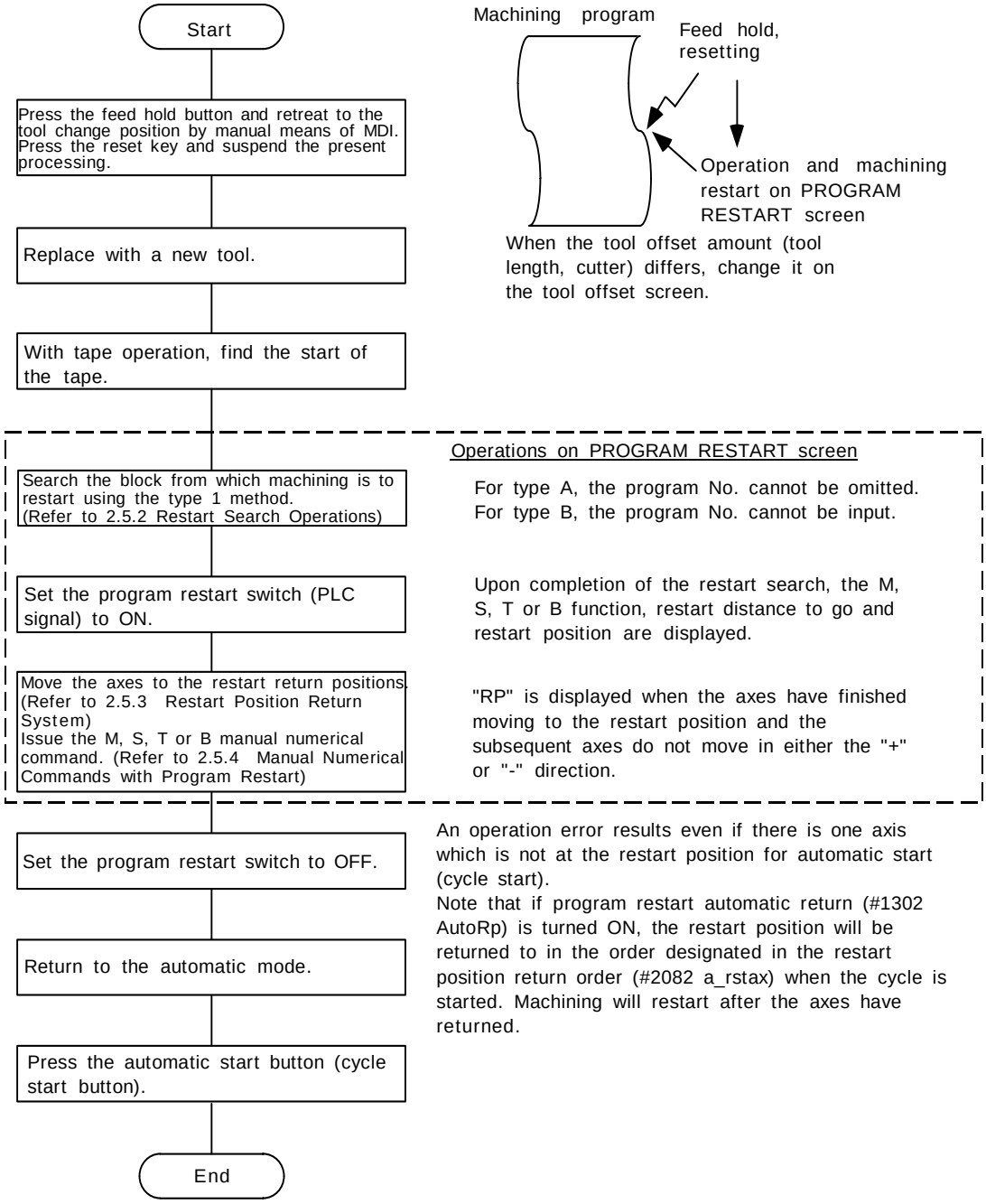
(Note 1) Restart type 3 is valid only with the lathe system.
(Note 2) When using the 2-part system, the system name of the currently selected system is displayed as $1 (system 1) and $2 (system 2). This is not displayed when using a 1-part system. (Only L system)
(Note 3) When using the 2-part system, the methods for searching for the program and block to be restarted can be switched with the parameters.

#1050 MemPrg	#1285 ext21/bit0	Details
0, 2, 4, 6	-	The machining program block registered in the memory common for systems is searched.
1, 3, 5, 7	OFF	The machining program block registered in the memory for the selected system is searched.
	ON	The machining program block registered in the memory for the selected system is searched. If the subprogram for the selected system is empty, the corresponding block is searched from the program with the same number saved in the $1 memory. (Only type B)

2.5.1 Operation Sequences for Program Restart

There are two restart methods, type 1 and type 2.

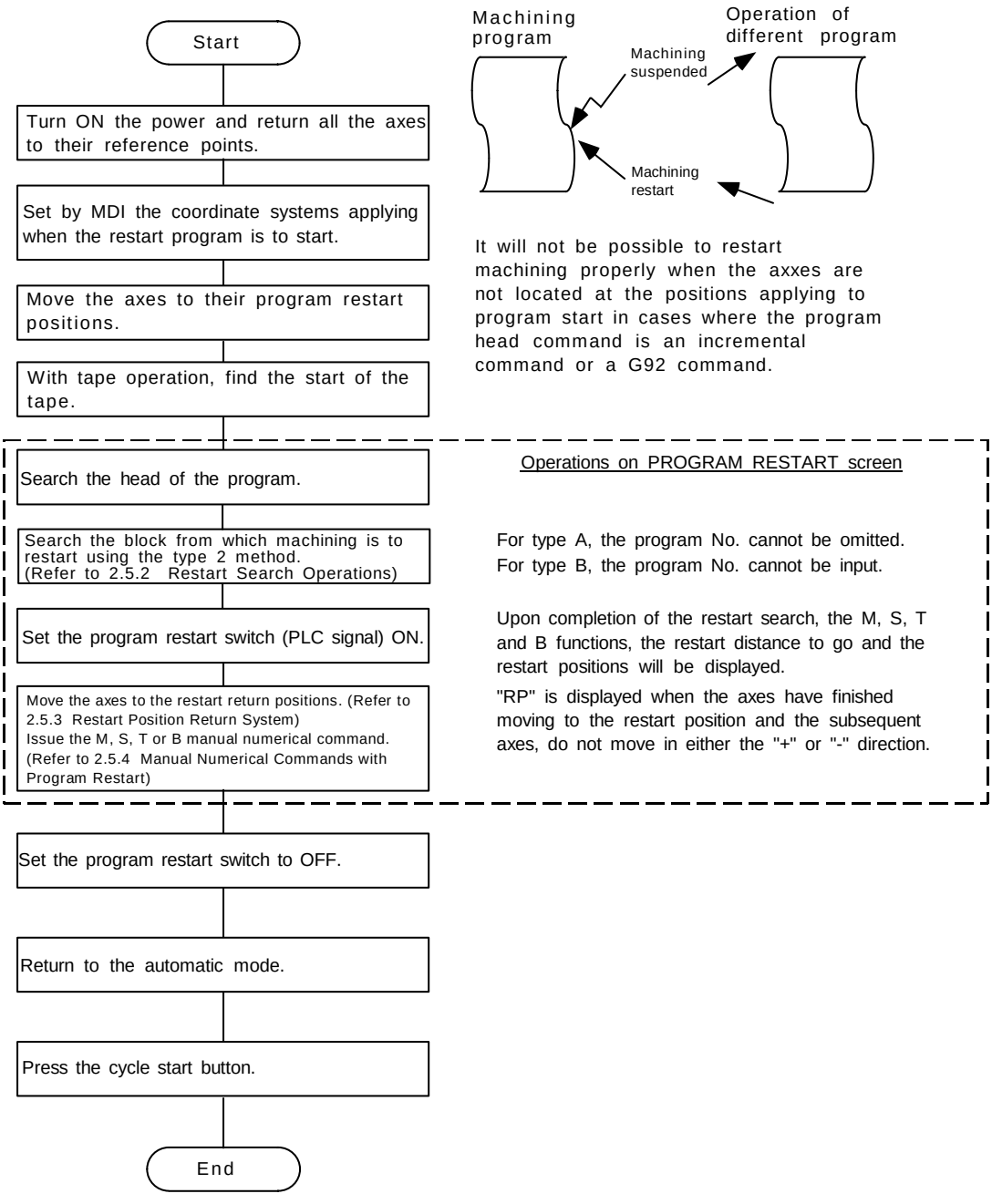
(1) Restart type 1
When feed hold and resetting due to a broken tool, etc.



(2) Restart type 2

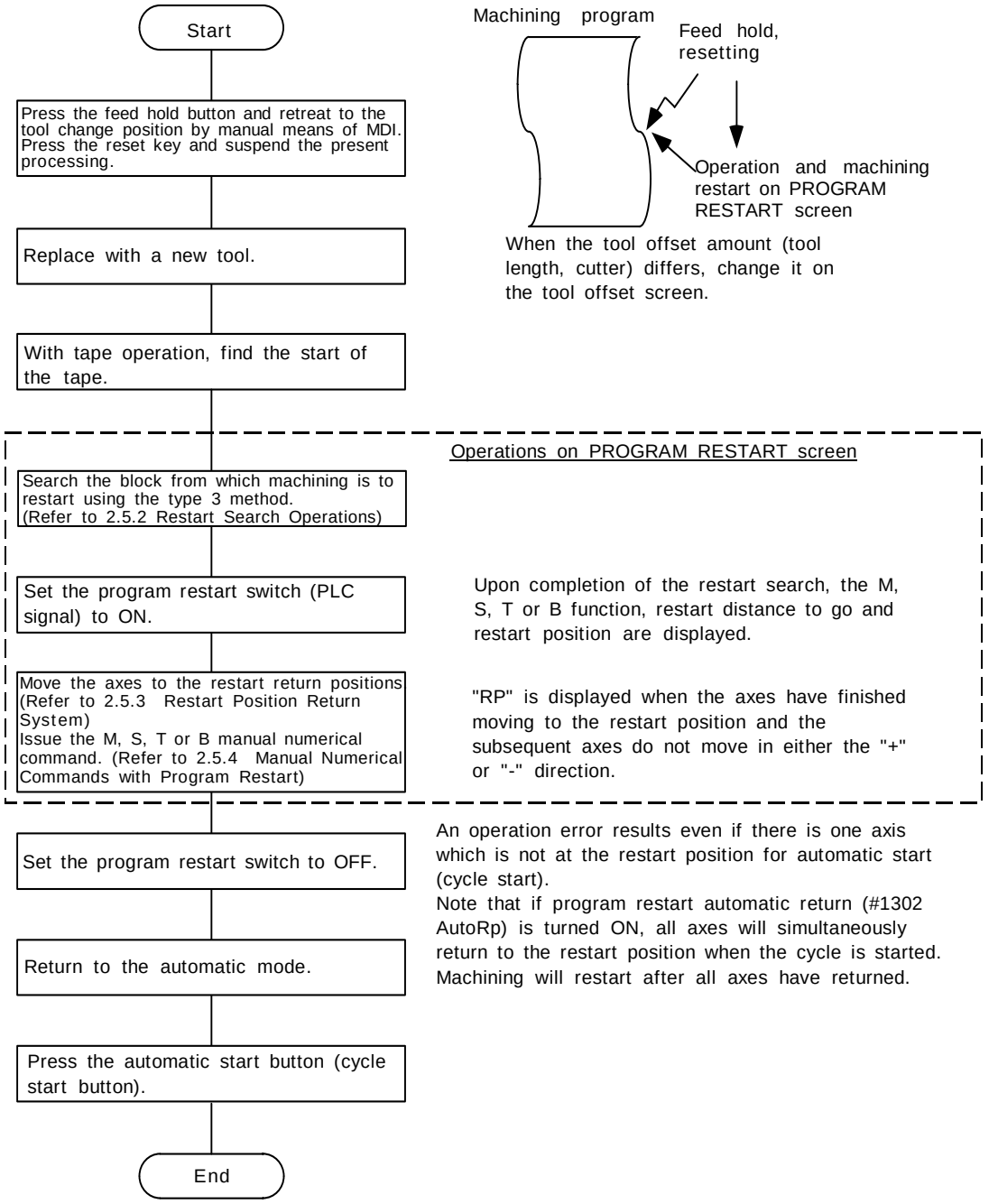
When a machining program, which differs from the machining program to be restarted, has been operated in the memory or tape mode prior to the restart search of the machining program to be restarted, and when the coordinate systems applying during the previous automatic operation and the systems applying during machining restart are to be changed.

The operating sequence for type 2 is the same as that for type 1 although before the restart search all the settings of the coordinate systems must be made before the machining program is operated. The main program to be restarted must be searched for just before executing restart start.



(3) Restart type 3 (T command restart)

To restart machining after stopping the machining program due to a tool breakage, etc., search for the T command block executed last in the stopped program.



(Note) Restart type 3 is valid only with the lathe system.

2.5.2 Restart Search Operations

(1) Type 1 restart search

a) Type A (Standard specifications)

Assign the block at which machining is to restart and proceed with the type 1 search.
(Example) To restart from the 01000 N6 block
O (1 0 0 0) N (6) - (0)
P (1) T (1) M (0)

```
M: MODE   <0> MEMORY      <1> TAPE
T: TYPE   <0> UNMODLE     <1> TYPE 1   <2> TYPE 2
 <3>T-TYP
O (    1000)   N (   6) - ( 0) P (         1) T(1) M(0)
```

Press the INPUT key.

```
M: MODE   <0> MEMORY      <1> TAPE
T: TYPE   <0> UNMODLE     <1> TYPE 1   <2> TYPE 2
 <3>T-TYP                 RESEARCH EXECUTION
O (    1000)   N (   6) - ( 0) P (         1) T(1) M(0)
```

The "RESEARCH EXECUTION" message appears during the search and upon its completion the "RESEARCH COMPLETE" message is displayed.

```
[PROGRAM RESTART]                          MONITOR 5 .1/ 2
          O    1000   N   5- 0
<SUB> O              N    -
[RESTART- (G54)]   [RESTART-R]
X  -130.000        X   -150.000
Y   -10.000        Y   -150.000
Z     0.000        Z     0.000
C     0.000        C     0.000


N6 Y-70.;
N7 X-20.;
M: MODE   <0> MEMORY      <1> TAPE
T: TYPE   <0> UNMODLE     <1> TYPE 1 <2> TYPE 2
 <3>T-TYP                 RESEARCH COMPLETE
O (         )   N (    ) - (   ) P (         1) T(   ) M(   )
```

b) Type B

Assign the block at which machining is to restart and proceed with the type 1 search.

(Example) To restart from the 01000 N6 block

O ( ) N (6) - (0)

P (1) T (1) M (0)


```
M: MODE   <0> MEMORY      <1> TAPE
T: TYPE   <0> UNMODLE     <1> TYPE 1   <2> TYPE 2
 <3>T-TYP
O (          )   N (   6) - ( 0) P (         1) T(1) M(0)
```

Press the INPUT key.


```
M: MODE   <0> MEMORY      <1> TAPE
T: TYPE   <0> UNMODLE     <1> TYPE 1   <2> TYPE 2
 <3>T-TYP                 RESEARCH EXECUTION
O (          )   N (   6) - ( 0) P (         1) T(1) M(0)
```

The "RESEARCH EXECUTION" message appears during the search and upon its completion the "RESEARCH COMPLETE" message is displayed.

```
[PROGRAM RESTART]                              MONITOR 5 .1/ 2
          O    1000   N    5- 0
<SUB> O              N     -
[RESTART- (G54)]   [RESTART-R]
X   -130.000       X    -150.000
Y    -10.000       Y    -150.000
Z      0.000       Z       0.000
C      0.000       C       0.000


N6 Y-70.;
N7 X-20.;
M: MODE   <0> MEMORY      <1> TAPE
T: TYPE   <0> UNMODLE     <1> TYPE 1 <2> TYPE 2
 <3>T-TYP                 RESEARCH COMPLETE
O (          )   N (    ) - (   ) P (          ) T(   ) M(   )
```

(2) Type 2 restart search

The program and block to be restarted are searched for with type 2.

(Example) To restart from block (a) in the following program.

(Program example)

```
O2000; ..... Main program
N1 G91 G28 X0 Y0;
N2 G90 G54 G00 X0 Y0 M98 P3000;
N3 G55 G00 X0  Y0 M98 P3000; ..... (a)
N4 M02;
%

O3000; ..... Subprogram
N1 G42 G01 X-10. Y-10. D05 F1000;
N2 X-40.;
N3 Y-40.;
N4 X-10.;
N5 Y-10.;
N6 G40 X0 Y0;
N7 M99;
%
```

a) Type A (Standard specifications)

Conduct a unmodal search to locate the head of the machining porgram.
O (2 0 0 0) N ( ) - ( )
P ( ) T (0) M (0)

```
M: MODE    <0> MEMORY      <1> TAPE
T: TYPE    <0> UNMODLE     <1> TYPE 1   <2> TYPE 2
 <3>T-TYP
O (    2000)   N (    ) - (   ) P (          ) T(0) M(0)
```

Press the INPUT key.

```
M: MODE    <0> MEMORY      <1> TAPE
T: TYPE    <0> UNMODLE     <1> TYPE 1   <2> TYPE 2
 <3>T-TYP                  SEARCH EXECUTION
O (    2000)   N (    ) - (   ) P (          ) T(0) M(0)
```

The "SEARCH EXECUTION" message appears during the search and upon its completion the "SEARCH COMPLETE" message is displayed.

⇩

```
[PROGRAM RESTART]                          MONITOR 5 .1/ 2
          O     2000    N    0 - 0
<SUB> O                 N     -
[RESTART- (G54)]   [RESTART-R]
X                    X
Y                    Y
Z                    Z
C                    C


O2000;
N1 G91 G28X0 Y0;
M: MODE    <0> MEMORY      <1> TAPE
T: TYPE    <0> UNMODLE     <1> TYPE 1 <2> TYPE 2
 <3>T-TYP                  SEARCH COMPLETE
O (          )   N (    ) - (   ) P (          ) T(   ) M(   )
```

Assign the block at which machining is to restart and proceed with the type 2 search.

O (3 0 0 0) N (0) - (0)
P (2) T (2) M (0)

```
M: MODE    <0> MEMORY      <1> TAPE
T: TYPE    <0> UNMODLE     <1> TYPE 1   <2> TYPE 2
 <3>T-TYP
O (    3000)   N (   0) - ( 0) P (         2) T(2) M(0)
```

Press the INPUT key.

```
M: MODE    <0> MEMORY      <1> TAPE
T: TYPE    <0> UNMODLE     <1> TYPE 1   <2> TYPE 2
 <3>T-TYP                  RESEARCH EXECUTION
O (    3000)   N (   0) - ( 0) P (         2) T(2) M(0)
```

The "RESEARCH EXECUTION" message appears during the search and upon its completion the "RESEARCH COMPLETE" message is displayed.

```
[PROGRAM RESTART]                          MONITOR 5 .1/ 2
          O    2000   N    0 - 0
<SUB> O    3000   N    0 - 0
[RESTART- (G54)]   [RESTART-R]
X   -80.000        X   -80.000
Y   -40.000        Y   -40.000
Z     0.000        Z     0.000
C     0.000        C     0.000

O3000;
N1 G42 G01 X-10. Y-10. D05 F100;
M: MODE    <0> MEMORY      <1> TAPE
T: TYPE    <0> UNMODLE     <1> TYPE 1 <2> TYPE 2
 <3>T-TYP                  RESEARCH COMPLETE
O (          )   N (    ) - (   ) P (         ) T(   ) M(   )
```

b) Type B

The main program to be restarted is searched for.

Assign the block at which machining is to restart and proceed with the type 2 search.

O ( ) N (0) - (0)
P (3) T (2) M (0)

```
M: MODE    <0> MEMORY      <1> TAPE
T: TYPE    <0> UNMODLE     <1> TYPE 1   <2> TYPE 2
 <3>T-TYP
O (          )   N (   0) - ( 0) P (         3) T(2) M(0)
```

Press the INPUT key.

```
M: MODE    <0> MEMORY      <1> TAPE
T: TYPE    <0> UNMODLE     <1> TYPE 1   <2> TYPE 2
 <3>T-TYP                  RESEARCH EXECUTION
O (          )   N (   0) - ( 0) P (          ) T(2) M(0)
```

The "RESEARCH EXECUTION" message appears during the search and upon its completion the "RESEARCH COMPLETE" message is displayed.

```
[PROGRAM RESTART]                          MONITOR 5 .1/ 2
          O     2000    N    0 - 0
 <SUB> O     3000    N    0 - 0
 [RESTART- (G54)]    [RESTART-R]
 X   -80.000          X   -80.000
 Y   -40.000          Y   -40.000
 Z     0.000          Z     0.000
 C     0.000          C     0.000

 O3000;
 N1 G42 G01 X-10. Y-10. D05 F100;
 M: MODE    <0> MEMORY      <1> TAPE
 T: TYPE    <0> UNMODLE     <1> TYPE 1 <2> TYPE 2
  <3>T-TYP                  RESEARCH COMPLETE
 O (          )   N (    ) - (   ) P (          ) T(   ) M(   )
```

(3) Type 3 (T command restart) restart search

Set "3" in TYPE T ( ) in the setting area.

O ( ) N ( ) - ( )

P ( ) T (3) M ( )

```
M: MODE   <0> MEMORY     <1> TAPE
T: TYPE   <0> UNMODLE    <1> TYPE 1   <2> TYPE 2
 <3>T-TYP
O (        )  N (    ) - (   ) P (         ) T(3) M(0)
```

1) The data set in the setting areas other than T ( ) and M ( ) will be ignored.
2) Designate "1" in M ( ) when using the tape mode.

Press the INPUT key.

```
M: MODE   <0> MEMORY     <1> TAPE
T: TYPE   <0> UNMODLE    <1> TYPE 1   <2> TYPE 2
 <3>T-TYP                RESEARCH EXECUTION
O (        )  N (    ) - (   ) P (         ) T(3) M(0)
```

```
[PROGRAM RESTART]                          MONITOR 5 .1/ 2
          O     1000   N    5 - 0
 <SUB> O              N      -
 [RESTART- (G54)]   [RESTART-R]
 X   -130.000        X   -150.000
 Y   -10.000         Y   -150.000
 Z     0.000         Z     0.000
 C     0.000         C     0.000


 N51 Y-70.;
 N52 X-20.;
 M: MODE   <0> MEMORY     <1> TAPE
 T: TYPE   <0> UNMODLE    <1> TYPE 1 <2> TYPE 2
  <3>T-TYP                RESEARCH COMPLETE
 O (        )  N (    ) - (   ) P (         ) T(   ) M(   )
```

1) The block containing the T command executed last will be searched for.
2) The "RESEARCH EXECUTION" message appears during the search and upon its completion the "RESEARCH COMPLETE" message is displayed.
3) If each axis is at a (–) position from the value set in the restart limit (#2072 rslimt), the error "E98 CAN'T RESEARCH" will occur. Manually return the axis to a position where the error will not occur, and then search.

(Note) Restart type 3 is valid only with the lathe system.

2.5.3 Restart Position Return System

Selection can be made by parameter setting as to whether the restart position return after the restart search is to be performed either manually or automatically.

(1) Manual restart position return

Set the program restart switch to ON and mode the axes manually to the restart position.

- Set the restart switch to ON.
- Set operation to the manual (JOG/rapid traverse) mode.
- Move the axes in the restart return direction.

```
[RESTART- (G54)] [RESTART-R]
X  -130.000   RP  X     0.000
Y   -10.000   RP  Y     0.000
Z     0.000   RP  Z     0.000
C     0.000   RP  C     0.000
```

After the axes have finished returning to the restart position, the [RESTART-R] values on the program restart screen are set to zero and "RP" appears by the side of the [RESTART-P] values.

Set the restart switch to OFF.

(Note 1) When the restart switch is ON, the axes cannot be moved in the reverse direction to the restart direction. An operation error (0003) will occur if the axis is moved in the opposite direction. When a tool bumps into the workpiece and must be retracted, set the restart switch to OFF and retract the tool manually.

(Note 2) After returning to the restart position, the axes cannot be moved with the restart switch ON. An operation error (0111) will occur if the axis is moved.

(Note 3) If, during cycle start, there is even one axis which has not returned to the restart position, an operation error (0112) (there is an axis which has not returned to the restart position) will result. However, and operation error will not result with an axis which has been returned to the restart position but which is no longer at that position.

(Note 4) If the restart position return axis is a machine lock axis, an operation error (0126) will occur. Release the machine lock before starting the return to the restart position.

(2) Automatic restart position return

If the program restart automatic return (#1302 AutoRP) is set to 1 and the cycle is started, the axes will return to the restart position with dry run in the order designated with the restart position return order (#2082 a_rstax). Machining will restart after the axes have returned.

(Note 1) Proceed with cycle start after the axes have been moved by manual means to positions where the tools do not make contact with the workpieces.
If the axis has been moved with MDI, restart search will be invalidated.

(Note 2) Even if the "#1302 AutoRP" is set to 1, the axes can be returned manually to the restart position by turning the restart switch ON.
In this case, move the axes in the order of manual restart position return → automatic restart position return.
The restart operation is completed when the automatic restart position return is completed. Thus, after completing automatic restart position return, if the operation is stopped temporarily and the restart switch is turned ON, an operation error will occur.

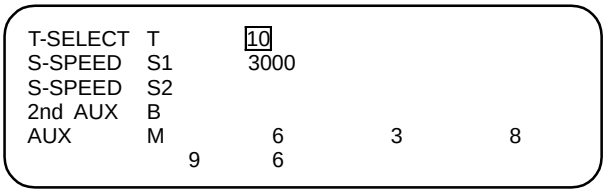
(Note 3) When any axis, which has already been returned manually to the restart position, is subsequently moved from the restart position, it will not return to that position even with the automatic restart position return.

(Note 4) The axis for which "#2082 a_rstax" is set to 0 will not return to the restart position. Note that if "#2082 a_rstax" is set to 0 for all axes, all axes will simultaneously return to the restart position. If the axis for which "#2082 a_rstax" is set to 0 has not completed manual restart position return when automatic restart position return is started, the error "T01 CAN'T CYCLE ST 112" (restart position return incomplete) will occur.

2.5.4 Manual Numeric Commands with Program Restart

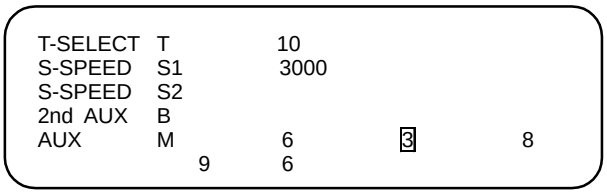
If restart search is completed, the M, S, T and B codes used for machining will appear on the second page of the PROGRAM RESTART screen. The M, S, T and B functions can be set in the time between the completion of the restart search and the resetting or start.
The maximum numbers of the codes that can be displayed are 35 for the M functions, 3 for the S functions, 3 for the T functions and 3 for the B functions. If these numbers of codes used for the machining are exceeded, the codes used first are not displayed. The codes that are not displayed cannot be commanded on this screen, and so manual numeric commands are executed on the POSITION screen.
The 2nd miscellaneous function code can be changed by setting the parameters.
Even if the miscellaneous function M is designated in the same block, these will appear in the commanded order.

Select page 2 of the PROGRAM RESTART screen.

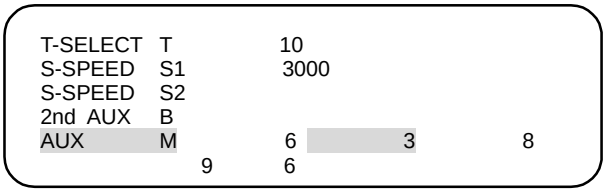


1) The M, S, T and B commands used for machining appear.
2) The cursor blinks at the right end of the data at the top left of the screen.

Using the ↑ ↓ ← → keys, move the cursor to the position of the data to be set.



Press the INPUT key.



1) The command value with the cursor and the command name display now change to a highlighted display.
2) The highlighting remains and the cursor does not appear during the time until the command has been executed.
3) Upon completion of the command execution, the normal display is resumed and the cursor appears at the next command value position.
4) The assigned codes are displayed on the POSITION screen.

2.5.5 Checkpoints for Program Restart

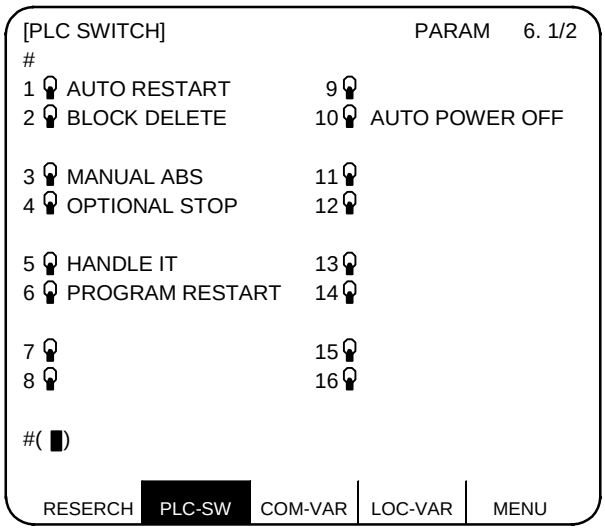
(1) Set the tool offset amounts and parameters before proceeding with the program restart search. If the conditions for such are not set beforehand, it will not be possible for the axes to return to the proper machining start position.
(2) Do not conduct automatic operations during program restart.
It will not be possible for the axes to return to the proper machining start position if either operation is conducted during program restart. ("During program restart" means the period from the program restart search to the start of the searched program.)
(3) It will not be possible for the axes to return to the proper machining start position if a program using user macro external signal input, machine coordinate readout or external mirror image commands is the object of the restart search.
(4) When an attempt is made to shift the coordinate systems by manual or MDI interrupt while the previous machining program is being executed, it will not be possible for the axes to return to the proper machining start position.
(5) If type 1 is used, there is no need to implement the unmodal search for the head of the program. The operation start block of the previously operated program is stored inside and the type 1 restart search commences the search (modal search) from the previous operation start block and it locates the designated block. Consequently, unmodal searches are invalid even if they are conducted.
(6) When type 2 is used with type A (standard specifications), the message "E80 TOP SEARCH ERR" will appear if an unmodal search is not executed and search is directly attempted with type 2. Conduct the unmodal search first.
(7) If either type 1 or type 2 is used, it is not possible to search blocks with macro statements during the restart search operation. (This will result in the "E13 NB NOT FOUND" error.)
To initiate a restart search for blocks with macro statements, first set the "#2 MACRO SINGLE" control parameter ON and then proceed. However, the tool path may change because of the relationship of the radius compensation, corner rounding/chamfering and geometric read ahead blocks.
(8) When conducting restart with a tape operation, it will not be possible to return to the proper machining start position if operation is started midway through the tape.
(9) WHILE/GOTO statements cannot be used during tape operations. This means that if such statements exist even when restart search is performed by tape, a program error (P29) results.
(10) The program number cannot be omitted when conducting a search with type 1 or 2 for type A (standard specification). If this number setting is omitted, the "E01 SETTING ERROR" message appears and restart search is not executed.
Reset the program number and proceed again with operation.
(11) If a type 2 search is executed when using type B, the designated sequence No. and block No. will be searched for in all programs within the designated program numbers. Note that if the same sequence No. is found in the main program and subprograms, the No. of appearances designation P will be the number of times for the entire program.
(12) Select type A and type B with the setup parameter "#1278 ext14/bit0".
0: Type A
1: Type B
(13) When using multiple systems, carry out restart search for each system.
(14) If the axis returning to the restart position is a linear type rotary axis, the axis will return to the workpiece coordinate position.
(15) When using type 3 (T command restart), the N number must be issued before the first T command block in the program to be searched.
(16) When using type 3 (T command restart), there must be one or more N number within 100 blocks of the program.
(17) When using type 3 (T command restart), the only T commands that can be searched are the main program and the subprogram (nesting 1) called from the main program.
(18) When using type 3 (T command restart), if the program is edited before restart search and the position of the T command block executed last differs from the execution, the error "E98 CAN'T RESEARCH" will occur when the search is started.

2.6 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.



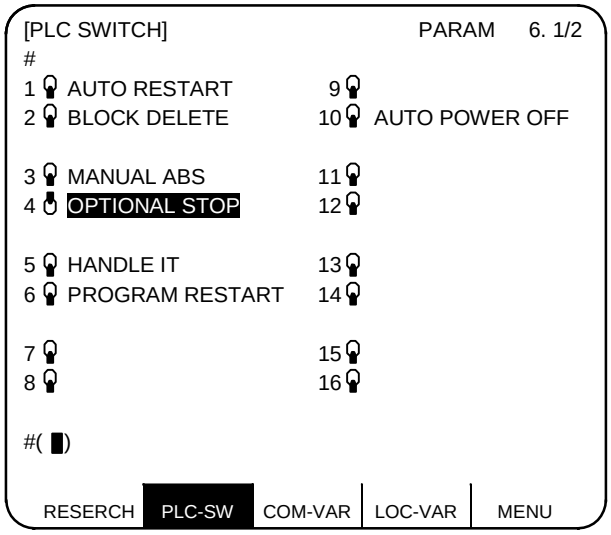
2.6.1 PLC Switch ON and OFF Operation

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.



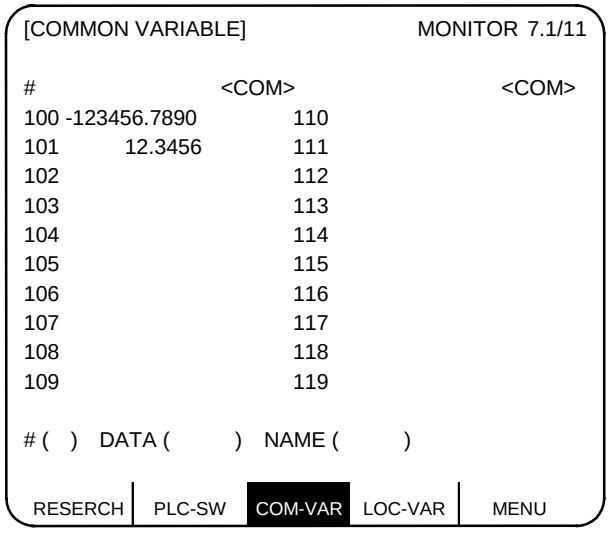
The switch mark of OPTIONAL STOP is set to the up position, indicating the switch ON state.

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 maker.

2.7 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).



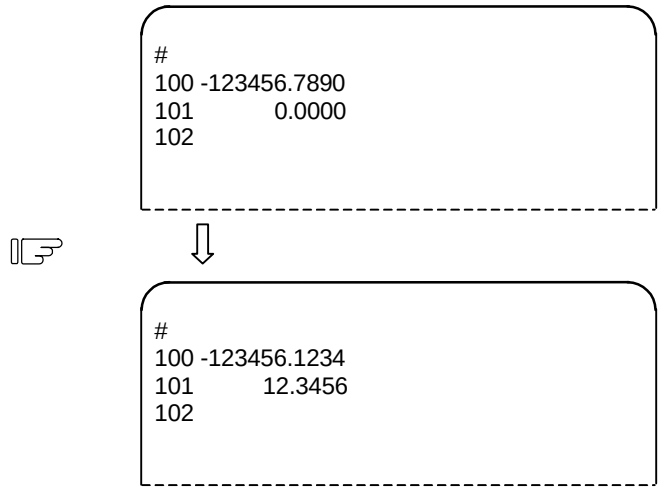
Display item	Explanation
# 100 -123456.7890 101 12.3456 102	The variable numbers and contents are displayed. If variable data is "null" **(Note)**, the data display field will be blank. If the number of columns of data is too large (the data contains more than six characters in the integer part or more than four characters in the fraction part), the exponent will be used for display. **(Note)** In terms of calculation, a "null" setting is handled in the same manner as "0". However, it is not handled in the same manner as "0" when using the condition expressions EQ and NE.
<COM>	"*" display: Indicates that this is a common variable common for all systems within a multiple system. (The number of common variables common for all systems is set in #1303 VlcomN and #1304 V0comN.)
	" " display: Indicates that this is a common variable independent for the systems in a multiple system. When there is only one system, " " will display regardless of the parameter (#1303, #1304) setting.

2.7.1 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.

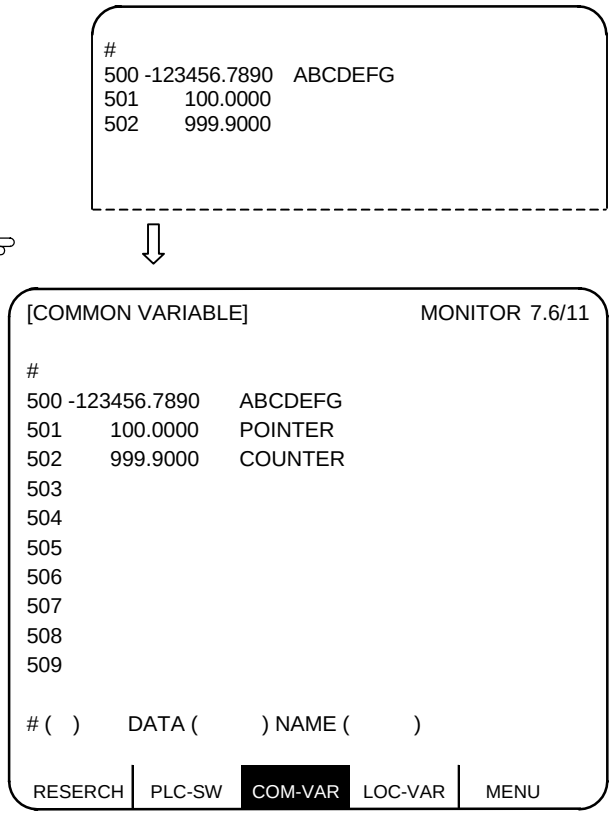
```
(
#101=12.3456
(
```



(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];
(
```



2.7.2 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.

2.7.3 Common Variable Data Deleting

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

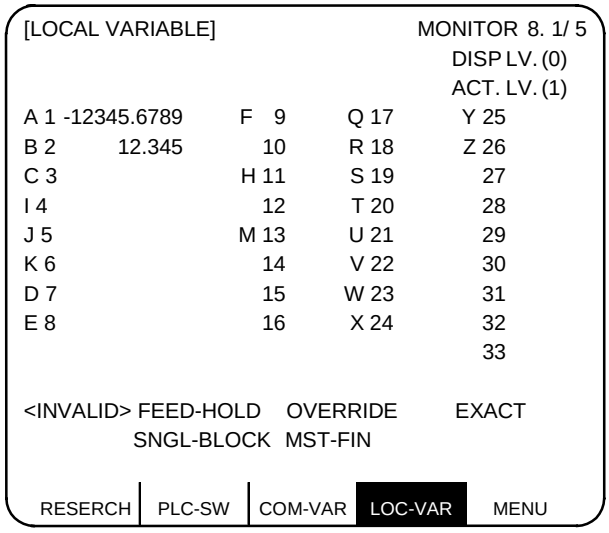
(Note1) If parameter #1128 RstVC1 is ON, the common variable data will be cleared to "null" when the system is reset. If parameter #1129 PwrVC1 is ON, the common variable data will be cleared to "null" when the power is turned ON.

(Note2) If any other key has been pressed before the [INPUT] key is pressed, the variable data will not be deleted.

2.8 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" **(Note)**, 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. **(Note)** In terms of calculation, a "null" setting is handled in the same manner as "0". However, it is not handled in the same manner as "0" when using the condition expressions EQ and NE.
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

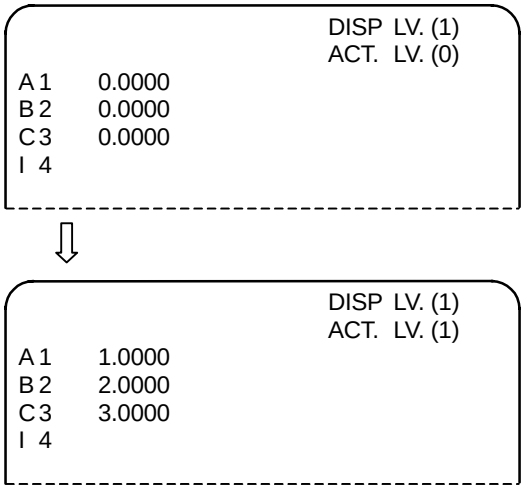
Display item	Explanation
<INVALID> FEED-HOLD OVERRIDE EXACT SNGL-BLOCK MST-FIN	This indicates the modal state of the operation control status by the #3003, #3004 command. FEED-HOLD : Is displayed when command is programmed with #3004 bit 0 set to 1, indicating that feed hold is invalid. OVERRIDE : Is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

2.8.1 Local Variable Data Display

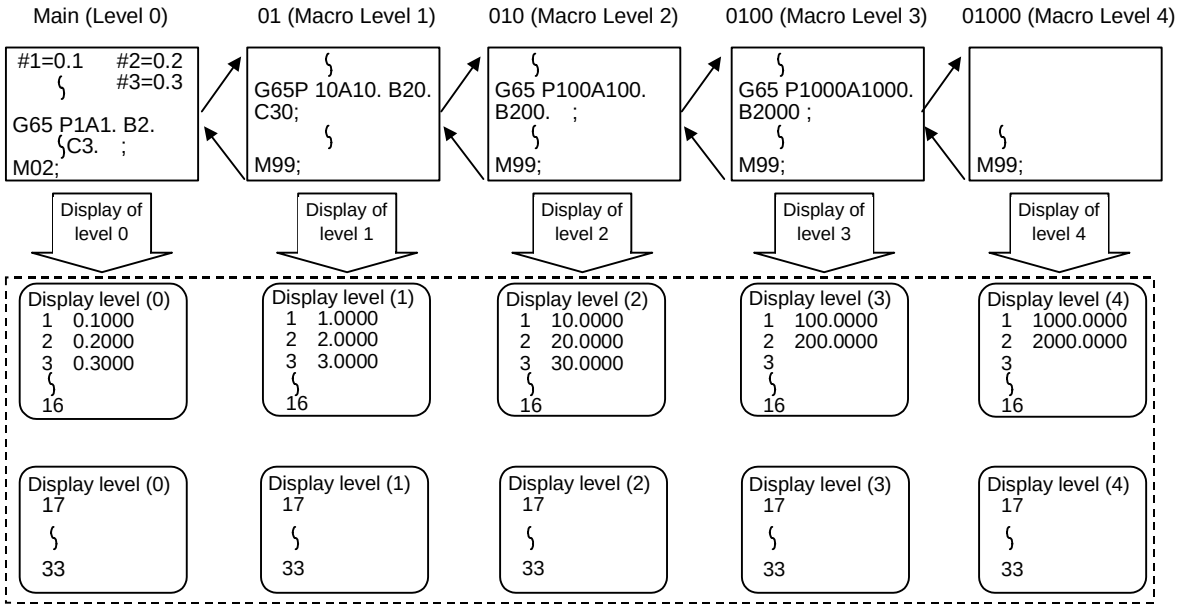
(1) When local variable #1~#33 command exists in user macro or argument specification is made in user macro subprogram call, if the block is executed, the execution result is displayed.

(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:

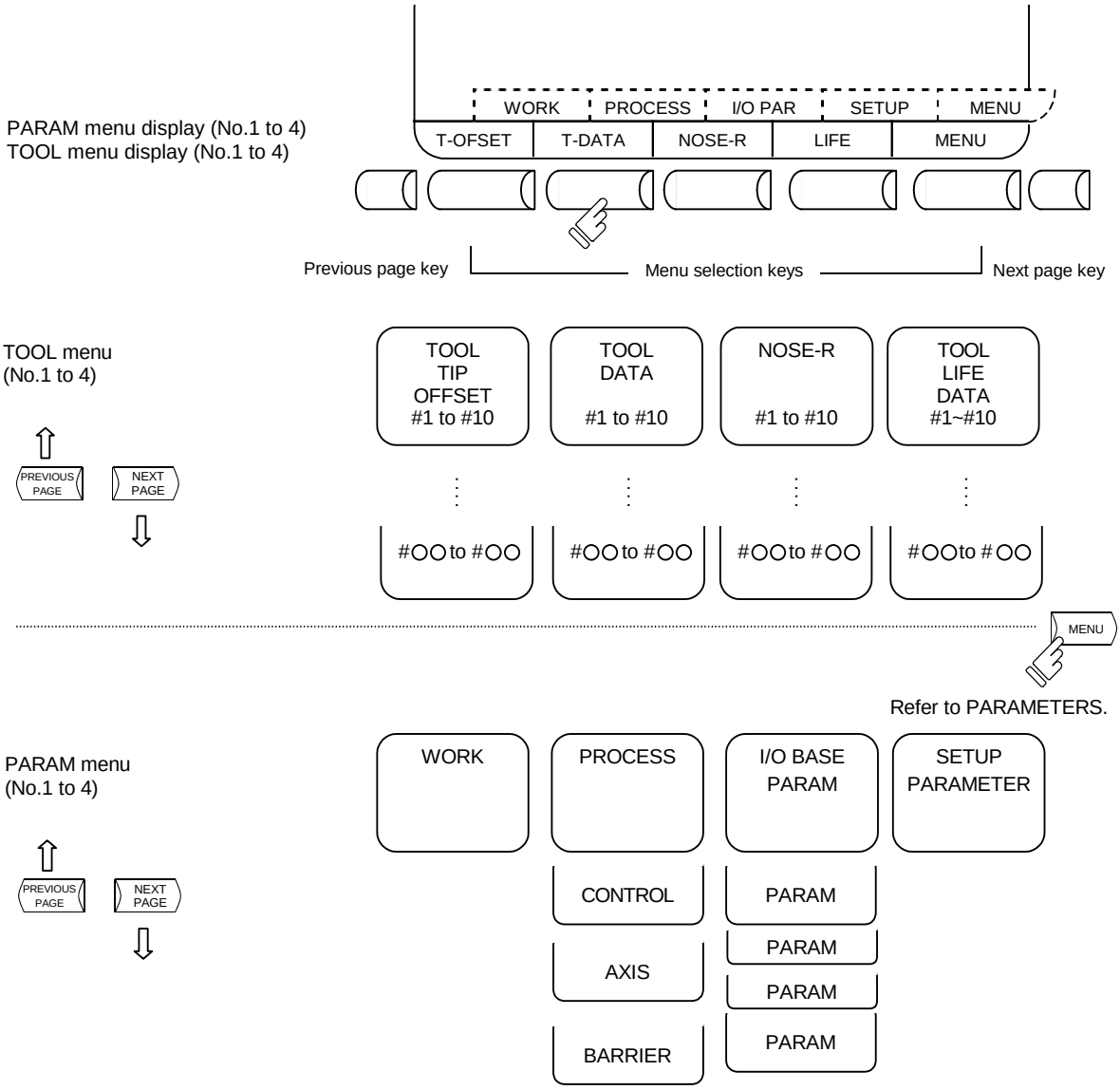


(3) A local variable display page is selected by using the page keys PREVIOUS PAGE, NEXT PAGE. Display can be changed as desired independently of the executing level.

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

3 (I). Tool Offset (L system)

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

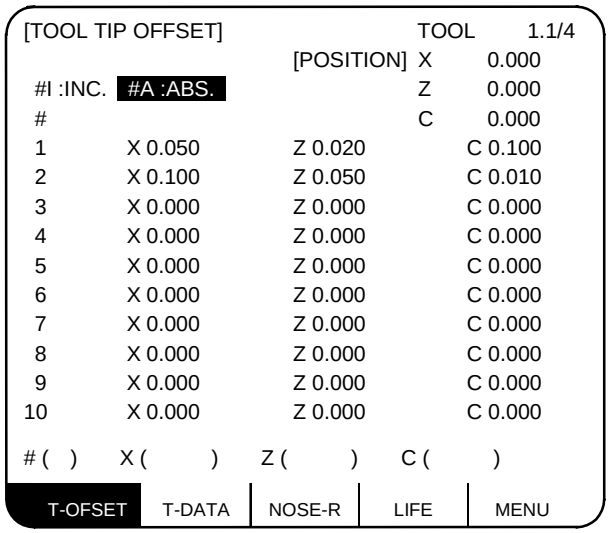


⚠ CAUTION

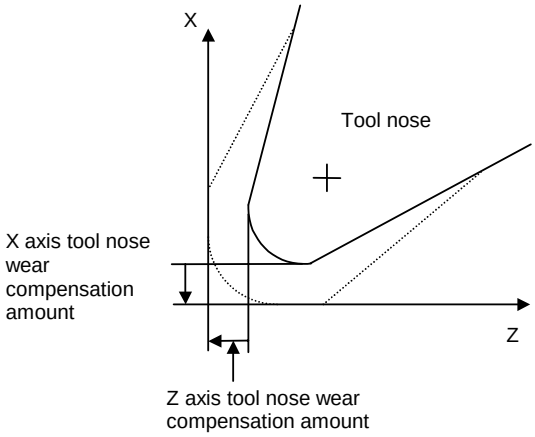
⚠ If a tool offset or workpiece coordinate system offset is changed during automatic operation (including during single block stop), the new offset is validated from the command of the next block or blocks onwards.

3.1 Wear Data

The TOOL TIP OFFSET screen will appear when the menu key ⟩T-OFSET⟨ is pressed.



Set the tool 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

(Note 1) Whether to apply the tool nose wear compensation of the additional axis on the 3rd axis or 4th axis can be selected with the parameter (#1520 Tchg34).

(Note 2) For multiple system
Tool data can be provided for each system, or common tool data can be used for the systems.
Select with parameter (#1501 MemTol).
Parameter #1501 MemTol 0: Tool data for each system
1: Tool data common for all systems
When common tool data is used for the systems, the tool data on the System 1 screen and System 2 screen will have the same values.

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] key 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	#1003 iunit	Setting range (unit)
TOOL TIP OFFSET	X, Z, C	Tool wear	B	±9999.999 (mm)
			C	±999.9999 (mm)
			D	±99.99999 (mm)
TOOL DATA	X, Z, C	Tool length offset	B	±9999.999 (mm)
			C	±999.9999 (mm)
			D	±99.99999 (mm)
NOSE-R	R	Tool radius (nose R)	B	±9999.999 (mm)
			C	±999.9999 (mm)
			D	±99.99999 (mm)
	r	Tool radius (nose R) wear	B	±999.999 (mm)
			C	±99.9999 (mm)
			D	±9.99999 (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.

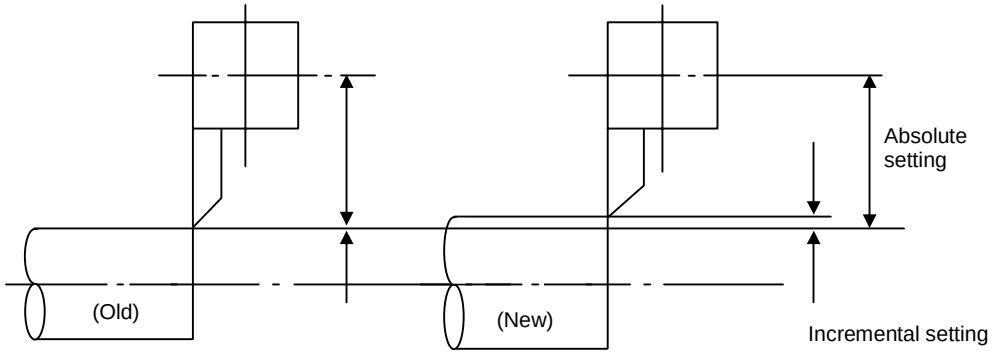
3.1.2 Erasing the Tool Offset Data

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

(Note) If any other key has been pressed before the [INPUT] key is pressed, the offset data will not be erased.

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

There are two types of selection method in the absolute value setting or incremental value setting for the tool offset data: the mode selection method and the menu selection method.
The required method is selected with the parameter #1136 optype.



(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) Mode selection method (#1136=0)

(a) Change over to the incremental value setting mode

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

☞

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

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

(b) 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.**

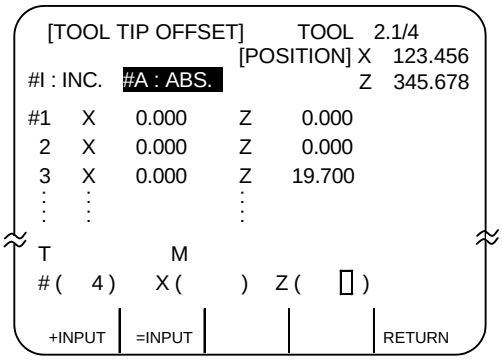
"#A: ABS." is highlighted, and the mode becomes the absolute value setting mode.

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

(2) Menu selection method (#1136=1)

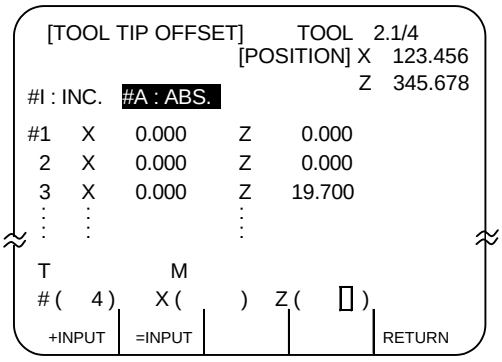
(a) Change over to the incremental value setting mode

Set value in #( ), X( ) or Z( ), then press the + INPUT key.



(b) Change over to the absolute value setting mode

Set value in #( ), X( ) or Z( ), then press the = INPUT key.



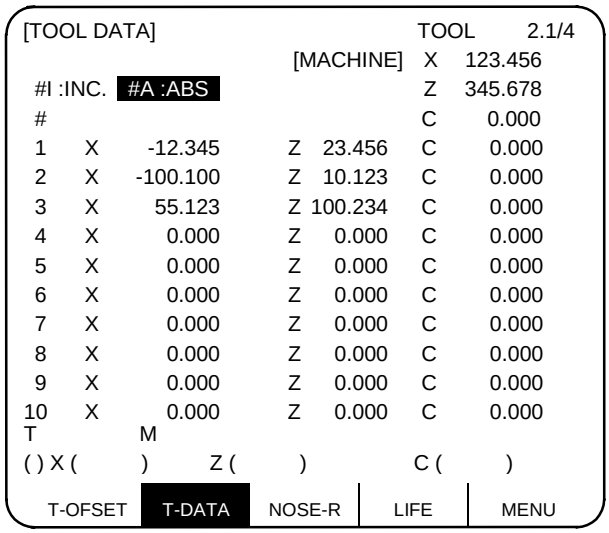
* When the cursor is moved to X( ) or Z( ) then the data key is pressed, =INPUT,+INPUT menu will appear automatically.

(c) Supplement

- If INPUT key is pressed while = INPUT, + INPUT are displayed, the offset data can be set for mode selection method.
- = INPUT, + INPUT are displayed until BACK menu key is pressed or other screens are selected. The screen selection menu is displayed when TOOL OFFSET screen is selected again after other screens are selected once.
- "↓ " Mark of the screen selection menu indicates that the operation menu exist on the screen being displayed.
- The operation menu is invalid during the tool measurement with manual tool length measurement function. The operation menu during the tool measurement operates as follows.
  1) When the screen select menu is displayed
  + INPUT, = INPUT don't appear during the tool measurement. If = INPUT, + INPUT with "↓ " mark are pressed during the tool measurement, ERROR (E74) will occur.
  2) When the operation menu is displayed
  If = INPUT, + INPUT are pressed during the tool measurement, ERROR (E74) will occur.
- = INPUT, + INPUT are invalid during the manual numerical command mode. If = INPUT, + INPUT are pressed during the manual numerical command mode, ERROR (E74) will occur. If the screen selection menu displayed with "↓ " mark is pressed during the manual numerical command mode, ERROR (E74) will occur.
- When the offset memory No. not displayed on the screen is set in setting area # ( ), the screen corresponding to the offset memory No. set in will appear by pressing = INPUT,+ INPUT key of one time. The offset data will be set by pressing = INPUT, + INPUT key again.
- When the tool nose point data is set on the NOSE-R screen, the absolute setting is selected whichever of = INPUT and + INPUT key is pressed.

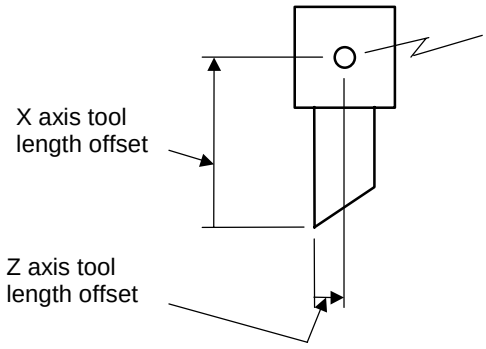
3.2 Tool Length Data

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



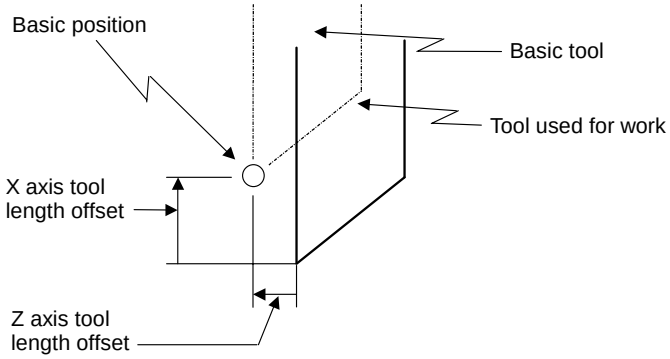
Set the tool length in respect to the program basic position of each tool used.
When the tool compensation No. is designated by the tool command (T command), compensation is carried out matching the wear data of the previous screen. Generally, the program basic point position is either the turret center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

(1) Turret center position



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

(2) Basic tool nose position



(Note) Whether to apply the tool length offset of the additional axis on the 3rd axis or 4th axis can be selected with the parameter (#1520 Tchg34).

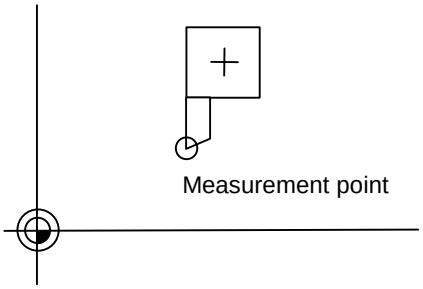
3.2.1 Manual Tool Length Measurement I

(1) Outline

This function automatically calculates the amount of tool length offset, 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 basic point method and the measurement value input method. The required method is selected by setting parameter #1102 tlm.

(a) Basic point method

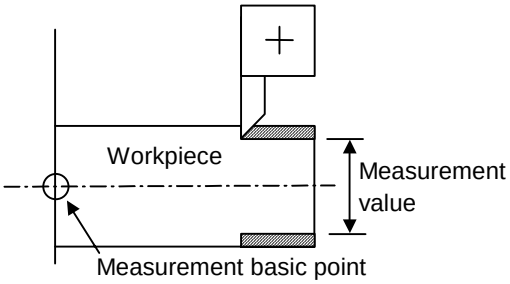
Obtain the tool length with the tool nose placed on the measurement point.



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

(b) Measurement value input method

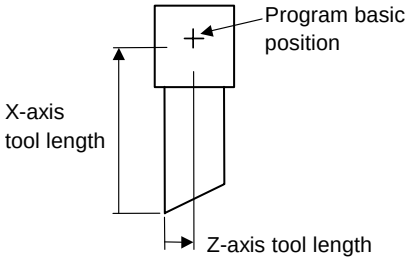
Actually cut the workpiece. Measure its dimensions, and obtain the tool length from the measured values.



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

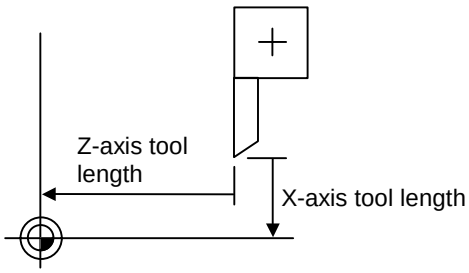
(Note) The tool length from tool length measurement I is as follows, depending on the whether the 1st reference point coordinate values have been set.

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 basic position.

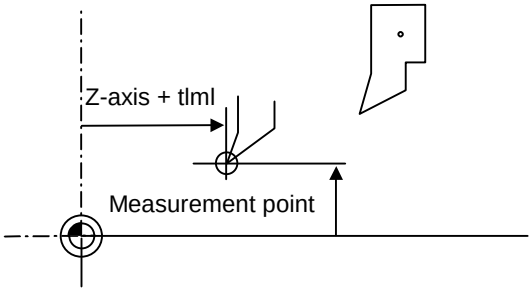
If the 1st reference point coordinate values have not been set:



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

(2) Basic point method

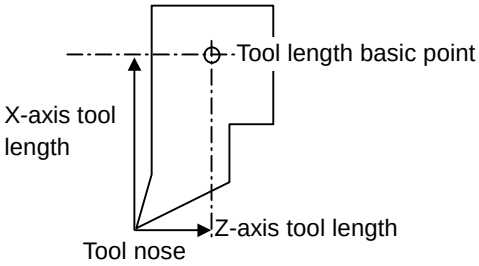
Set the type selection to the basic point method. (Set #1102 tlm to 0).
To carry out the basic point method, a point to place the tool nose on (measurement point) is required.
Set the measurement point in parameter #2015 tlml– beforehand.



(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)

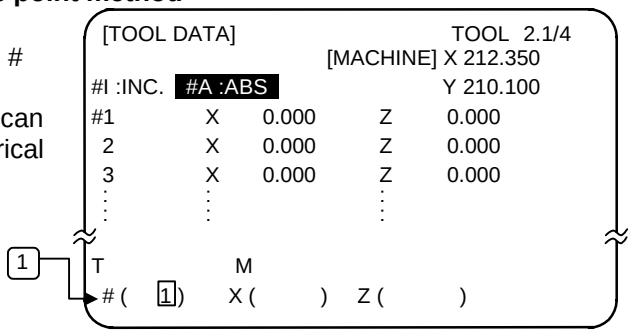
The expression above is used for automatic calculation in the basic point method. When the tool nose is placed on the measuring point, the distance from the tool nose to the tool length basic point is calculated.



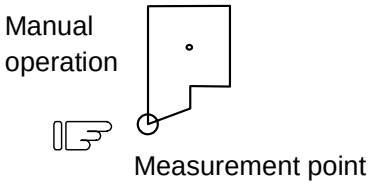
< Measuring procedure for the basic point method >

(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".



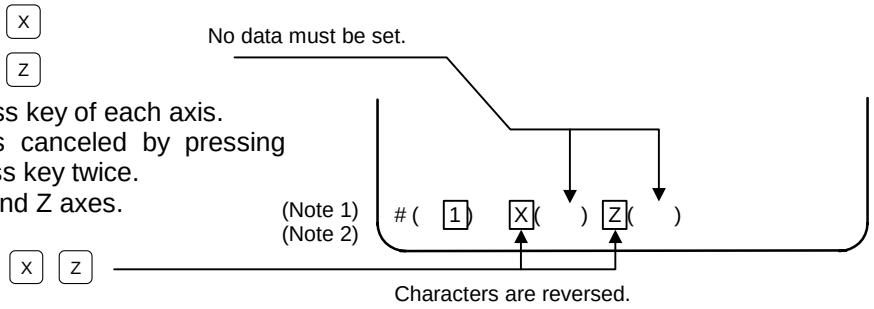
(3) Manually place the tool nose on the measuring point.



(4) Select the axis to be measured.

X axis — X
Z axis — Z

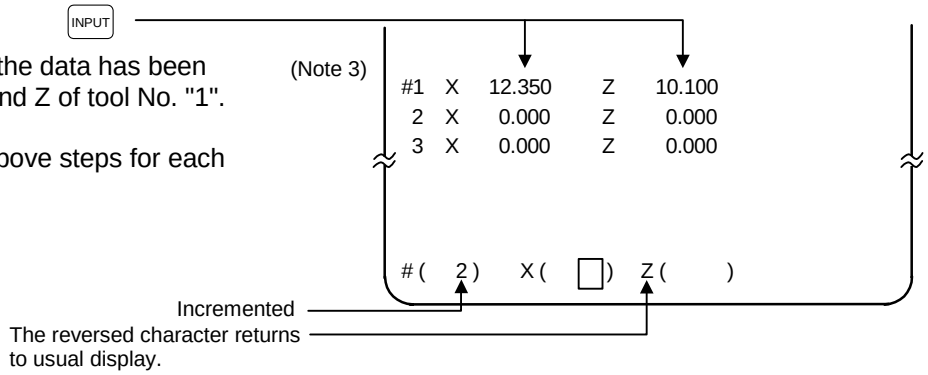
Press the address key of each axis.
The selection is canceled by pressing the same address key twice.
Measure the X and Z axes.



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

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

Repeat the above steps for each tool.



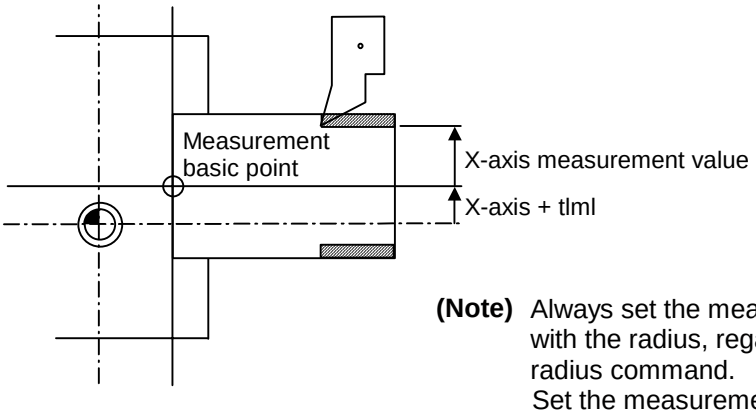
(Note 1) If the screen is changed back to the TOOL DATA screen after axis selection (after the characters are highlighted), the selection is invalidated (the characters are not highlighted).

(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.

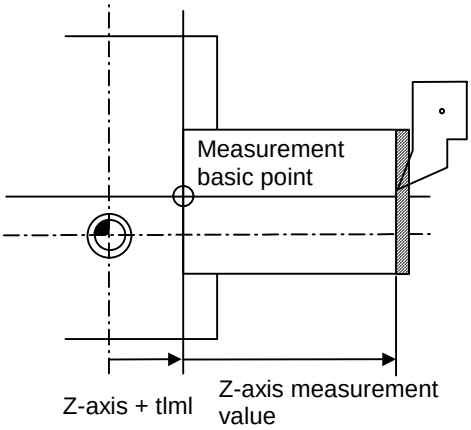
(Note 3) For a diameter command, the diameter value is written.
For a radius command, the radius value is written.

(3) Measurement value input method

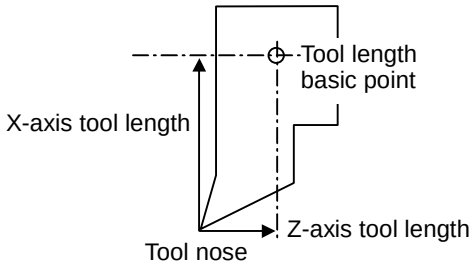
Set the type selection to the measurement value input method. (Set #1102 tlm to 1).
To carry out the measurement value input method, a workpiece for measuring is required. To measure the workpiece, set the basic point in parameter #2015 tlml– beforehand.



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

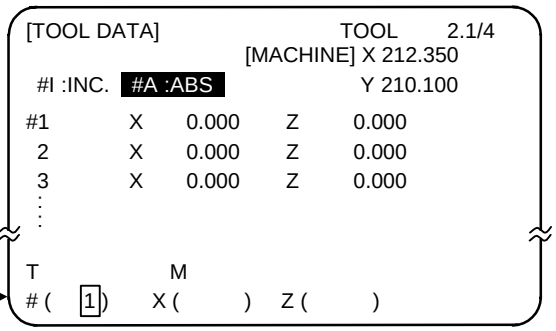


Tool length = Machine value - Measurement basic point (tlml) - Measurement value
The expression above is used for automatic calculation in the measurement value input method.



< 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.)

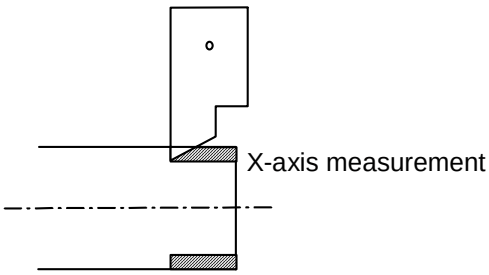


(Example) Select tool length No."1".

(3) Cut the surface corresponding to the axis to be measured.
To measure the X axis, cut the workpiece in the longitudinal direction.

(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.

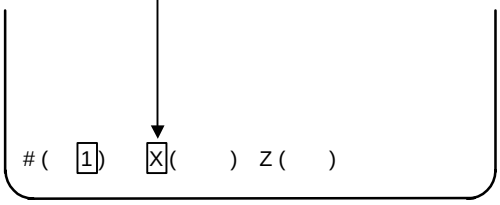


X axis measurement... [X]

Character is reversed.

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.

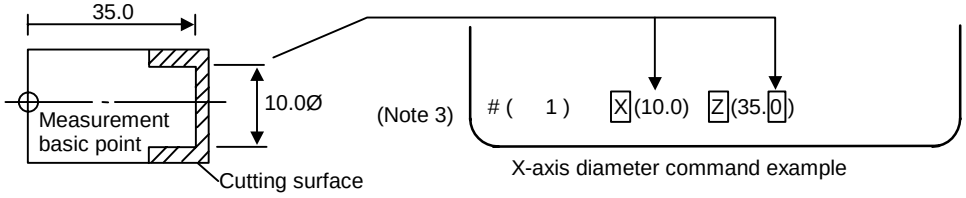
(Note 1)
(Note 2)



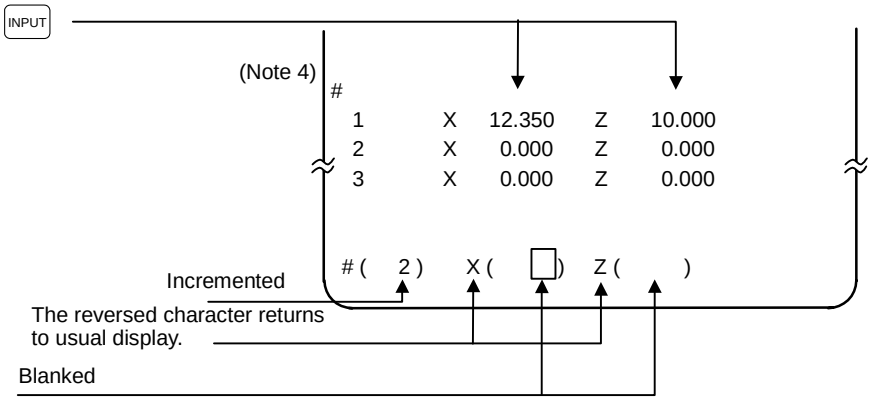
* Also repeat steps (3) and (4) for the Z axis

(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:

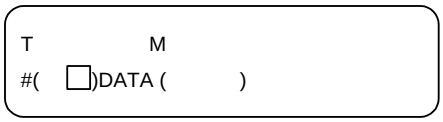
- # (1) [X] ( ) [Z] (35.0) ... The X axis measurement value was not set.
- # (1) X (10.0) [Z] (35.0) ... The character was not highlighted although the X axis measurement value was set.

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

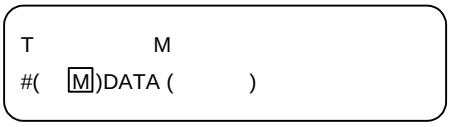
3.2.2 Manual Numeric Command Operation on the TOOL DATA Screen (M, T)

When carrying out a manual numeric command of the TOOL OFFSET screen, the mode must first be changed from the normal data setting mode to the manual numeric 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:

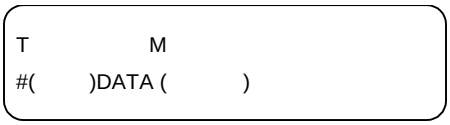


Set [M] (manual) in the first set of parentheses in the setting area.



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

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



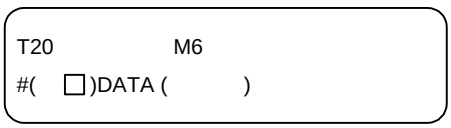
1) The data in the setting area is cleared, and the cursor disappears from the screen.

(2) Executing the manual numeric 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 numeric 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 numeric command operation is the same as the operation on the POSITION screen. Refer to the section on manual numeric commands for the MONITOR and POSITION screens for details.

(3) Operation for returning the mode from the manual numeric 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.

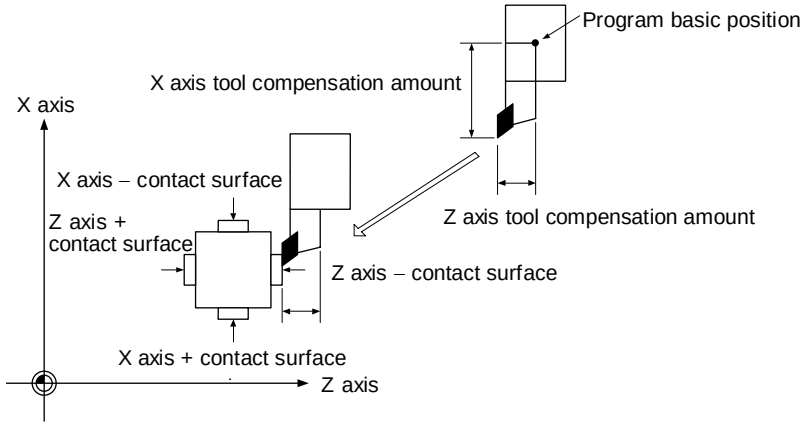
3.2.3 Manual Tool Length Measurement II

(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) Detailed explanation

(a) Tool compensation amount measurement



1) Measurement method

1. Set the machine coordinate values of the touch sensor's contact surface in the parameters beforehand as the measurement basic value.
2. Select the tool for which the tool compensation amount is to be measured.
3. Using manual feed, contact the nose of the tool against the touch sensor.
   The tool compensation amount will be calculated from the machine coordinate value when the touch sensor is contacted and the measurement basic value, and will be saved in the memory as the tool compensation amount.

Tool compensation amount = Machine coordinate value – measurement basic value (sensor position)

After measuring, the tool wear amounts for the individually designated tool numbers are cleared.

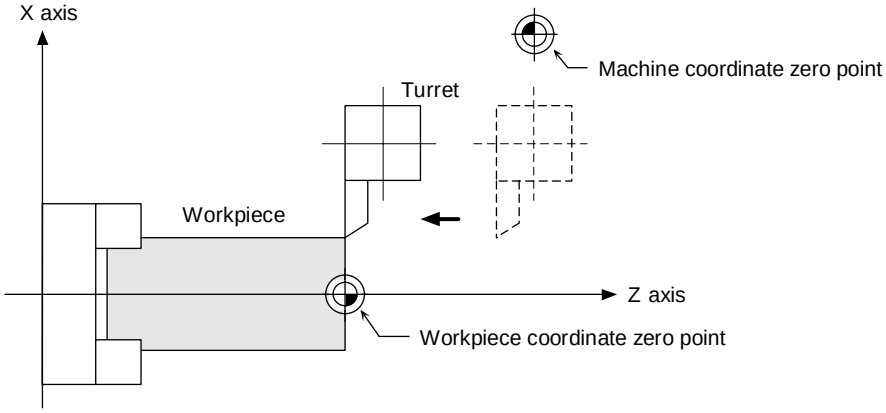
2) Number of set systems and axes

The system 1 X (1st axis), Z (2nd axis), additional axis, and system 2 X (1st axis), Z (2nd axis), additional axis can be set. The additional axis is determined with the #1520 Tchg34 additional axis tool compensation operation selection parameter.

#1520 Tchg34	Additional axis
0	3rd axis selection
1	4th axis selection

Note that the tools in the two systems cannot be measured simultaneously.

(b) Z axis workpiece coordinate offset data measurement



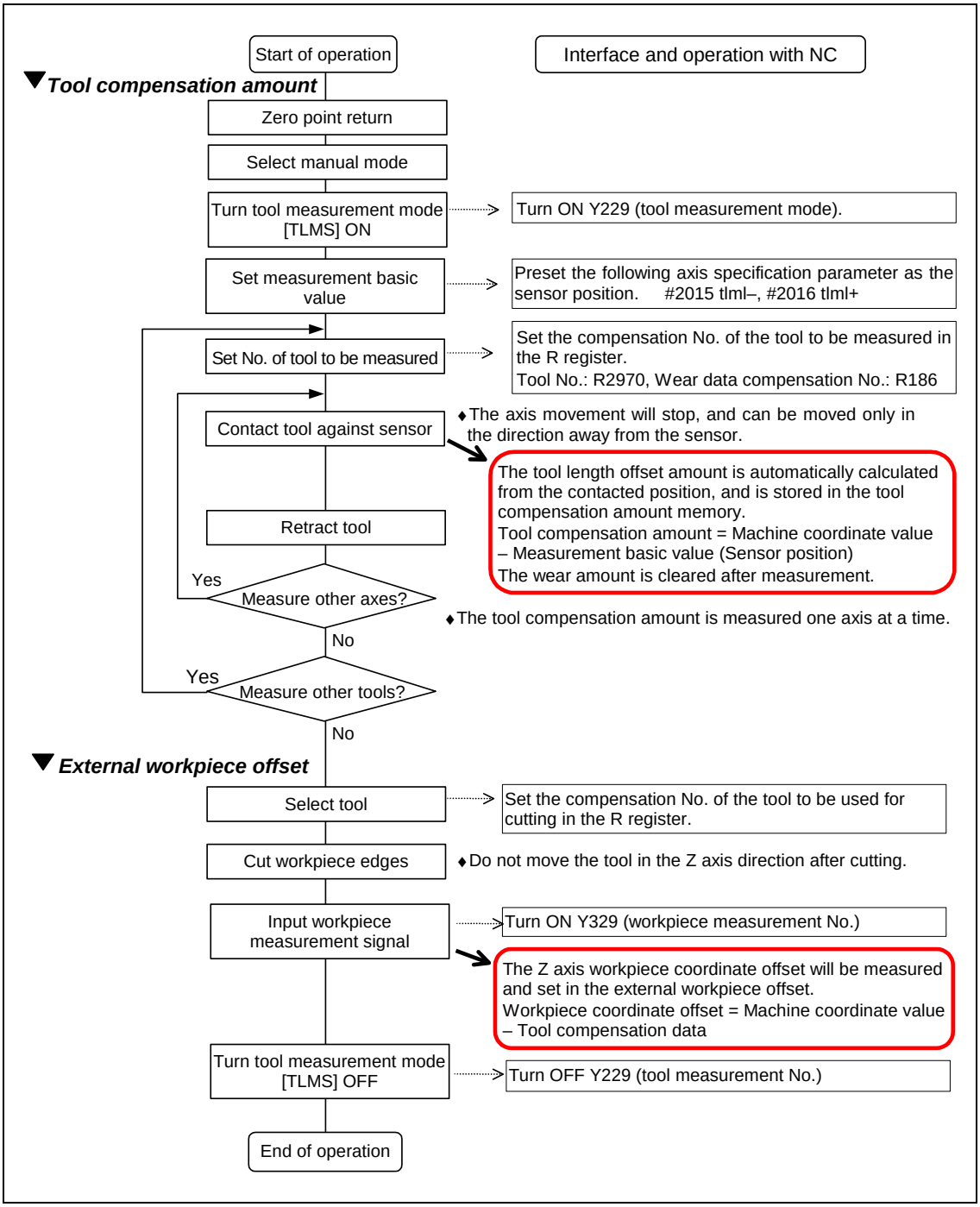
1) Setting method

1. Select the tool and cut the workpiece edge.
2. When the workpiece measurement signal is input, the Z axis external workpiece coordinate offset data will be calculated from the machine coordinate value, the length of the tool used and the tool nose wear compensation amount. This value will be saved in the memory.

2) No. of set systems and axes

The external workpiece coordinate offset for the system 1 Z axis (2nd axis) and system 2 Z axis (2nd axis) can be set.
Note that the workpiece coordinate values for the two systems cannot be measured simultaneously.

(3) Operation flow



(4) Explanation of operations

(a) Setting the tool compensation amount

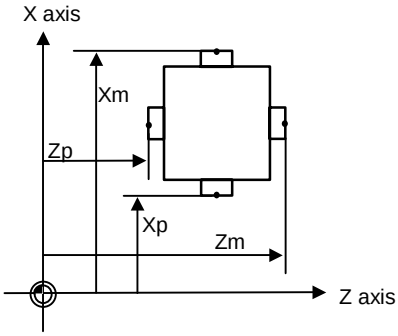
1) Zero point return
After turning the power ON, establish the coordinate system by carrying out dog-type zero point return. When using the absolute position detection specifications, carry out initialization if the absolute position is not established.

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 basic value (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

If the axis to be measured is the additional axis, the axis set with #2015 tlml-/#2016 tlml+ will differ according to the additional axis tool compensation operation selection parameter (#1520 Tchg34).

#1520 Tchg34	#2015 tlml–/#2016 tlml+ setting
0	3rd axis
1	4th axis

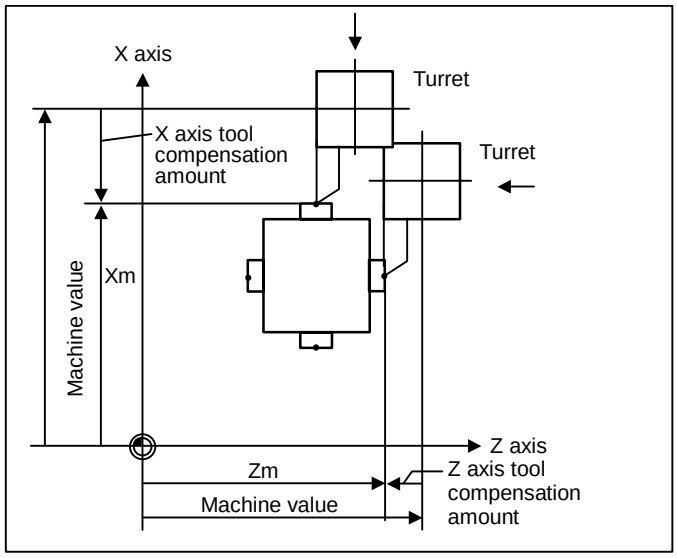
5) Select the tool
Select the tool to be measured.
Set the compensation No. of the tool to be selected as a BCD code in R2970.
Set the compensation No. of the wear data to be cleared after measurement as a BCD code in R186.
(The tool No. data is input from the PLC to the NC.)

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 tool nose contacts the sensor.
The tool length offset 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. The direction of the axis movement when the sensor contacts the tool will be output to R90 (R290).

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

Tool compensation amount = Machine coordinate value – Measurement basic value



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) 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) 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.

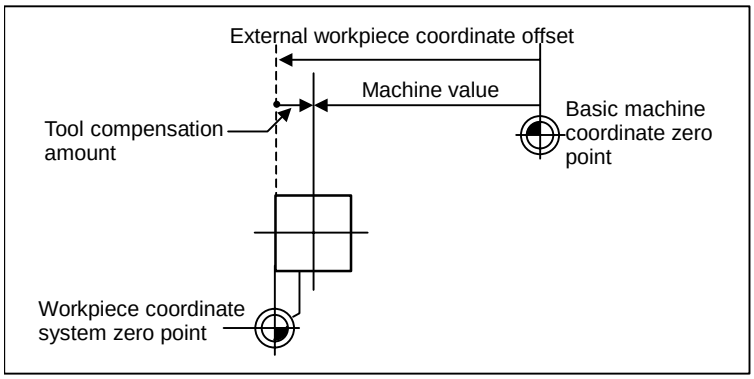
6) Set the Z axis external workpiece offset data with the workpiece measurement signal input
Turn ON the workpiece measurement signal. The Z axis external workpiece coordinate offset data will be automatically calculated from the machine value at the time the signal is turned ON and the tool compensation 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 + tool 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 tool nose wear data compensation numbers for automatic calculation.

Compensation No. R registers

#1098 Tlno.	#1130 set_t	#1218 aux02 bit4	Tool length compensation No.	Tool nose wear compensation No.
0	0	0/1	R192, R193	R192, R193
	1	0/1		
1	0	0	R36, R37	R192, R193
		1	R194, R195	R192, R193
	1	0/1	R194, R195	R192, R193

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	Specify the No. of digits in the tool length offset No. in the T command. 0: The 2 or 3 high-order digits are the tool No. The 2 or 1 low-order digits are the tool length offset and wear compensation Nos. 1: The 2 or 3 high-order digits are the tool No. and tool length offset Nos. The 2 or 1 low-order digits are the wear compensation No.
1130	set_t	Display selected tool number	Specify the tool command value display on the POSITION screen. 0: T-modal value of program command is displayed. 1: Tool number sent from PLC is displayed.
1218	aux02 (bit4)	Tool number selection	Specify the R register that contains the tool number used for automatic calculation when measuring the coordinate offset of an external work piece. 0: Conforms to #1130 set_t. 1: Uses the tool number indicated by user PLC

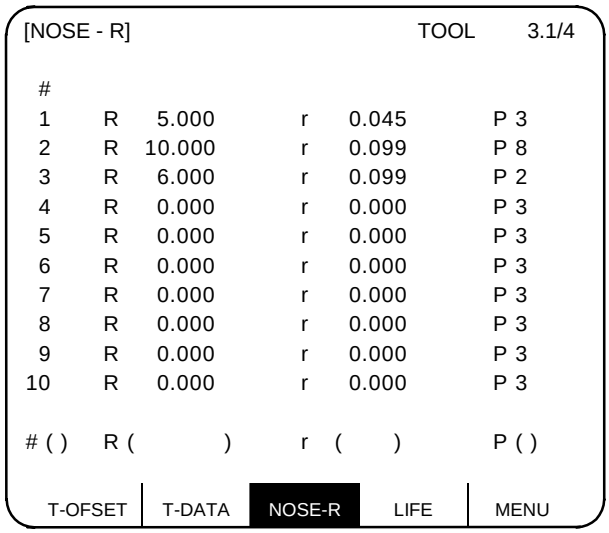
7) Turn the tool measurement mode signal OFF.
This completes the measurement of the external workpiece coordinate offset.
When carrying out this operation independently, follow steps 1) to 7), and when carrying out after measuring the tool compensation amount, carry out steps 4) to 6) between 9) and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

(5) Precautions

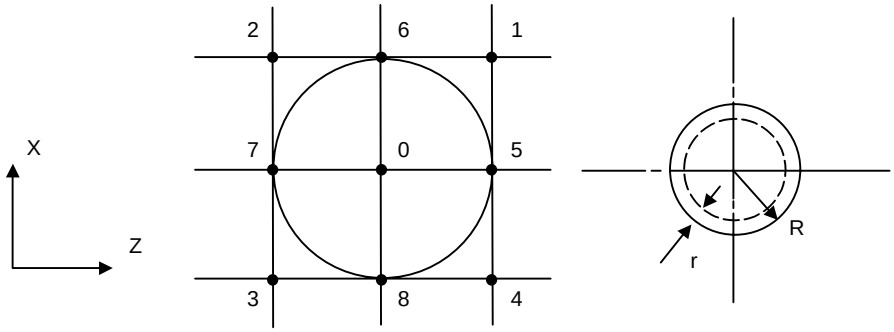
1) When entering the sensor area, the axis can move only in one direction selected from +X, –X, +Z, –Z, (+Y, –Y).
If two axes (ex. +X, –Z) are moved simultaneously, it will not be clear which contact surface was contacted, so the measurement will not be made. Note that the error "E78 AX UNMATCH (TLM)" will occur and the movement will stop for safety purposes.
2) After entering the sensor area, if the tool 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 tool nose is separated from the sensor.
The conditions for the axis to move in both directions are as follow:
1. The sensor signal has been OFF for more than 500ms
2. The axis has moved 100µm or more after the sensor signal has turned OFF.
1 and 2 can be selected with the parameter #1227 aux11/bit 2 tool setter chattering measures.
The interlock direction during interlock is output to R91 (R291).

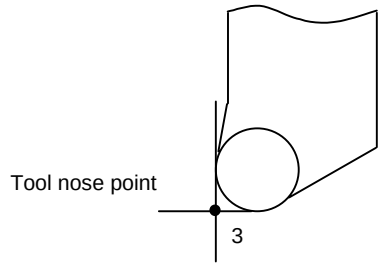
3.3 Tool Nose Data

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.



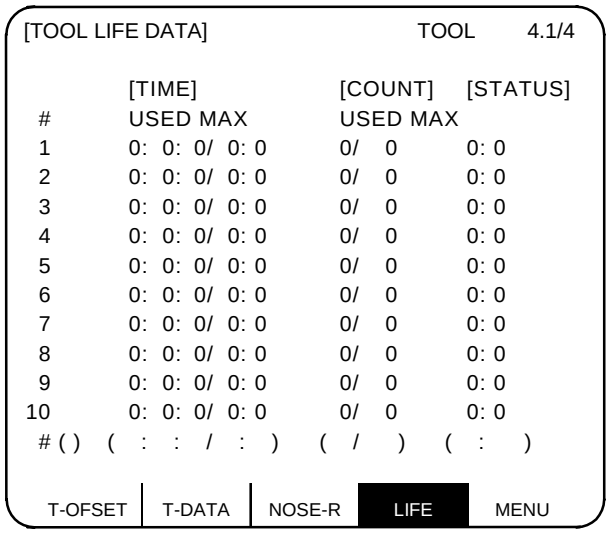


Data	Function
P0 to P8	Tool nose point
R	Tool radius (nose R) (no sign)
r	Wear (no sign)

(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.

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 (X20E) 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

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 reaches the service lifetime, or when the count exceeds the service life count, the corresponding tool No. (#) on the TOOL LIFE DATA screen is highlighted.

(4) No management

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

3.4.2 Conditions for Counting (incrementing)

The usage time (or count) is incremented when a cutting feed (G1, G2, G3, G33) is executed. Note that they are not incremented in the following conditions:

- When the base specification 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)

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.

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 [SHIFT] key, and then pressing the [C.B CAN] key and [INPUT] key.

(Note) If any other key has been pressed before the [INPUT] key is pressed, the tool life management data will not be erased.

3.4.5 Precautions

(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: X20E) is not output.
(2) For tool life management by time, a warning will not be output to the user 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.

3.5 Tool Life Management II (#1096 T_Ltype is 2)

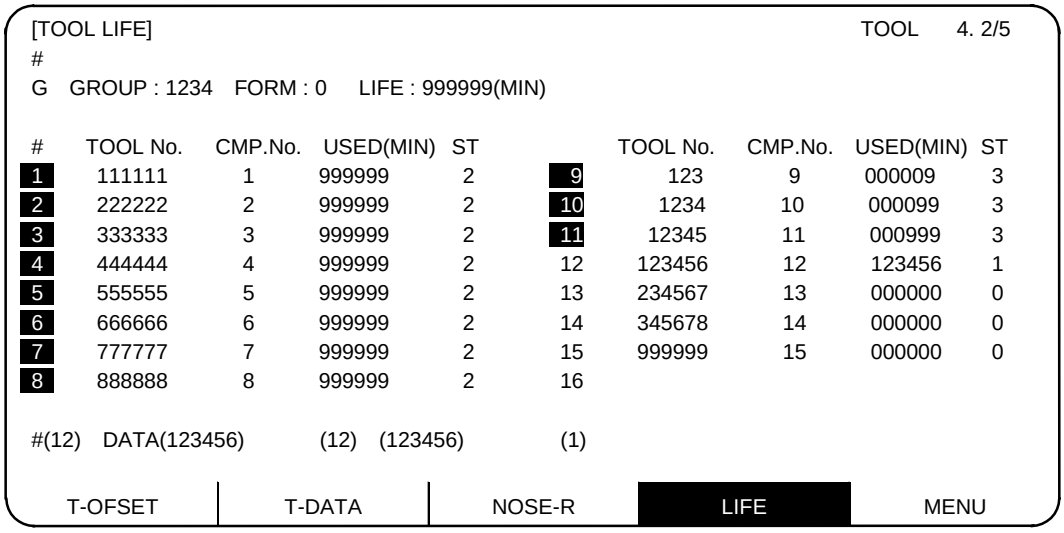
The tools used are classified into several groups. With this tool life management with spare tool function, tool life (usage time, count) is managed for each group. When a tool's life is reached, an equivalent spare tool is selected in order from the group to which that tool belongs.

(1) No. of tool life management tools : 1-system: max. 80 tool, 2-system: max. 40 tools/system
(2) No. of groups : 1-system: max. 80 tool, 2-system: max. 40 tools/system
(3) Group No. : 1 to 9999
(4) No. of tools per group : Max. 16 tools
(5) Service lifetime : 0 to 999999 min. (approx. 16667 hours)
(6) Service life count : 0 to 999999 times

3.5.1 Group Registration

(1) Tool life management screen

A group's life management information is set and displayed.



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

1) Selecting a display group

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

2) Registering a group

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

• Designate a group No. from 1 to 9999.
• Set FORM with for group life management by either time or count.
0: Time 1: Count
If the FORM setting is omitted, the method becomes "0" (time).
• Set LIFE with the service life setting value for that group's tools. (0 to 999999).
If the LIFE setting is omitted, the life setting value becomes "0".

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

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

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

3) Deleting a group registration

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

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.

6) Clearing a tool's USED data

Setting up the following can clear the USED data:
0 is set in the USED data of each tool.
Put "-" in front of the # number of tool, and set that number with "-" in # ( ).
When the USED data of the all registration tools of 1 group are cleared, -99 is set in #( ).

7) Displaying multiple groups

The LIFE management information of multiple groups is set and displayed in 1 screen according to the parameter (#1107 Tllfsc) setting.

"#1107 Tllfsc" setting value	0	1	2
No. of display groups	1	2	4
Maximum number of registered tools	16	8	4

[TOOL LIFE] TOOL 4.1/12

GROUP: 1 FORM:0 LIFE: 100(MIN) GROUP: 2 FORM:1 LIFE: 100(MIN)

#	TOOLNo.	CMP.No.	USED(MIN)	ST	#	TOOLNo.	CMP.No.	USED(SET)	ST
1	1	1	100	2	1	21	21	100	2
2	2	2	0	3	2	22	22	100	2
3	3	3	0	3	3	23	23	100	2
4	4	4	50	1	4	24	24	100	2
5	5	5	0	0	5	25	25	100	2
6	6	6	0	0	6	26	26	100	2
7	7	7	0	0	7	27	27	100	2
8	8	8	0	0	8	28	28	100	2

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

T-OFSET T-DATA NOSE-R **LIFE** MENU

The LIFE management information of 2 groups in 1 screen

[TOOL LIFE] TOOL 4.1/7

GROUP: 1 FORM:0 LIFE: 100(MIN) GROUP: 2 FORM:1 LIFE: 100(MIN)

#	TOOLNo.	CMP.No.	USED(MIN)	ST	#	TOOLNo.	CMP.No.	USED(SET)	ST
1	1	1	100	2	1	21	21	100	2
2	2	2	0	3	2	22	22	100	2
3	3	3	0	3	3	23	23	100	2
4	4	4	50	1	4	24	24	100	2

GROUP: 3 FORM:0 LIFE: 300(MIN) GROUP: 4 FORM:1 LIFE: 400(MIN)

#	TOOLNo.	CMP.No.	USED(MIN)	ST	#	TOOLNo.	CMP.No.	USED(SET)	ST
1	29	29	100	1	1	33	33	0	0
2	30	30	0	0	2	34	34	0	0
3	31	31	0	0	3	35	35	0	0
4	32	32	0	0	4	36	36	0	0

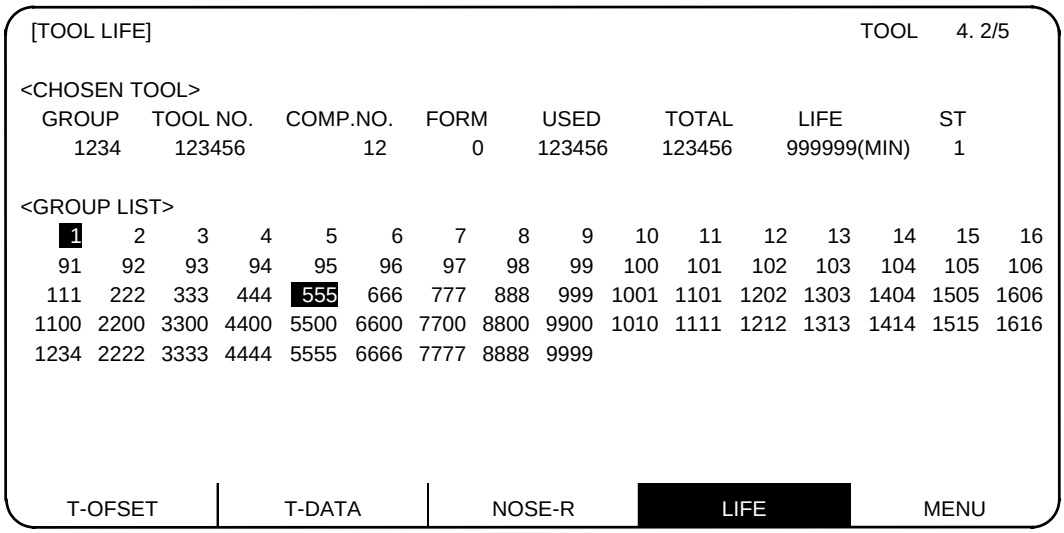
#( )DATA( )( )( )( )

T-OFSET T-DATA NOSE-R **LIFE** MENU

The LIFE management information of 4 groups in 1 screen

(2) Registration group list screen

The life management data of the tool currently being used and the list of registered groups of tools are displayed. This page is mainly used for monitoring tool life data in group units.



1) Display details

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

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

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

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

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

2) Erasing all registered data of a group

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

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. The count method and timing for the No. of uses (count) method can be changed to type 2 with the parameter setting (#1277 ext13/bit0).
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.

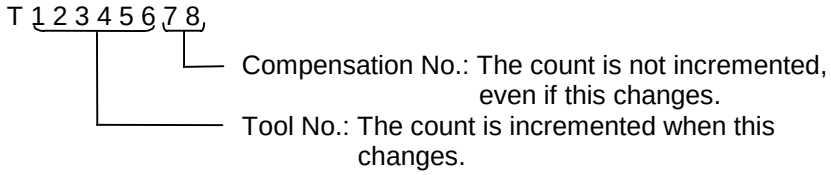
(2) No. of uses (count) incrementation with the count method

(a) Type 1 (#1277 ext13/bit0: 0)

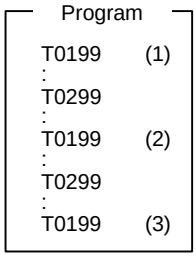
Incrementation is carried out when the No. of the tool being used changes by the execution of a tool selection command (T****99) during the cutting mode (except during machine lock, miscellaneous function lock, dry run, and single block states). (If the mode never changes to the cutting mode after the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

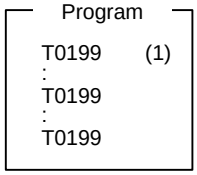
If the T code of the current tool is 12345678:



<< 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.

(b) Type 2 (#1277 ext13/bit0: 1)

Only the group used for cutting from when the machining program starts to when it is reset is incremented by "1". The count is made at the reset.

If recount M is commanded, the group used up to that point will be incremented by "1" in the counter.

(Note 1) A count is not made in the machine lock, miscellaneous function lock or dryrun states.
(Note 2) During single block, select whether to count with the parameter.
(Note 3) The maximum value of the life is 999999 times.

(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.

3.5.3 Parameters

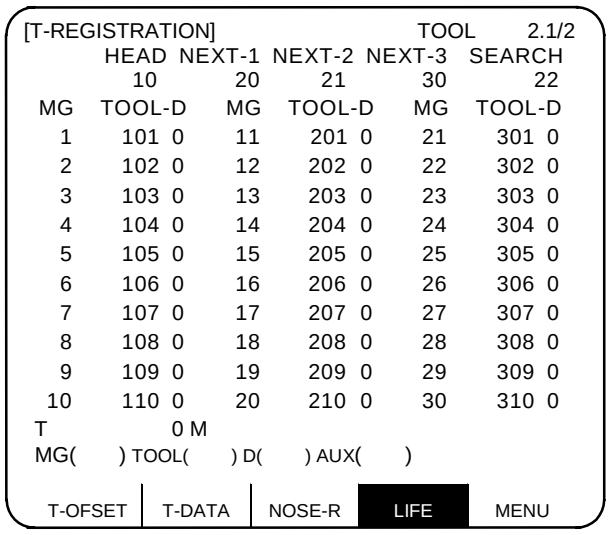
The tool life management specifications will differ according to parameter #1096 T_Ltype and #1106 Tcount. Confirm the explanation for the relevant setup parameter data item.

3.6 Tool Registration

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

3.6.1 Outline of Functions

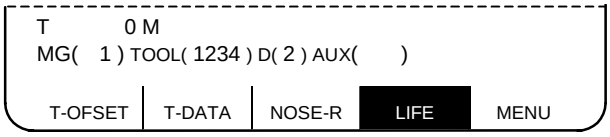
(1) Tools used can be registered in the magazine pot.
(2) When the magazine pot and the tool No. are changed by a tool selection command or a tool replacement command, the new tool No. is displayed.
(3) Random data can be set in AUX ( ) in the setting area and processed as a sequence with the user PLC.
(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 numeric command.



3.6.2 Tool Registration in the Magazine Pot

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

☞



Press the INPUT key. ☞

```
[T-REGISTRATION]                    TOOL     2.1/2
      HEAD  NEXT-1  NEXT-2  NEXT-3  SEARCH

 MG  TOOL-D   MG  TOOL-D   MG  TOOL-D
  1  1234-0   11          21
  2           12          22
  3           13          23
  4           14          24
      (           (
```

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 key is pressed the 2nd time, the data set in the data area appears.

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

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.

```
[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
  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(   )
 T-OFSET | T-DATA | NOSE-R | LIFE | MENU
```

"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 machine maker, the data is always set by an input of N0.

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.

(Note) If any other key has been pressed before the INPUT key is pressed, the tool registration data will not be deleted.

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

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

(1) Changing from the normal data setting mode to the manual numeric 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 numeric command
mode.
Confirm that the mode has changed over by checking this difference. The operation is as follows:

```
T          M
MG( □) TOOL(          )  D( ) AUX( )
```

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

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

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
MG(    ) TOOL(          )  D( ) AUX( )
```

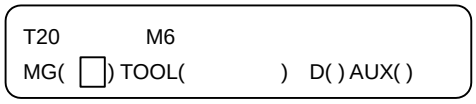
1) The data in the setting area is cleared, and the cursor disappears from the screen.

(2) Executing the manual numeric 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 numeric 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 numeric command operation is the same as the operation on the POSITION screen. Refer to the section on manual numeric commands for the MONITOR and POSITION screens for details.

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

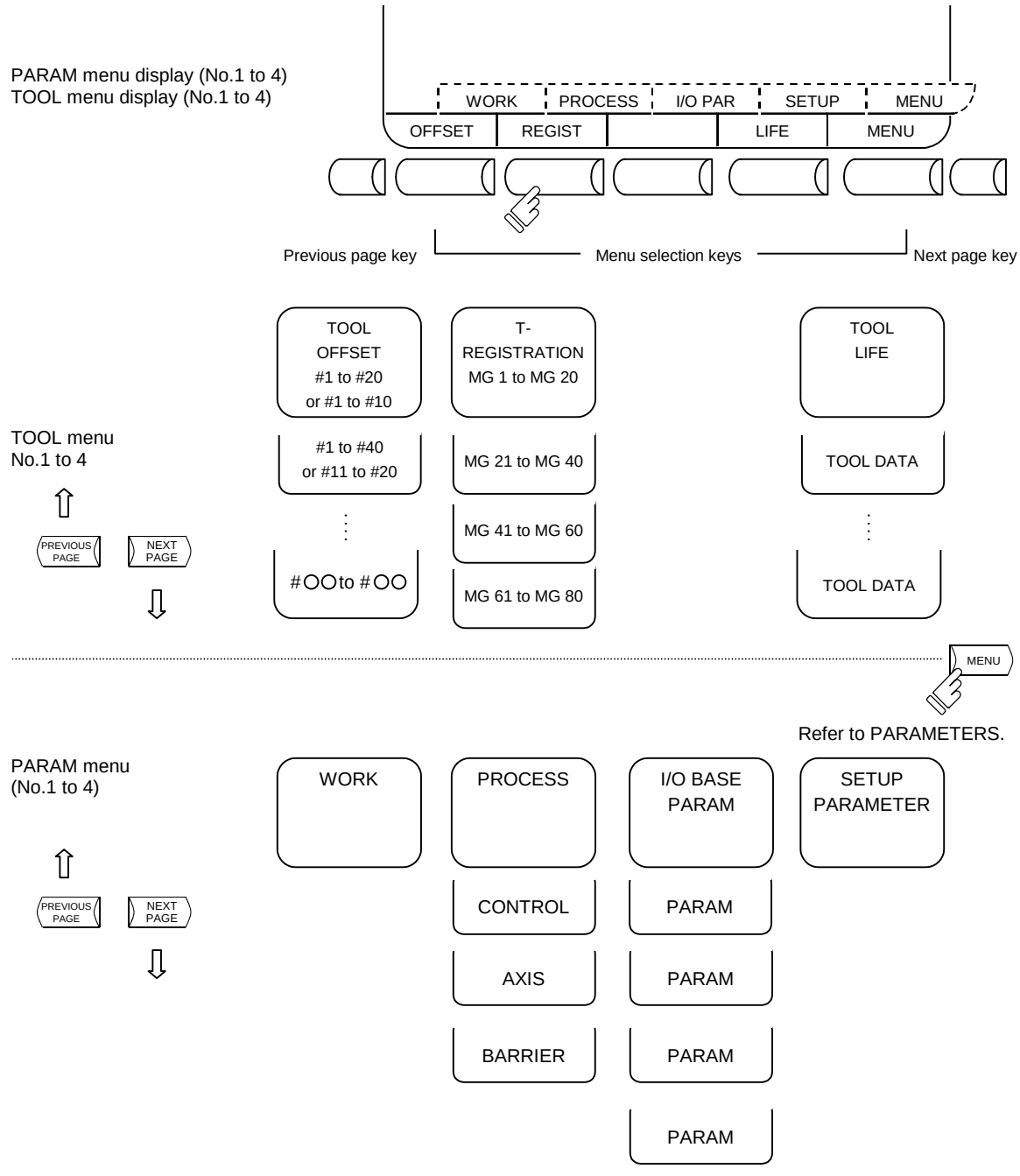
Press the [↓] key. The normal data setting mode returns.



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

3 (II). Tool Offset (M system)

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



⚠ CAUTION

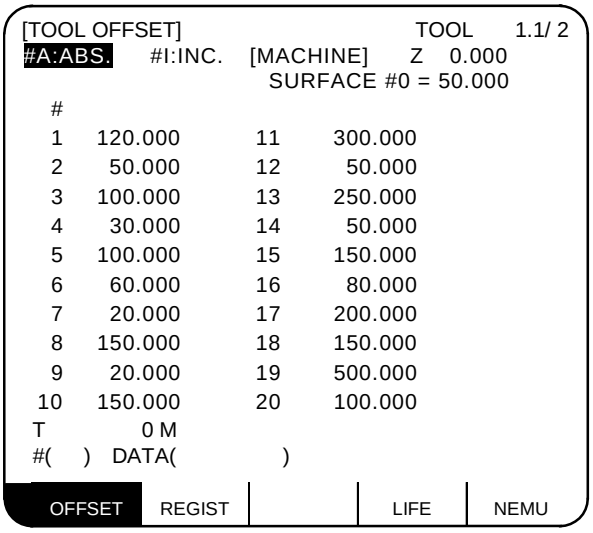
⚠ If a tool offset or workpiece coordinate system offset is changed during automatic operation (including during single block stop), the new offset is validated from the command of the next block or blocks onwards.

3.1 Tool Offset

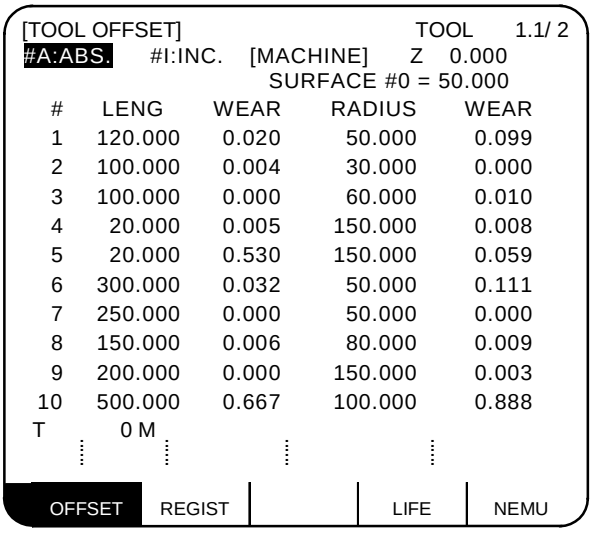
When the menu key ⟩OFFSET⟨ is presented, the TOOL OFFSET screen is displayed.

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

(2) Tool offset memory (type II: parameter #1037 cmdtyp 2)
Set the form 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 radius 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.

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 key.

(2) For type II
To set tool offset data, set the offset memory number in # ( ) and offset data in the setting area corresponding to LENG, WEAR, RADIUS and WEAR, then press the INPUT key.

(3) If the INPUT key is pressed after the offset memory number and tool offset data are set, the tool offset data set in the offset memory number position is displayed, the offset memory number in the setting area # ( ) is incremented by one, and the contents of DATA ( ) disappear. At the time, the cursor moves to the right end of the same setting field as the input time.

(4) If tool offset data is set with an offset memory number not contained in the displayed offset memory numbers, the screen changes to the screen corresponding to the setup offset memory number when the INPUT key is first pressed. When the INPUT key is pressed again, the tool offset data set in the offset memory number position is displayed.

(5) The offset memory number displayed in # ( ) can be consecutively incremented or decreased by one by pressing the ↑ or ↓ key.

(6) To set the incremental mode, enter I in # ( ), then press the INPUT key.
In incremental mode, the set data is added to the data indicated in the display area. To cancel the incremental mode, enter A in # ( ), then press the INPUT key; the absolute mode is set. (For details, see Sections 3.1.3.)

3.1.2 Tool Offset Data Clear

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

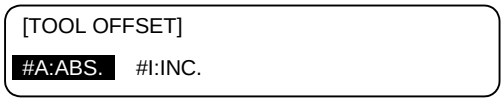
(Note) If any other key has been pressed before the INPUT key is pressed, the offset data will not be cleared.

3.1.3 Tool Offset Data Setting Modes (Absolute and Incremental)

There are two types of selection method in the absolute value setting or incremental value setting for the tool offset data: the mode selection method and the menu selection method.
The required method is selected with the parameter #1136 optype.

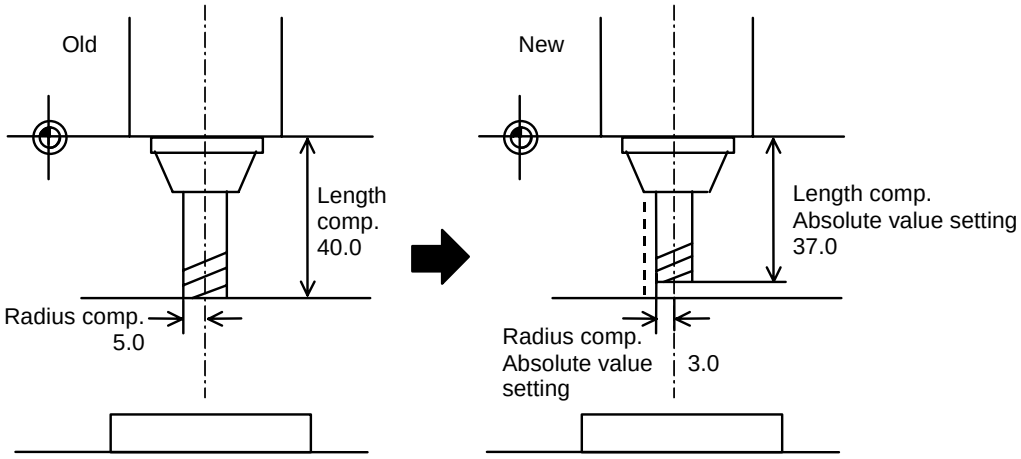
(1) Mode selection method (#1136=0)
(a) Absolute value setting
Change to the absolute value setting mode as follows:

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

(b) Incremental value setting

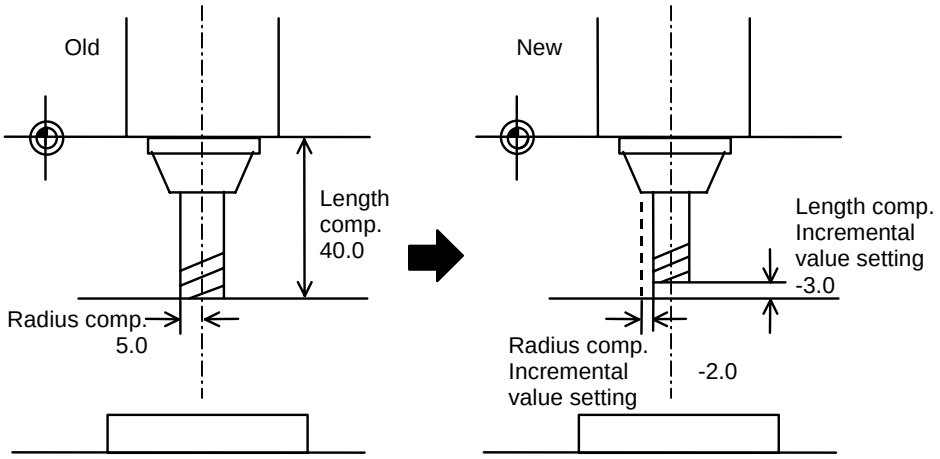
Change to the incremental value setting mode as follows:

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 value setting mode is valid.

Example of setting tool offset data in incremental value setting mode



(#3 length compensation data)

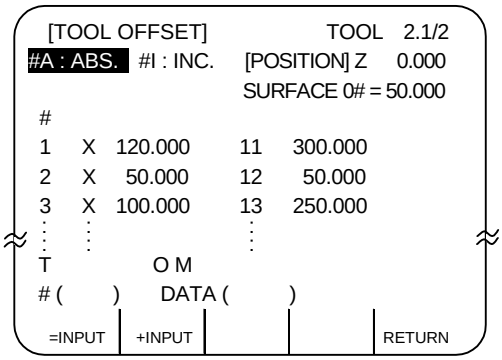
Display	Setting	Display
#3 40.000	# (3) (-3)	#3 37.000

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

(2) Menu selection method (#1136=1)

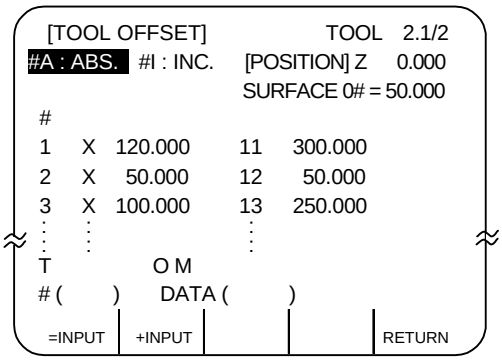
(a) Change over to the absolute value setting mode

Set value in #( ), X( ) or Z( ), then press the = INPUT key.



(b) Change over to the incremental value setting mode

Set value in #( ), X( ) or Z( ), then press the + INPUT key.



* When the cursor is moved to X( ) or Z( ) then the data key is pressed, =INPUT,+INPUT menu will appear automatically.

(c) Supplement

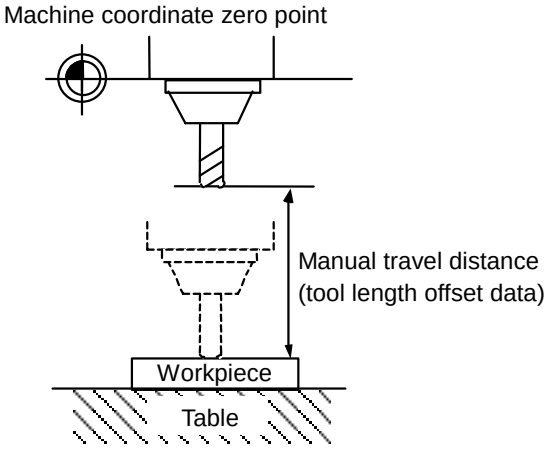
- If INPUT key is pressed while = INPUT, + INPUT are displayed, the offset data can be set for mode selection method.
- = INPUT, + INPUT are displayed until BACK menu key is pressed or other screens are selected. The screen selection menu is displayed when TOOL OFFSET screen is selected again after other screens are selected once.
- "↓ " Mark of the screen selection menu indicates that the operation menu exist on the screen being displayed.
- The operation menu is invalid during the tool measurement with manual tool length measurement function. The operation menu during the tool measurement operates as follows.
  1) When the screen select menu is displayed
  + INPUT, = INPUT don't appear during the tool measurement. If = INPUT, + INPUT with "↓ " mark are pressed during the tool measurement, ERROR (E74) will occur.
  2) When the operation menu is displayed
  If = INPUT, + INPUT are pressed during the tool measurement, ERROR (E74) will occur.
- = INPUT, + INPUT are invalid during the manual numerical command mode. If = INPUT, + INPUT are pressed during the manual numerical command mode, ERROR (E74) will occur. If the screen selection menu displayed with "↓ " mark is pressed during the manual numerical command mode, ERROR (E74) will occur.
- When the offset memory No. not displayed on the screen is set in setting area # ( ), the screen corresponding to the offset memory No. set in will appear by pressing = INPUT,+ INPUT key of one time. The offset data will be set by pressing = INPUT, + INPUT key again.
- When the tool nose point data is set on the NOSE-R screen, the absolute setting is selected whichever of = INPUT and + INPUT key is pressed.

3.1.4 Manual Tool Length Measurement

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

(1) Tool length measurement I

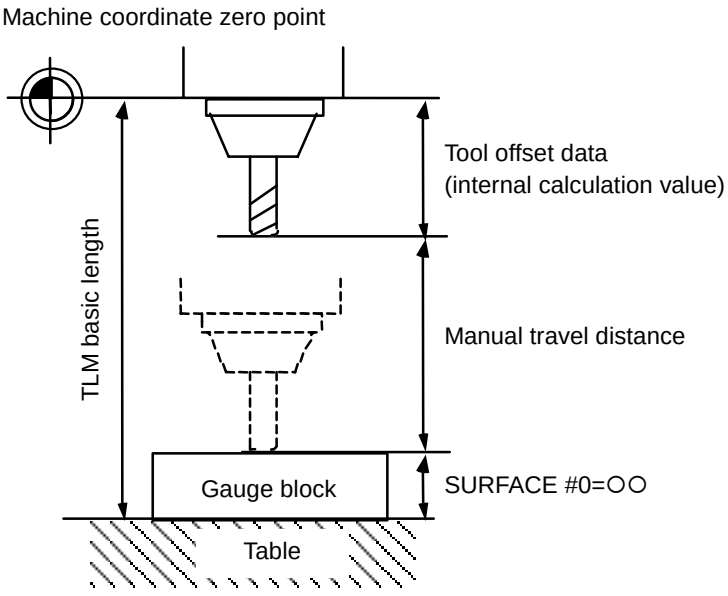
When the tool is placed in the machine coordinate zero point, the distance from the tool tip to measurement point (workpiece top end) can be measured and set as tool offset data.



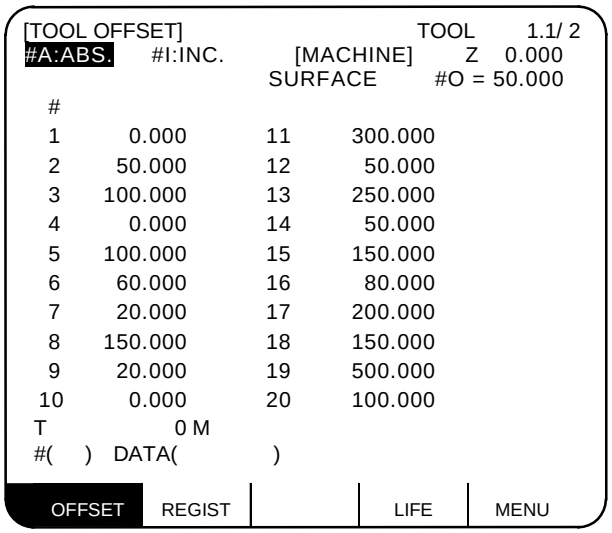
When TLM basic length (#1102 tlm) = 0 and SURFACE #0 = 0 are set, tool length measurement I mode is set.

(2) Tool length measurement II

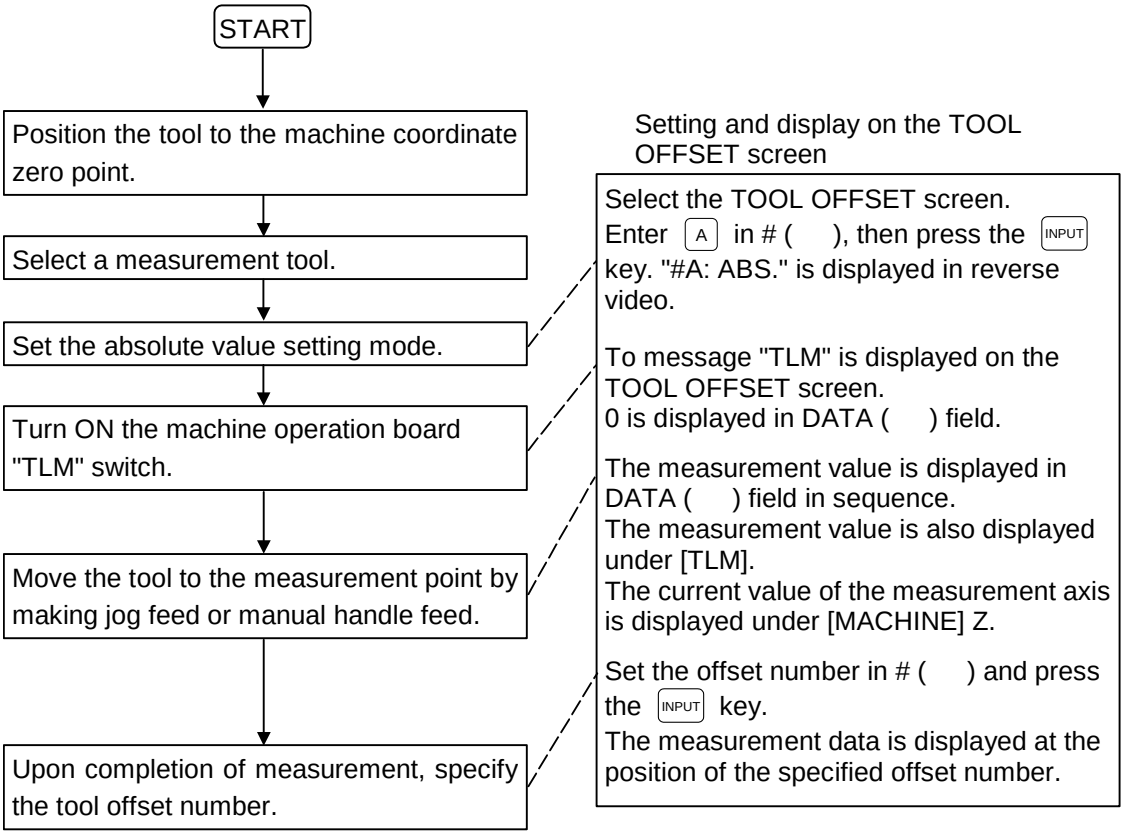
When the tool is placed in the machine coordinate zero point, the distance from the reference point to tool tip can be measured and set as tool offset data.



(3) Tool offset data setting by tool length measurement

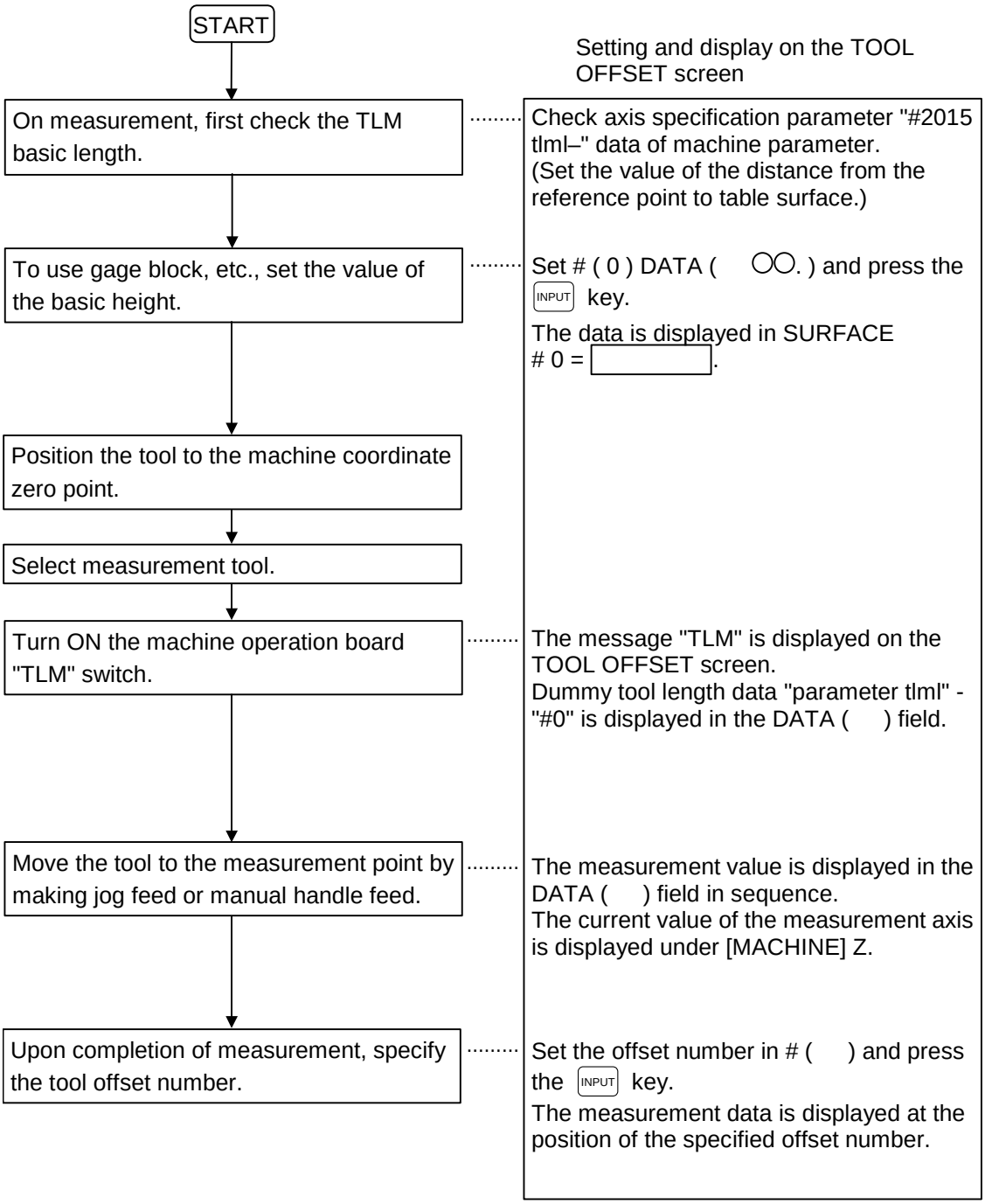


1) Tool length measurement I



(Note) For operation procedure, see Machine Operation manual.

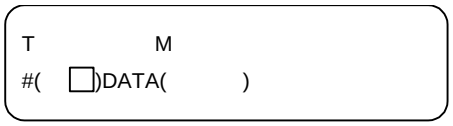
2) Tool length measurement II



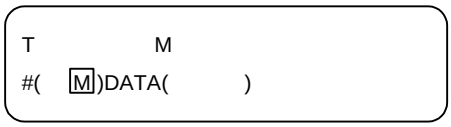
3.1.5 Manual Numeric Command Operation on the TOOL OFFSET Screen (M, T)

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:

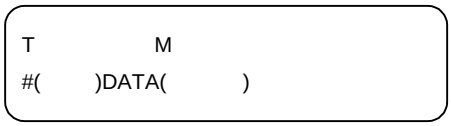


Set [M] (Manual) in the first parenthesis pair of the setting field.



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

Press the [INPUT] key. The mode changes to manual numerical 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 "Manual numeric 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.

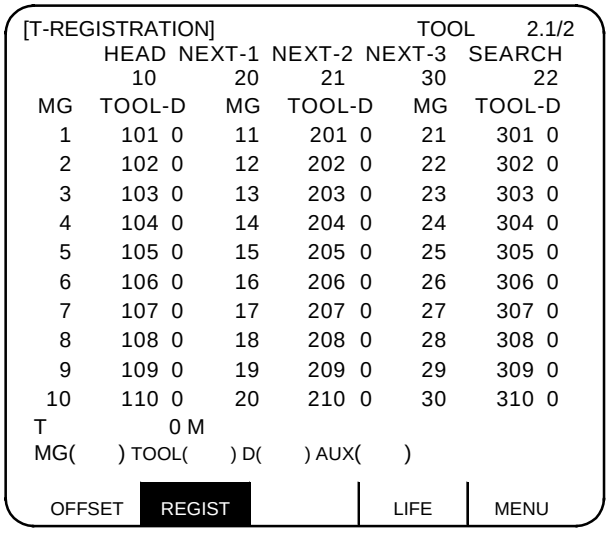
1) The cursor is displayed in first parenthesis pairs. After this, usual data setting is enabled.

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 maker.

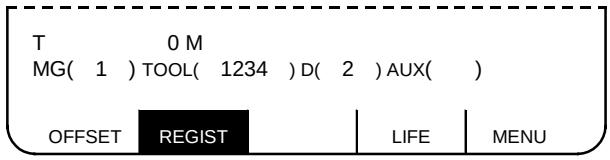
3.2.1 Function Outline

(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.

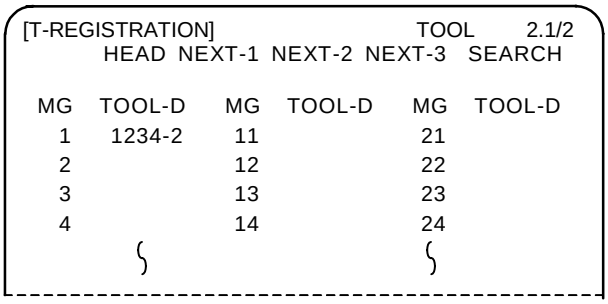


3.2.2 Tool Registration in Magazine Pot

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



Press the INPUT key.



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

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

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.

☞

[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
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

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 maker, data is set by using SP And N1 to N4.

3.2.4 Tool Registration Data Clear

Set CL in MG ( ).

Press the INPUT key.

☞

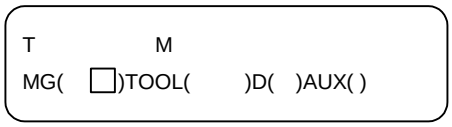
All data displayed in HEAD, NEXT 1 to NEXT 3, INDEX, and MG1 to MGn is cleared.

(Note) If any other key has been pressed before the INPUT key is pressed, the tool registration data will not be cleared.

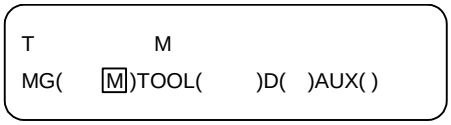
3.2.5 Manual numeric Command Operation on the TOOL REGISTRATION Screen (M, T)

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 numerical command.

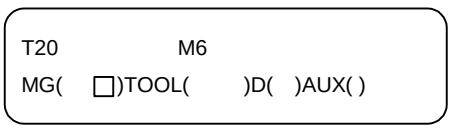
1) Data is cleared from the setting field. The cursor is also cleared from the screen.

(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 "2.1.3 Manual numeric 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.

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.

3.3.1 Function Outline

Tool life management is configured of the following two functions.

(1) The use time or count of the tool mounted on the spindle is accumulated and the tool use state is monitored.
(2) A spare tool is selected among programmed tool commands. Tool position offset and tool diameter compensation are performed for the selected tool.

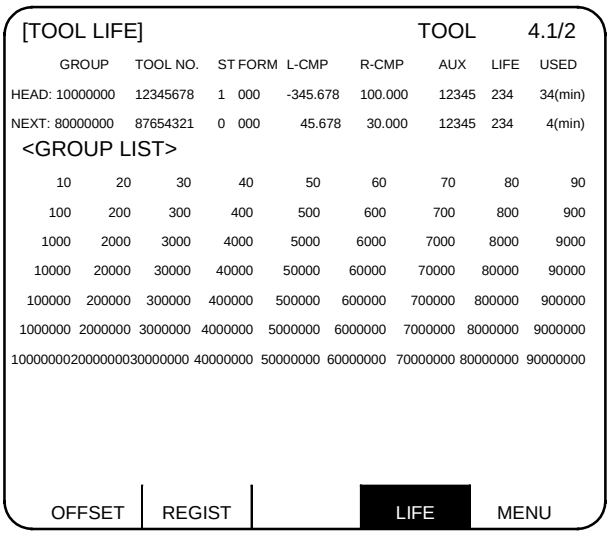
Related parameters

#	Items		Details	Setting range (unit)
1103	T_Life	Validate life management	Select the usage of the tool life management function.	0: Do not use. 1: Perform tool life management control.
1104	T_Com2	Tool command method 2	Select the command method for when #1103 T_Life is set to 1. 0: Handle the program tool command as the group No. 1: Handle the program tool command as the tool No.	0/1
1105	T_sel2	Tool selection method 2	Select the tool selection method for when #1103 T_Life is set to 1. 0: Select in order of registered No. from the tools used in the same group. 1: Select the tool with the longest remaining life from tools used in the same group and the unused tools.	0/1

(Note) Mainly the screen operations are explained in this manual. Refer to the "PLC Programming Manual (Ladder Section)" for details on using the tool life management function.

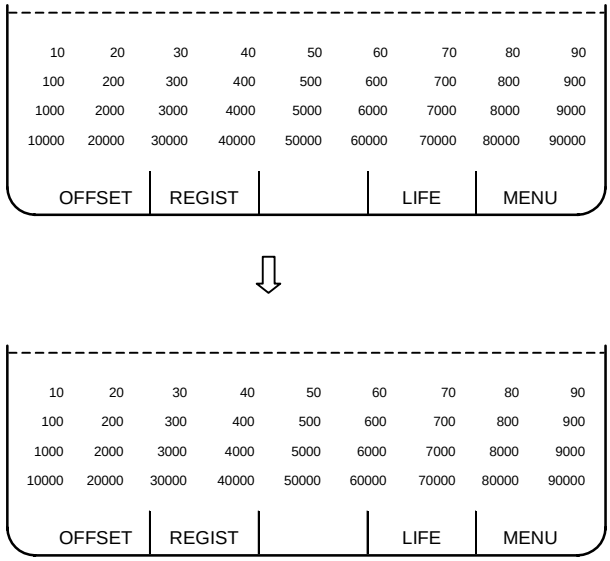
3.3.2 TOOL LIFE Screen Data Display

(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.

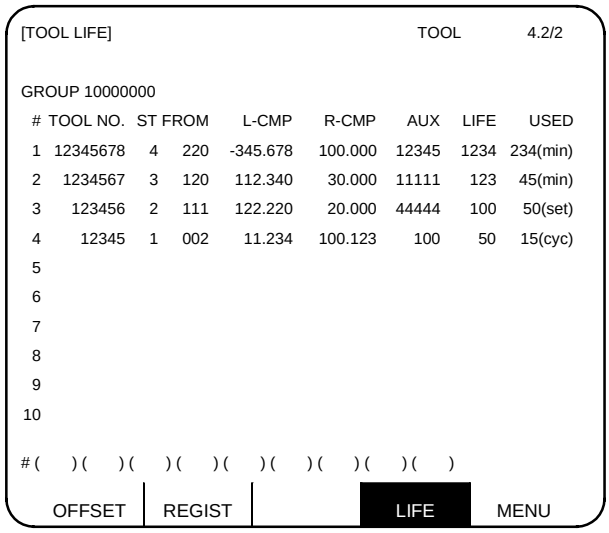
A maximum of 90 group numbers are displayed on one screen. If the number of the group numbers exceeds 90, the screen is scrolled every line by using the [↑] or [↓] key.

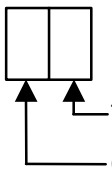


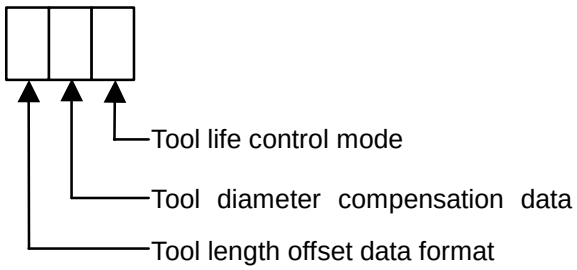
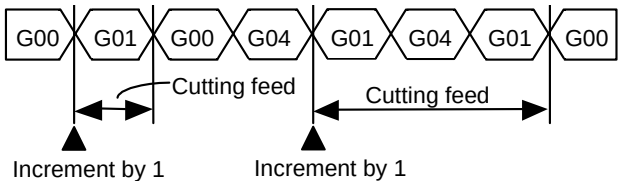
(2) TOOL LIFE data screen

The TOOL LIFE data displays tool data in group units. If the number of lines displayed exceeds one screen area, press the NEXT PAGE or PREVIOUS PAGE 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. A tool having the same group number is handled as a spare tool.	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. This is a number unique to the tool actually output during the tool command, etc.	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 maker 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 spindle tool. Note that if cutting feed (G01, G02, G03, etc.) is not commanded even once after the tool is set as the spindle tool, the mounting will not be counted. 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 diameter 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 work program for compensation. 1: Addition compensation amount 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 work program for compensation. 2: Direct compensation amount 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 work program for compensation.	

Display item	Explanation	Setting range
FORM	(c) Tool length offset data format 0: Offset number 1: Addition offset amount 2: Direct offset amount The functions are the same as in (b) above.	
L-CMP R-CMP	These depend on the data format specified in "FORM".	Compensation number 1 to 400 Addition compensation amount ±1 to 99999.999 Direct compensation amount ±1 to 99999.999
AUX	This depends on the machine maker 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 hole 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 9999/65000 (times) Work count 0 to 9999/65000 (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 9999/65000 (times) Work count 0 to 9999/65000 (times)

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

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

(1) Data display

When the menu key [NEXT PAGE] 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 PAGE] key.

[TOOL LIFE] TOOL 4.2/2

GROUP 123

#	TOOL NO.	ST	FROM	L-CMP	R-CMP	AUX	LIFE	USED
1	12345678	4	220	-345.678	100.000	12345	1234	234(min)
2	1234567	3	120	112.340	30.000	11111	123	45(min)
3	123456	2	111	122.220	20.000	44444	100	50(set)
4	12345	1	002	11.234	100.123	100	50	15(cyc)
5								
6								
7								
8								
9								
10								

( ) ( ) ( ) ( ) ( ) ( ) ( ) ( )

OFFSET | REGIST | | LIFE | MENU

(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 is registered, and a setting number incremented by one is set in #().

[TOOL LIFE] TOOL 4.2/2

GROUP 123

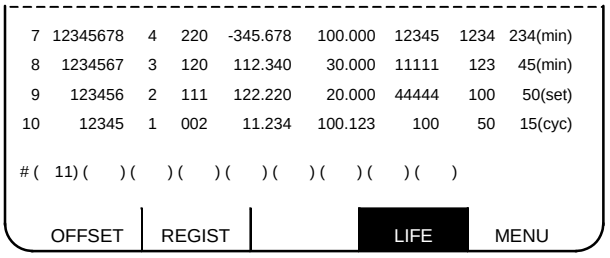
#	TOOL NO.	ST	FROM	L-CMP	R-CMP	AUX	LIFE	USED
1	12345678	4	220	-345.678	100.000	12345	1234	234(min)
2	1234567	3	120	112.340	30.000	11111	123	45(min)
3	123456	2	111	122.220	20.000	44444	100	50(set)
4	12345	1	002	11.234	100.123	100	50	15(cyc)
5								
6								
7								
8								
9								
10								

(5) ( ) ( ) ( ) ( ) ( ) ( ) ( )

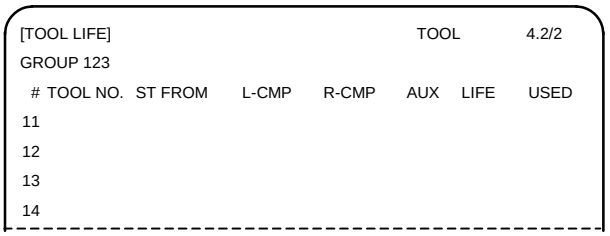
OFFSET | REGIST | | LIFE | MENU

(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 PAGE or PREVIOUS PAGE key or setting the number in setting area # ( ).
5) By changing a registered tool number to 0, the tool can be deleted.

(4) Deletion in group units

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

(Note) If any other key has been pressed before the INPUT key is pressed, the TOOL LIFE data will not be deleted.

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

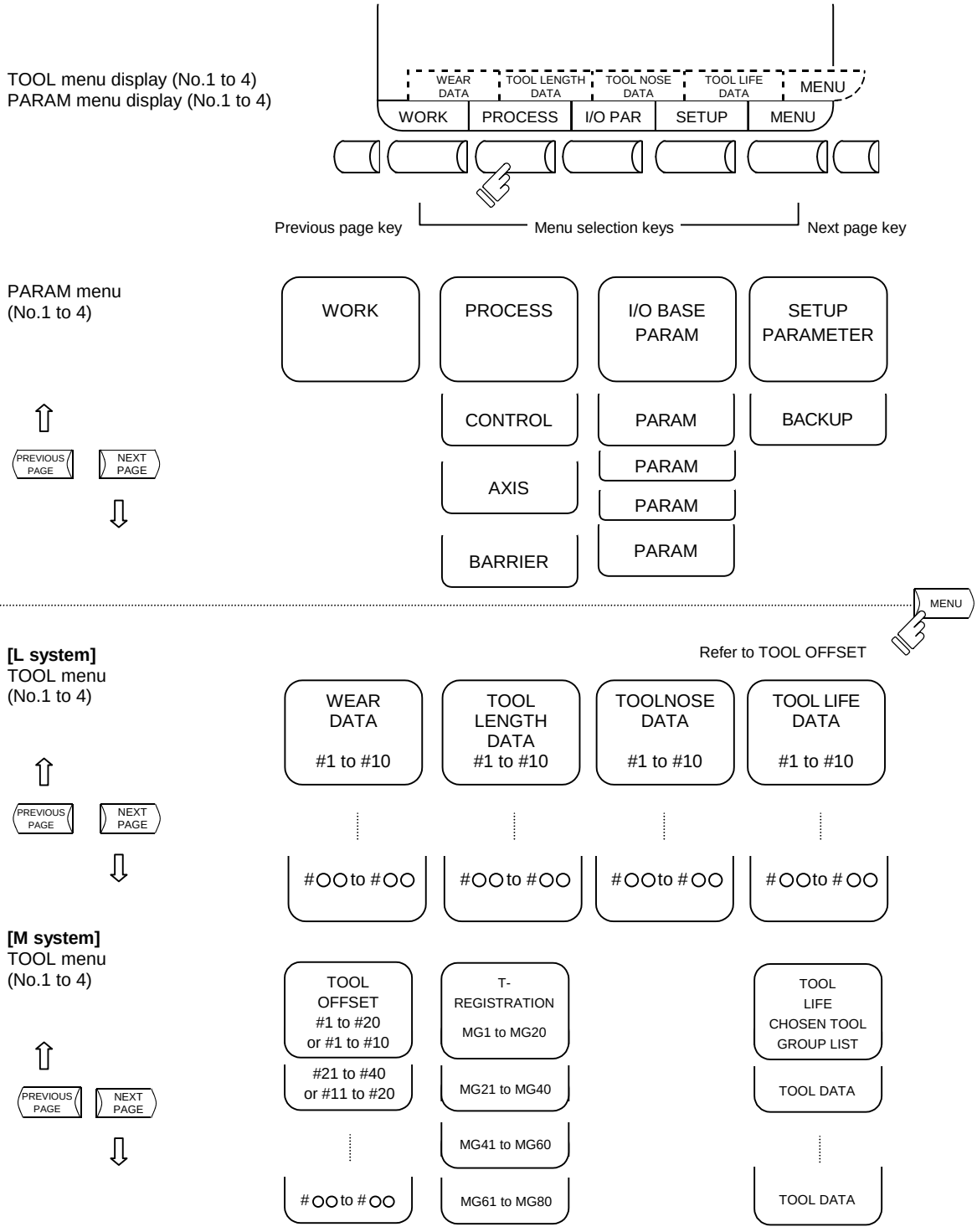
To clear all data, select the HEAD, NEXT, GROUP LIST screen and press SHIFT key, C.B CAN key, then INPUT key.

(Note) If any other key has been pressed before the INPUT key is pressed, the TOOL LIFE data will not be cleared.

4. Parameters (User)

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

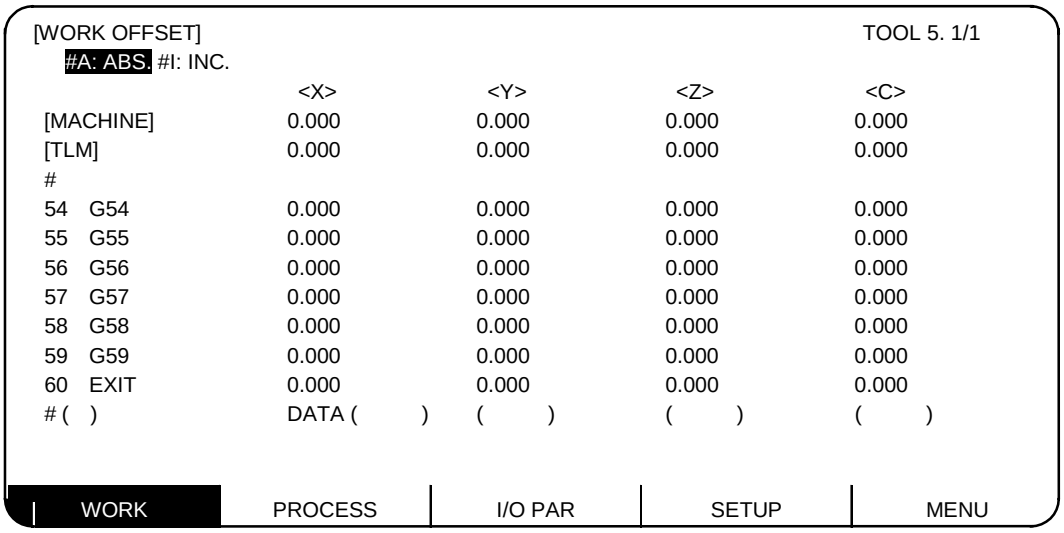
TOOL menu is displayed after the power is turned ON. To display PARAM menu, use menu key [MENU] on the TOOL screen.



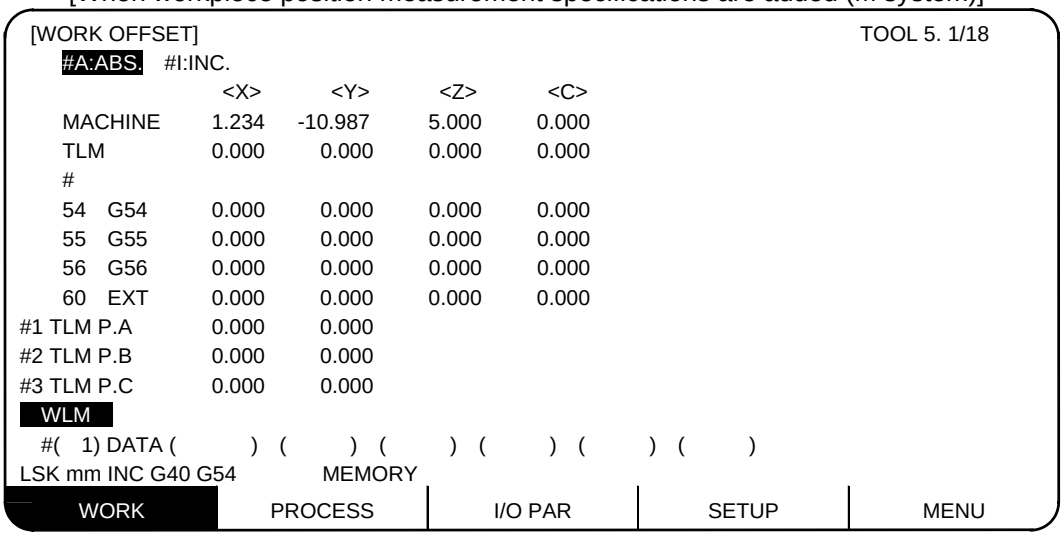
4.1 Workpiece Coordinate

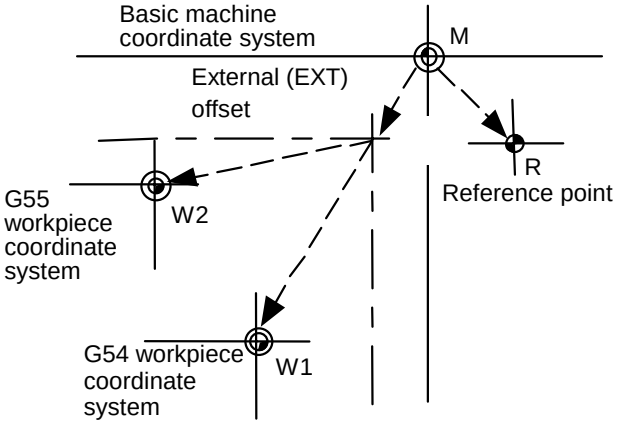
Pressing the menu key 〉WORK〈 displays the WORK OFFSET screen.

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



[When workpiece position measurement specifications are added (M system)]



#	Parameter	Explanation	Setting range (units)
54 55 56 57 58 59 60 101 : 148	G54 offset G55 offset G56 offset G57 offset G58 offset G59 offset EXT offset P1 : P48	Specify the workpiece coordinate system and external workpiece coordinate offset from G54 to G59, and P1 to P48. Workpiece coordinate system offset data can be specified in absolute or incremental values.  **(Note)** P1 to P48 are options.	±99999.999 (mm)
#1	TLM P.A (M system)	The coordinate value of the measured position (X, Y) of the first point of he hole center workpiece offset Measurement or the width center workpiece offset Measurement is set. Then “TLM P.A” is highlighted. If the measurements switch ON or workpiece coordinate is set, the coordinates value of measured position is cleared to 0.	
#2	TLM P.B (M system)	The coordinate value of the measured position (X, Y) of the first point of he hole center workpiece offset Measurement or the width center workpiece offset Measurement is set. Then “TLM P.B” is highlighted. If the measurements switch ON or workpiece coordinate is set, the coordinates value of measured position is cleared to 0.	
#3	TLM P.C (M system)	The coordinate value of the measured position (X, Y) of the first point of he hole center workpiece offset Measurement or the width center workpiece offset Measurement is set. Then “TLM P.C” is highlighted. If the measurements switch ON or workpiece coordinate is set, the coordinates value of measured position is cleared to 0.	

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.

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 pressing the arrow keys [↑] [↓].

(4) Typing [I] in # ( ) and pressing the [INPUT] key puts the setting mode to the incremental mode. Data entered in the incremental mode is added to the data in the setting field.
Typing [A] in # ( ) and pressing the [INPUT] key cancels the incremental mode and restores the absolute mode.

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.

4.1.3 Displaying Machine Position Data

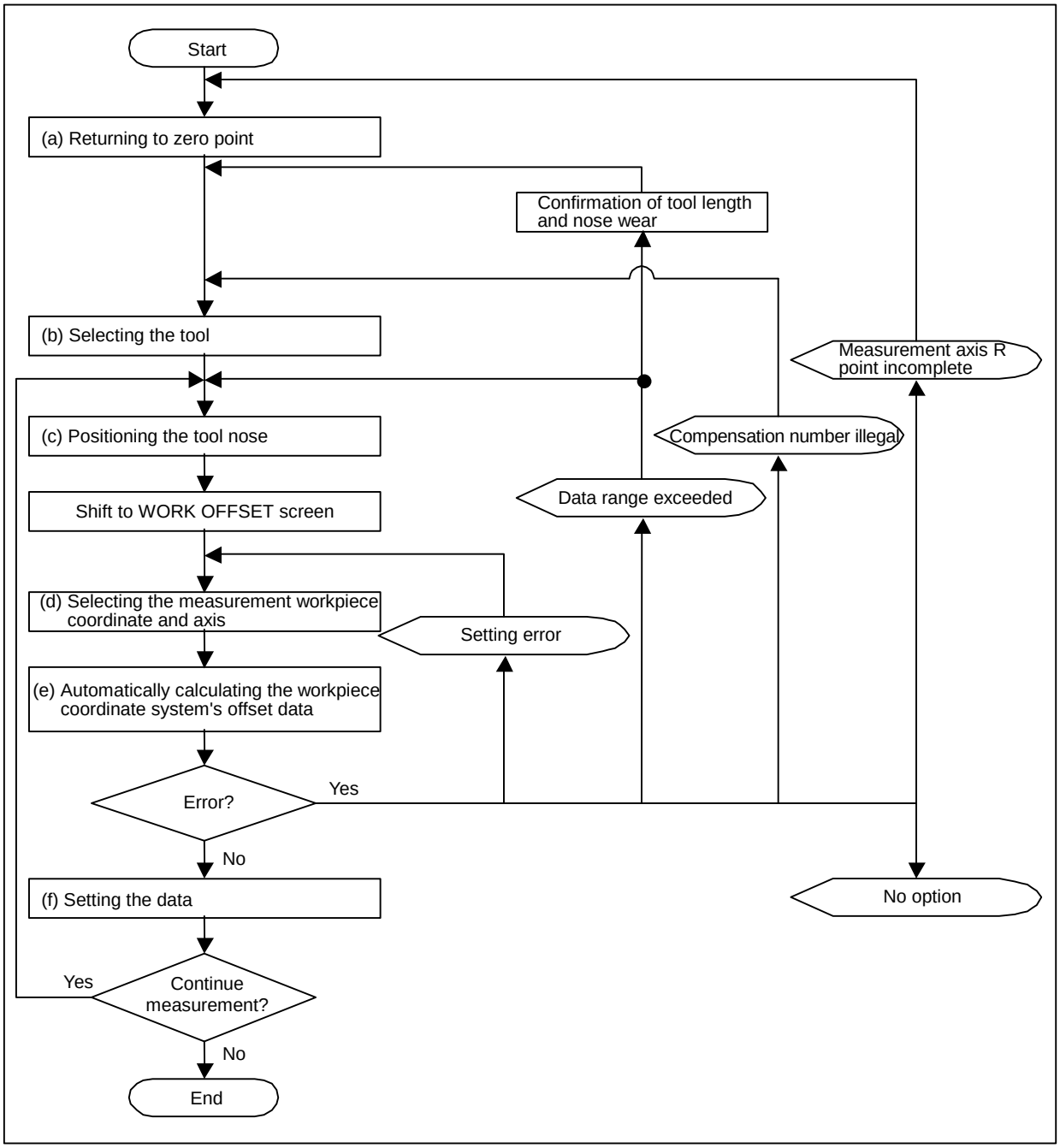
As with the POSITION screen, data of each axis displayed at the [MACHINE] on the WORK OFFSET screen indicates the current machine position in reference to the zero point on the basic machine coordinate system; it cannot be changed on this screen.

4.1.4 Workpiece Coordinate Offset Measurement Function (L System)

(1) Outline

The workpiece coordinate offset data is automatically calculated when the tool nose is aligned to the workpiece coordinate zero point and the SHIFT INPUT keys (CALC key) are pressed. The calculation results are displayed in the setting area on the screen.

(2) Operation procedures



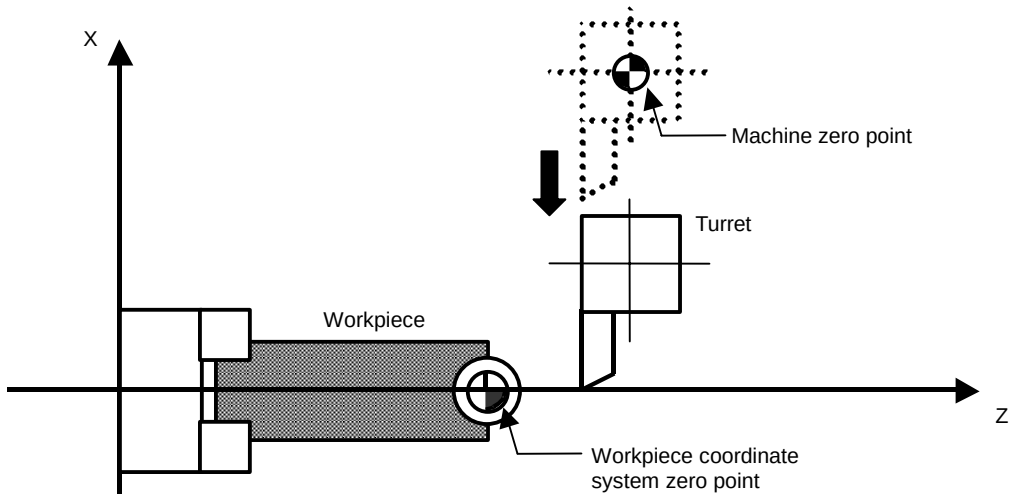
Steps for measuring workpiece coordinate offset (L system)

(a) Returning to zero point
After turning the power ON, establish the coordinate system with the dog-type zero point return. If the absolute position is not established when using the absolute position detection specifications, carry out initialization first.
(Note) This is not required if the axis to be measured is an axis with no zero point ("#2031 noref" = 1).

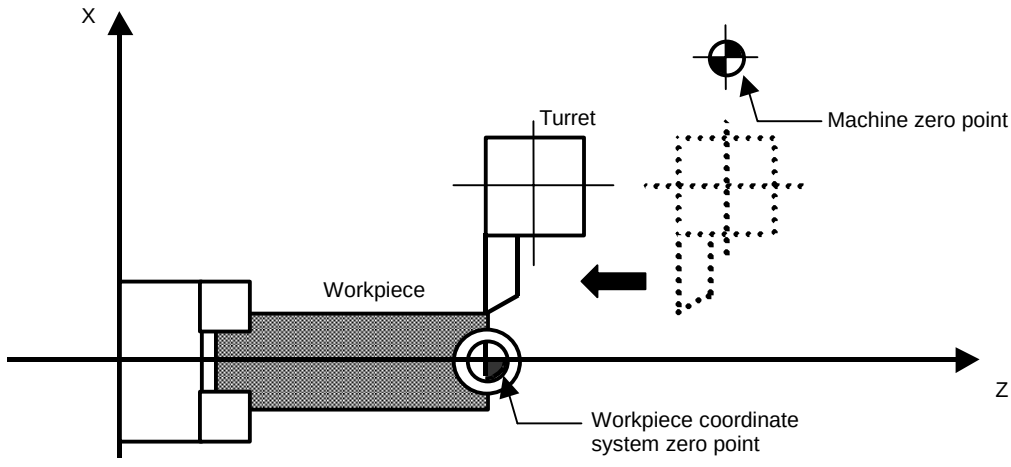
(b) Selecting the tool
Execute the T command with the "manual numeric command" on the "POSITION" screen or with the MDI operation, select the tool.
(Note 1) Set the offset number of the selected tool in the R register.
(When setting from the user PLC, set as a BCD code.)
(Note 2) Preset the "tool length/wear data" for the tool to be used.

(c) Positioning the tool nose
Using JOG or the handle, move the nose of the axis to be measured above the workpiece coordinate system zero point. The workpiece coordinate system offset data is measured one axis at a time.

1) 1st axis (X axis) workpiece coordinate offset measurement



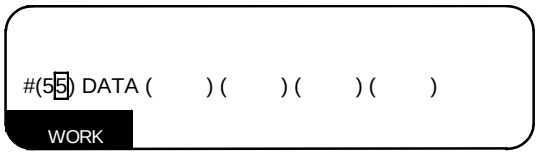
2) 2nd axis (Z axis) workpiece coordinate offset measurement



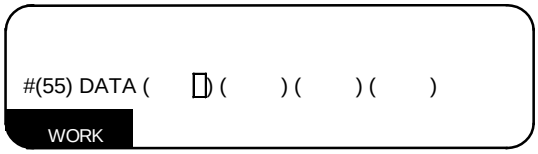
(d) Selecting the measurement workpiece coordinate and axis
Set the workpiece coordinates to be measured in the # ( ) setting area, and then move the cursor to the setting area of the axis to be measured.

(Example) To measure the X axis (1st axis) of the G55 workpiece coordinate system.

1) Set 55 in # ( ).



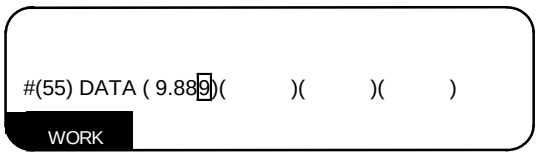
2) Move the cursor to the X axis setting area.



(e) Automatically calculating the workpiece coordinate system offset data
When the SHIFT INPUT keys are pressed, the selected axis' workpiece coordinate offset data will be automatically calculated from the machine value, tool length data and tool nose wear data. The calculation results will be displayed in the setting area.
The setting mode (absolute value setting/increment value setting) will automatically be set to the absolute value setting.

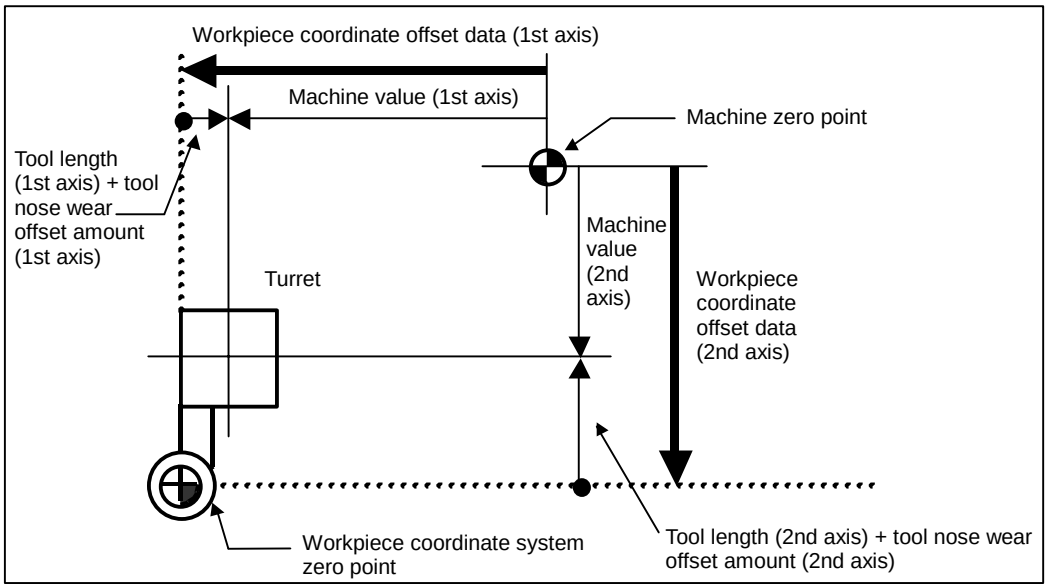
(Example) To calculate X axis (1st axis) in G55 workpiece coordinate system
(Step after (d) Selecting the measurement workpiece coordinate and axis.)

3) Press the SHIFT INPUT keys.



The automatically calculated workpiece coordinate offset data will be displayed at the cursor position (X axis).

1) Details of automatic calculation expression
The workpiece coordinate offset data is calculated with the following expression.
Workpiece coordinate offset data = machine value – (tool length data + tool nose wear data)



2) Offset number for selected tool
The number set in the following R register is used as the offset number of the tool length and tool nose wear data used for the automatic calculation.

#1098 TLno.	#1130 set_t	#1218 aux02 /bit4	Tool length offset number	Tool nose wear offset number
0	0/1	0/1	$1: R192,R193 $2: R392,R393	$1: R192,R193 $2: R392,R393
1	0	0	$1: R36,R37 $2: R236,R237	
		1	$1: R194,R195 $2: R394,R395	
	1	0/1		

(Note 1) The offset amount will be calculated as "0" when the offset number is 0.
(Note 2) If the offset number exceeds the specified number of offset sets, the error "#76 TOOL No. ERROR" will occur.
(Note 3) The tool nose wear offset amount will be calculated as "0" when "#1226 aux10/ bit0" is set to 1.

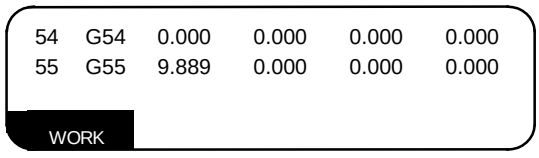
3) Calculating the workpiece coordinate offset data for an additional axis
The workpiece coordinate offset data for the 5th to 8th axes is calculated with the tool length data/tool nose wear data set to "0". As a result, the value will be the same as the machine value. The workpiece coordinate offset data for the 3rd and 4th axes follows the value set in "#1520 Tchg34".

#1520 Tchg34	Workpiece coordinate offset data for 3rd axis	Workpiece coordinate offset data for 4th axis
0	Calculated using the machine value, tool length and wear offset data.	Same as 4th axis' machine value.
1	Same as 3rd axis' machine value.	Calculated using the machine value, tool length and wear offset data.

(f) Setting the data
If the calculation results displayed in the setting area are correct, press the [INPUT] key and set the data.

(Example) To set the G55 workpiece coordinate system's X axis (1st axis)
(Step after (e) Automatically calculating the workpiece coordinate system offset data)

4) Press the [INPUT] key. ☞



The data is set in G55 X axis.

(3) Precautions
(a) If data is set at the cursor position, it will be overwritten with the calculated value when the [SHIFT][INPUT] keys are pressed.
(b) This data cannot be set while the program is running.

4.1.5 Workpiece Coordinate Offset Measurement Function (M System)

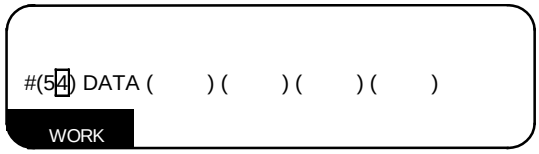
(1) Outline

The current machine position is displayed in the setting area when the SHIFT INPUT keys (CALC key) are pressed.

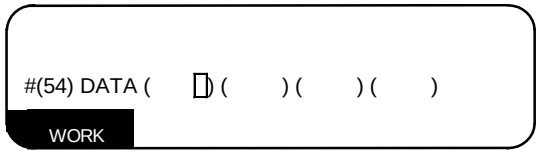
(2) Operation procedures

(Example) To measure the X axis (1st axis) of the G54 workpiece coordinate system.

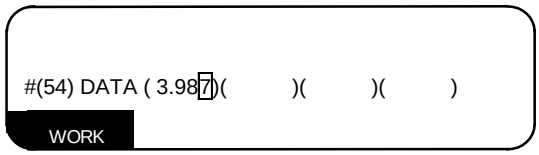
1) Set 54 in # ( ).



2) Move the cursor to the X axis setting area.

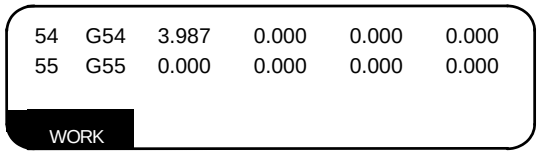


3) Press the SHIFT INPUT keys.



The machine position is displayed at the cursor position (X axis).

4) Press the INPUT key.



The data is set in G54 X axis.

(3) Precautions

(a) The workpiece coordinate offset measurement function will not activate while the tool measurement mode signal is ON (while the TLM switch is ON.) (The SHIFT INPUT keys will be ignored.)

(b) If data is set at the cursor position, it will be overwritten with the calculated value when the SHIFT INPUT keys are pressed.

(c) This data cannot be set while the program is running.

4.1.6 Workpiece position Measurement Function (M System)

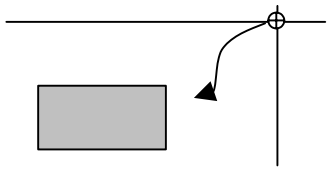
The workpiece position measurement function is used to measure each axis' coordinate point by the sensor installed on the spindle contacting the workpiece with the manual feed or handle feed.
The surface, hole center and width center coordinates are calculated from the measured coordinates, and those calculated results are set in the workpiece coordinate offset.
Only 1st part system is available for the workpiece position measurement.

(1) Surface workpiece offset measurement procedure

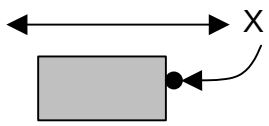
1) Perform an operation such as a reference point return to position the tool on the reference point.

2) Turn ON the measurement switch on the machine operation board. ☞ The message " WLM " appears.

3) Move the sensor near the workpiece using manual feed and manual handle feed. ☞



4) Put the sensor in contact with the workpiece in the X-axis direction. ☞



Automatic re-contact movements are performed by the axis at the time of contact.
The measurement coordinate value of the moved axis is displayed to the setting column.

#(□)(123.456)( )( )

5) Set the setting number of workpiece coordinate system, then press the INPUT key. ☞
#(54)(123.456)( )( )

The value that subtracted external workpiece offset value from the measurement value is set to the X-axis specified workpiece coordinate system offset.
The setting column will change to blanks.

6) Carry out the operations in steps 3) to 5) in the same way for the Y-axis and Z-axis.

7) Return the sensor to the reference point, and turn OFF the measurement switch.

☞ The message " WLM " disappears.

(2) Hole center workpiece offset measurement procedure

1) Perform an operation such as a reference point return to position the tool on the reference point.

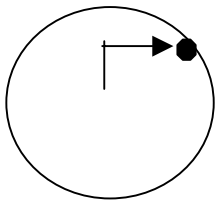
2) Turn ON the measurement switch on the machine operation board.

☞ The message " WLM " appears.

3) Move the sensor into the hole using manual feed and manual handle feed.

4) Put the sensor in contact with the inner walls of the hole.
Only one axis performs contact to the workpiece.

☞



Automatic re-contact movements are performed by the axis that did contact.
The measurement coordinate value of the moved axis is displayed to the setting column.

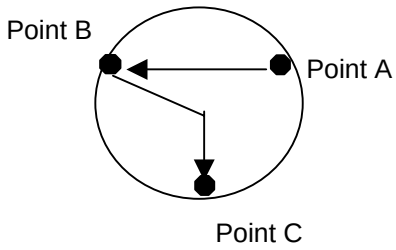
#( )(123.45)( )( )

5) Set the contact position data (measurement coordinate) as point A.

#(1) INPUT

☞ The measurement coordinate value is set to point A (X, Y).
The setting column is updated to #(2).
The measurement A and data of movement axis are highlighted.
The setting column will change to blanks.
#1 TLM P. A 12.345 45.678

6) Measure points B and C in the same way, and set them.



(Note) Width center workpiece offset measurement is performed when only points A and B are set.
Perform the measurement by one axis.

7) Set the setting number of workpiece coordinate system, then press the INPUT key.
#(54)( )( )( )

The hole center is calculated from points A, B and C.
The value that subtracted external workpiece offset value from the calculated hole center value is set.
The setting column highlight returns to normal and the value is cleared to 0.

8) Return the sensor to the reference point, and turn OFF the measurement switch.

The message " WLM " disappears.

(3) Width center workpiece offset measurement procedure

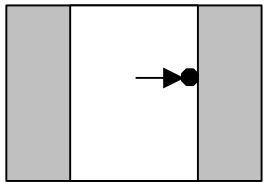
1) Perform an operation such as a reference point return to position the tool on the reference point.

2) Turn ON the measurement switch on the machine operation board.

The message " WLM " appears.

3) Move the sensor into the groove using manual feed and manual handle feed.

4) Put the sensor in contact with the inner walls of the groove.
Only one axis performs contact to the workpiece.



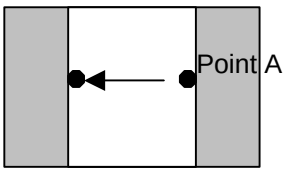
Automatic re-contact movements are performed by the axis at the time of contact.
The measurement coordinate value of the moved axis is displayed to the setting column.

#( )(10.567)( )( )

5) Set the contact position data (measurement coordinate) as point A.

The measurement coordinate value is set to #1 point A (X, Y).
The setting column is updated to #(2).
TLM P. A and data of movement axis are highlighted.
The setting column will change to blanks.
#1 TLM P. A 10.567 5.678

6) Put the sensor in contact with the opposite side of the groove.



7) Set the contact position data (measurement coordinate) as point B.

The measurement coordinate value is set to #2 point B (X, Y).
TLM P. B and data of movement axis are highlighted.

(Note) Hole center workpiece offset measurement is performed when point C is set to data.

8) Set the setting number of workpiece coordinate system, then press the INPUT key.
#(54)( )( )( )

The width center is calculated from points A and B.
The value that subtracted external workpiece offset value from the calculated width center value is set.
The setting column highlight returns to normal and the value is cleared to 0.

9) Return the sensor to the reference point, and turn OFF the measurement switch. ☞ The message " WLM " disappears.

(4) Subtracting external workpiece offset when workpiece coordinate offset is set

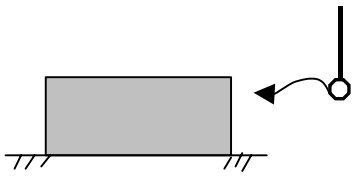
When workpiece coordinate G54 to G59, offset of expend workpiece coordinate offset P101 to P148 are set ([INPUT]) in the surface, hole center and width center workpiece offset measurement, the coordinate value that subtracted external workpiece offset value from the measurement position coordinate (or hole, width center coordinate) is set.

When “External workpiece offset” (#1237 set09/bit0) is OFF
The workpiece coordinate offset = The measurement (center) coordinate value - The external workpiece offset
When “External work offset” (#1237 set09/bit0) is ON
The workpiece coordinate offset = The measurement (center) coordinate value + The external workpiece offset

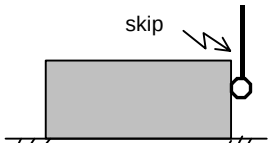
(5) Automatic re-contact movements at the time of contact

When the measurement is performed during jog mode or handle mode, perform following operation after contact.

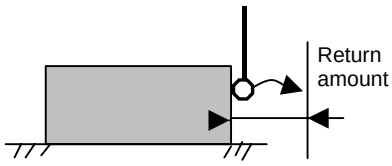
1) Move to the measurement point in jog mode. ☞



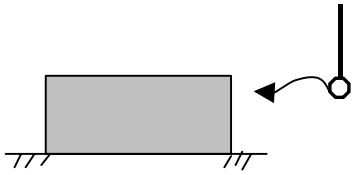
2) Stop on the measurement point. ☞



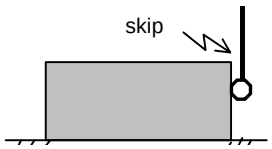
3) Return by the return amount of measurement parameter (#8705). ☞



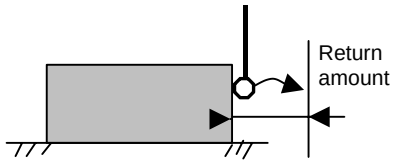
4) Move to the measurement point again by feed rate of measurement parameter (#8706). ☞



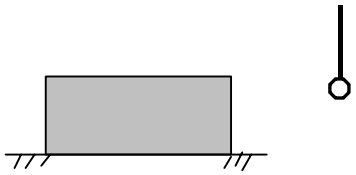
5) Stop on the measurement point. (Readout of skip coordinates) ☞



6) Return by the return amount of measurement parameter (#8705). ☞



7) Complete the measurement. Next measurement can be performed. ☞



The speed for returning of 3), 6) are 40 times higher than the feed rate of measurement parameter (#8706). However, when the speed for returning is higher than rapid traverse feed rate (override 100%), the rapid traverse feed rate of parameter (override 100%) will be applied.

(6) Restrictions

a) Measurement points A, B and C are unhighlighted and reset to 0 (not set) when the measurement switch is turned ON/OFF, setting to the workpiece coordinate offset ends, and when the reset key is pressed.
b) Only 1st system axes can be used as measurement target axes. The measurement target axis corresponds to base-axis (#1026 base_I, #1027 base_J, #1028 base_K).
c) An error will not occur during the measurement, even if there is the axis movement other than that of the base-axis.
d) The skip machine position and measurement points A, B and C are held even when the screen changes during measurement.
e) An axis other than the measurement axis cannot be moved during the automatic re-contact movement.
f) When the return amount of measurement parameter or feed rate is 0, first contact point is measurement point, so automatic re-contact movement is not performed.
g) Perform the measurement during the movement by moving only one axis.
h) The settings of the workpiece coordinate offset of surface, hole center and width center measurement during measurement are set in absolute value mode.
i) If rotary axis is set to measurement axis, the hole center / width center cannot be calculated properly by the angle of the rotation axis.

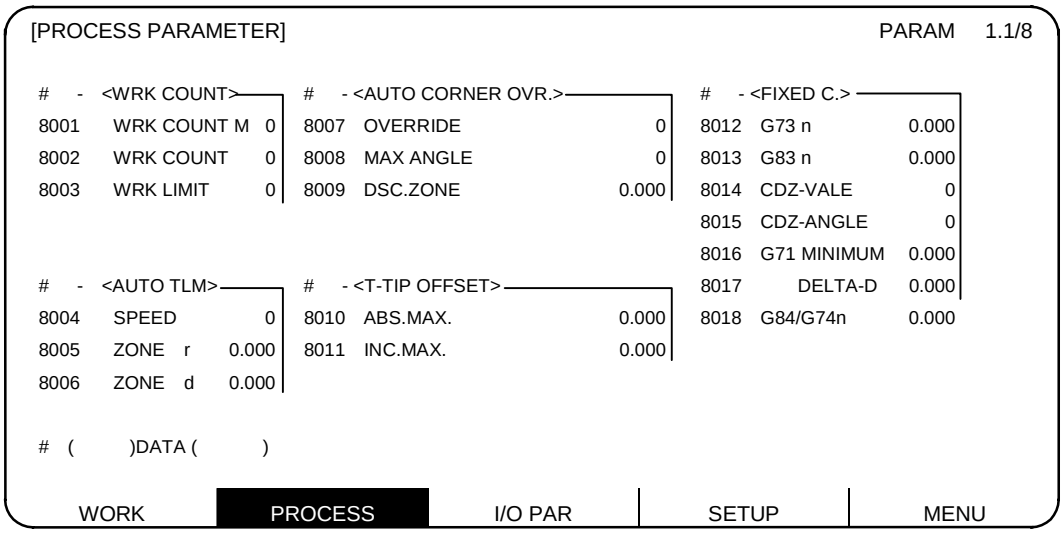
4.2 Machining Parameters

Pressing the menu key ⟩PROCESS⟨ displays the PROCESS PARAMERTER screen.
The number of digits in the decimal section of the parameters related to length is determined by the input setting unit.
The input setting unit is set with parameter "#1003 iunit".

Input setting unit	No. of digits in decimal section	Example of setting range
B	3	0 to 999.999 (mm)
C	4	0 to 99.9999 (mm)
D	5	0 to 9.99999 (mm)

The setting ranges indicated in this manual use the input setting unit "B".

4.2.1 PROCESS PARAMERTER



<WRK COUNT> (No. of workpieces machined)

#	Item	Contents	Setup 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 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)

#	Item	Contents	Setup range (unit)
8004	SPEED	Set the feedrate during automatic tool length measurement.	1 to 60000 (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 tolerable zone of the measurement position. If the sensor signal turns ON in front of d before the measurement position, or if the signal does not turn ON after d is passed, an alarm will occur.	0 to 99999.999 (mm)

<AUTO CORNER OVR> (Automatic corner override)

#	Item	Contents	Setup range (unit)
8007	OVERRIDE	Set the override value for automatic corner override.	0 to 100 (%)
8008	MAX ANGLE	Set the max. corner opening angle where deceleration should start automatically. If the angle is larger than this value, deceleration will not start.	0 to 180 (°)
8009	DSC. ZONE	Set the position where deceleration starts at the corner. Designate at which length point before the corner deceleration should start.	0 to 99999.999 (mm)

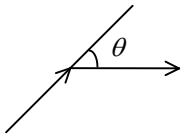
<T-TIP OFFSET> (Wear data input)

#	Item	Contents	Setup range (unit)
8010	ABS. MAX. (For L system only)	Set the max. value when inputting the tool wear compensation amount. A value exceeding this setting value cannot be set.	0 to 999.999 (mm)
8011	INC. MAX. (For L system only)	Set the max. value for when inputting the tool wear compensation amount in the addition mode.	0 to 999.999 (mm)

<FIXED C.> (Fixed cycle)

#	Item	Contents	Setup range (unit)
8012	G73 n (For M system only)	Set the return amount for G73 (step cycle).	0 to 99999.999 (mm)
8013	G83 n	Set the return amount for G83 (deep hole drilling cycle).	0 to 99999.999 (mm)
8014	CDZ-VALE (For L system only)	Set the screw cut up amount for G76, G78 (thread cutting cycle).	0 to 127 (0.1 lead)
8015	CDZ-ANGLE (For L system only)	Set the screw cut up angle for G76, G78 (thread cutting cycle).	0 to 89 (°)
8016	G71 MINIMUM (For L system only)	Set the minimum cut amount for the final cutting in G71, G72 (rough cutting cycle). If the final cutting amount is smaller than this value, the final cut will not be performed.	0 to 999.999 (mm)
8017	DELTA-D (For L system only)	Set the change amount to the command cut amount D for G71, G72 (rough cutting cycle). Each cut amount will be the value obtained by adding or subtracting this value from command D, and thus, the amount can be changed each cut.	0 to 999.999 (mm)
8018	G84/G74 return (For M system only)	Set up return length m at a G84/G74 pecking tap cycle. Note: Set “0” to specify a usual tap cycle.	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. Indicates 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. Coefficient = 100 – setting value Note: This is valid when "#8021 COMP CHANGE" is set to "0".	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 degrees. The standard setting value is "0".	0 to 89 (degrees) 0: The angle will be 5°.
8021	COMP CHANGE	Select whether to share or separate the compensation coefficient at the corner/curve during the high-accuracy control mode. 0: Share (#8019 R COMP) 1: Separate • Corner (#8022 CORNER COMP) • Curve (#8023 CURVE COMP) Note: Set "1" when using SSS control.	0/1
8022	CORNER COMP	Set the compensation coefficient to further reduce or increase the roundness at the corner during the high-accuracy control mode. Coefficient = 100 – setting value Note: This is valid when "#8021 COMP CHANGE" is set to "1".	–1000 to 99 (%)
8023	CURVE COMP	Set the compensation coefficient to further reduce or increase the radius reduction amount at the curve (arc, involute, spline) during the high-accuracy control mode. Coefficient = 100 – setting value Note: This is valid when "#8021 COMP CHANGE" is set to "1".	–1000 to 99 (%)
8024	EDGE ANGLE	Not used.	–

<PRECISION> (High precision control)

#	Item	Contents	Setup range (unit)
8025	SPLINE ON (for M system only)	Specify whether to enable the fine spline function. 0: Disable the fine spline function. 1: Enable the fine spline function.	0/1
8026	CANCEL ANG. (for M system only)	When the angle made by blocks exceeds the set value, spline interpolation is canceled temporarily. In consideration of the pick feed, set a value a little smaller than the pick feed angle.	0 to 180° 0: 180°
8027	Toler-1 (for M system only)	Specify the maximum chord error in a block that includes an inflection point. Set the tolerance applicable when the applicable block is developed to fine segments by CAM. (normally about 10 μm) When 0.000 is set, the applicable block is linear.	At 1μm 0.000 to 100.000mm At 0.1μm 0.0000 to 10.0000mm
8028	Toler-2 (for M system only)	Specify the maximum chord error in a block that includes no inflection point. Set the tolerance applicable when the applicable block is developed to fine segments by CAM. (normally about 10 μm) When 0.000 is set, the applicable block is linear.	At 1μm 0.000 to 100.000mm At 0.1μm 0.0000 to 10.0000mm
8029	FairingL (for M system only)	Set the length of the block subject to fairing. (Valid when #8033 Fairing ON is set to 1.)	0 to 100.000mm
8030	MINUTE LENGTH (for M system only)	When the length of one block exceeds the set value, spline interpolation is canceled temporarily and linear interpolation is performed. Set a value a little smaller than linear block length of the workpiece to be machined. If - 1 is set, spline interpolation is performed regardless of block length.	–1 to 127mm 0: 1mm
8033	Fairing ON (for M system only)	Set whether to use the fairing function. 0: Fairing invalid 1: Fairing valid	0/1
8034	AccClamp ON (for M system only)	Set the method for clamping the cutting speed. 0: Clamp with parameter “#2002 clamp” or the corner deceleration function. 1: Clamp the cutting speed with acceleration judgment. (Valid when #8033 Fairing ON is set to 1.)	0/1
8035	AccClampMag	Not used.	
8036	CordecJudge (for M system only)	Change the conditions for judging a corner. 0: Judge the corner from the angle of the neighboring block. 1: Judge the corner from the angle of the neighboring block, excluding minute blocks. (Valid when #8033 Fairing ON is set to 1.)	0/1
8037	CorJudgeL (for M system only)	Set the length of the block to be excluded. (Valid when #8036 CordecJudge is set to 1.)	0 to 99999.999 (mm)

<Normal line control>

#	Item	Contents	Setup range (unit)
8041	C-rot.R	This is valid with normal line control type II. Set the length from the center of the normal line control axis to the end of the tool. This is used to calculate the turning speed at the block joint.	0.000 to 99999.999 (mm)
8042	C-ins.R	This is valid with normal line control type I. Set the radius of the arc to be automatically inserted into the corner during normal line control.	0.000 to 99999.999 (mm)

<Fixed cycle>

#	Item	Contents	Setup range (unit)
8051	G71 THICK	Set the amount of cut-in by the rough cutting cycle (G71, G72)	0 to 99999.999 (mm)
8052	PULL UP	Set the amount of recess after cutting by the rough cutting cycle (G71, G72).	0 to 99999.999 (mm)
8053	G73 U	Set the X-axis cutting margin of the forming rough cutting cycle (G73).	–99999.999 to 99999.999 (mm)
8054	W	Set the Z-axis cutting margin of the forming rough cutting cycle (G73).	–99999.999 to 99999.999 (mm)
8055	R	Set the number of times cutting is performed by the forming rough cutting cycle (G73).	0 to 99999 (times)
8056	G74 RETRACT	Set the amount of retract (amount of cut-up) of the push-cut cycle (G74, G75).	0 to 999.999 (mm)
8057	G76 LAST-D	Set the amount of final cut-in by the composite threading cycle (G76).	0 to 999.999 (mm)
8058	TIMES	Set the number of times the amount of final cut-in (G76 finish margin) is divided in the composite threading cycle (G76).	0 to 99 (times)
8059	ANGLE	Set the angle (thread angle) of the tool nose in the composite threading cycle (G76).	0 to 99 (°)

<Three-dimensional cutter compensation>

#	Item	Contents	Setup range (unit)
8071	3-D CMP (for M system only)	Value of p in the following denominator constants for three-dimensional tool radius compensation Vx = i x r/p, Vy = j x r/p, Vz = k x r/p Vx, Vy, Vz: X, Y, and Z axes or vectors of horizontal axes i, j, k: Program command value r: Offset p = $\sqrt{i^2 + j^2 + k^2}$ when the set value is 0.	0 to 99999.999

<Scale factor>

#	Item	Contents	Setup range (unit)
8072	SCALING P (for M system only)	Set the scale factor for reduction or magnification for the machining program for which the G50 or G51 command is issued. This parameter is effective when the program specifies no scale factor.	0 to 99.999999

<Tool ID>

#	Item	Contents	Setup range (unit)
8073	OfsetPosition (for M system only)	Set the tool offset memory number position for writing the tool information data's tool length offset amount, tool radius compensation amount, tool length wear amount and tool radius wear amount into the tool offset data. Note: If 0 or a value exceeding the number of tool compensation sets is set, the data will not be written into the tool offset data.	0 to 999
8074	IDMacroTop (for M system only)	Set the head position when writing the tool information data's user areas 4 to 9 in the macro variables.	0 to 999

<Spiral interpolation>

#	Item	Contents	Setup range (unit)
8075	SpiralEndErr (for M system only)	Designate the tolerable error range (absolute value) when the end point position commanded with the command format type 2 spiral interpolation or conical interpolation command differs from the end point position obtained from the speed and increment/ decrement amount.	0 to 99999.999 (mm)
8076	SpiralMinRad (for M system only)	Not used.	

<Involute interpolation>

#	Item	Contents	Setup range (unit)
8077	InvoluteErr (for M system only)	Set the tolerable error value of the involute curve that passes through the start point and the involute curve that passes through the end point during involute interpolation.	0 to 99999.999 (mm)

<Screen saver>

#	Item	Contents	Setup range (unit)
8078	Screen Saver	Set the time to turn the screen OFF. The screen saver will not turn ON if 0 is set. (Note) This parameter setting is valid only for the LCD display unit.	0 to 60 (min) 0: Do not turn screen OFF.

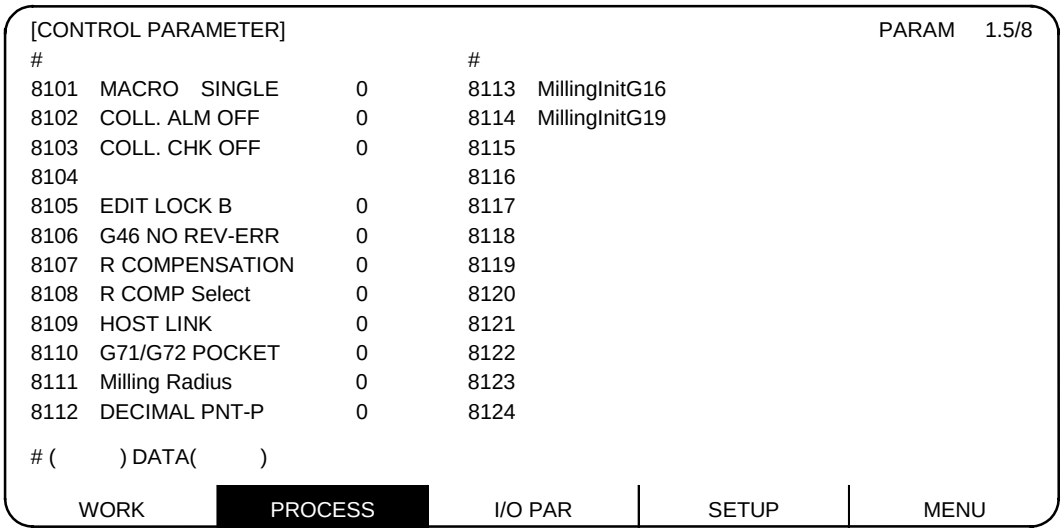
<Deep hole drilling cycle>

#	Item	Contents	Setup range (unit)
8083	G83S modeM (for M system only)	Set the M command code for changing to the small diameter deep hole drilling cycle mode.	1 to 99999999
8084	G83S Clearanse (for M system only)	Set the clearance amount for the G83 small diameter deep hole drilling cycle.	0 to 999.999 (mm)
8085	G83S Forward F (for M system only)	Set the feedrate from the R point to the cutting start position in the G83 small diameter deep hole drilling cycle.	0 to 99999 (mm/min)
8086	G83S Back F (for M system only)	Set the speed for returning from the hole base during the G83 small diameter deep hole drilling cycle.	0 to 99999 (mm/min)

<SSS control>

#	Item	Contents	Setup range (unit)
8090	SSS ON (for M system only)	Set whether to validate SSS control with G05 P10000. 0: Invalid 1: Valid	0/1
8091	StdLength (for M system only)	Adjust the maximum value of the range for recognizing the shape. To eliminate the effect of steps or errors, etc., set a large value. To enable sufficient deceleration, set a small value. If "0.000" is set, the standard value (1.000mm) will be applied.	0 to 100.000 (mm)
8092	ClampCoeff (for M system only)	Adjust the clamp speed at the curved section configured of fine segments. Coefficient = $\sqrt{\ }$ setting value	1 to 100
8093	StepLeng (for M system only)	Set the width of the step at which the speed is not to be decelerated. (Approximately the same as the CAM path difference [Tolerance].) If 0 is set, the standard value (5μm) will be applied. If a minus value is set, the speed will decelerate at all minute steps.	-0.001 to 0.100 (mm)
8094	DccWaitAdd (for M system only)	Set the time to wait for deceleration when the speed FB does not drop to the clamp speed.	0 to 100 (ms)
8095	Tolerance (for M system only)	Set the tolerable error when the error between the command path and tool path is large. The error will decrease when a small value is set, but the machining time will increase. If "0.000" is set, the error will not be adjusted.	0 to 100.000 (mm)

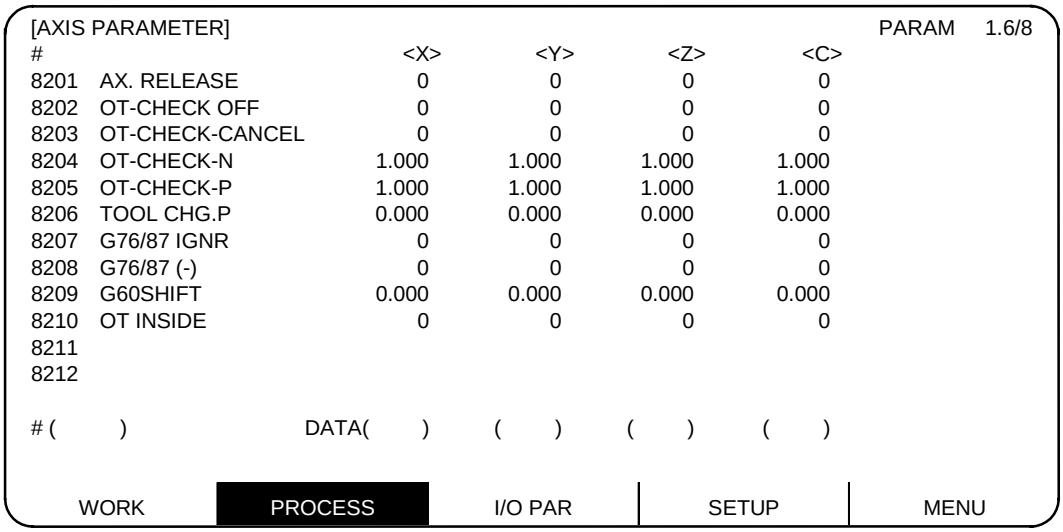
4.2.2 Control Parameters



#	Item	Contents	Setup 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 compensation. 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 compensation. 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 the compensation direction reversal in G46 (nose R compensation). 0: An alarm is output and operation stops when the compensation direction is reversed (G41 $\rightarrow$ G42, G42 $\rightarrow$ G41). 1: An alarm does not occur when the compensation direction is reversed, and the current compensation direction is maintained.	0/1

<table>
<tr><th>#</th><th>Item</th><th>Contents</th><th>Setup range (unit)</th></tr>
<tr><td>8107</td><td>R COMPENSATION</td><td>0: In arc cutting mode, the machine moves to the inside because of a delay in servo response to a command, making the arc smaller than the command value.
1: In arc cutting mode, the machine compensates the movement to the inside because of a delay in servo response to a command</td><td>0/1</td></tr>
<tr><td>8108</td><td>R COMP Select</td><td>Specify whether to perform arc radius error correction over all axes or axis by axis.
0: Perform correction over all axes.
1: Perform correction over axis by axis.
Note: This parameter is effective only when #8107 R COMPENSATION is 1.</td><td>0/1</td></tr>
<tr><td>8109</td><td>HOST LINK</td><td>Specify whether to enable computer link B instead of the RS-232C port.
0: Disable computer link B to enable normal RS-232C communication.
1: Enable computer link B to disable normal RS-232C communication.</td><td>0/1</td></tr>
<tr><td>8110</td><td>G71/G72 POCKET</td><td>Set the pocket machining if there is a dimple (pocket) in the rough cutting cycle (G71, G72) finishing program.
0: Pocket machining OFF
1: Pocket machining ON</td><td>0/1</td></tr>
<tr><td>8111</td><td>Milling Radius</td><td>Select the diameter and radius of the linear axis for milling (cylindrical/pole coordinate) interpolation.
0: All axes radius command
1: Each axis setting (follows #1019 dia diameter designated axis)
Note: This parameter is valid only in the milling (cylindrical/pole coordinate) interpolation mode.</td><td>0/1</td></tr>
<tr><td>8112</td><td>DECIMAL PNT-P</td><td>0: The decimal point command for G04 address P is invalidated.
1: The decimal point command for G04 address P is validated.</td><td>0/1</td></tr>
<tr><td>8113</td><td>MillingInitG16</td><td rowspan="2">Designate which plane to use for milling machining after the power is turned ON or reset.
<table><tr><th>#8113</th><th>#8114</th><th>Plane</th></tr><tr><td>0</td><td>0</td><td>G17 plane</td></tr><tr><td>0</td><td>1</td><td>G19 plane</td></tr><tr><td>1</td><td>0</td><td rowspan="2">G16 plane</td></tr><tr><td>1</td><td>1</td></tr></table>Note: This parameter is valid for the G code system 2, 3 (#1037 cmdtyp=3, 4).</td><td>0: Plane other than G16
1: Select G16 plane</td></tr>
<tr><td>8114</td><td>MillingInitG19</td><td>0: Plane other than G19
1: Select G19 plane</td></tr>
</table>

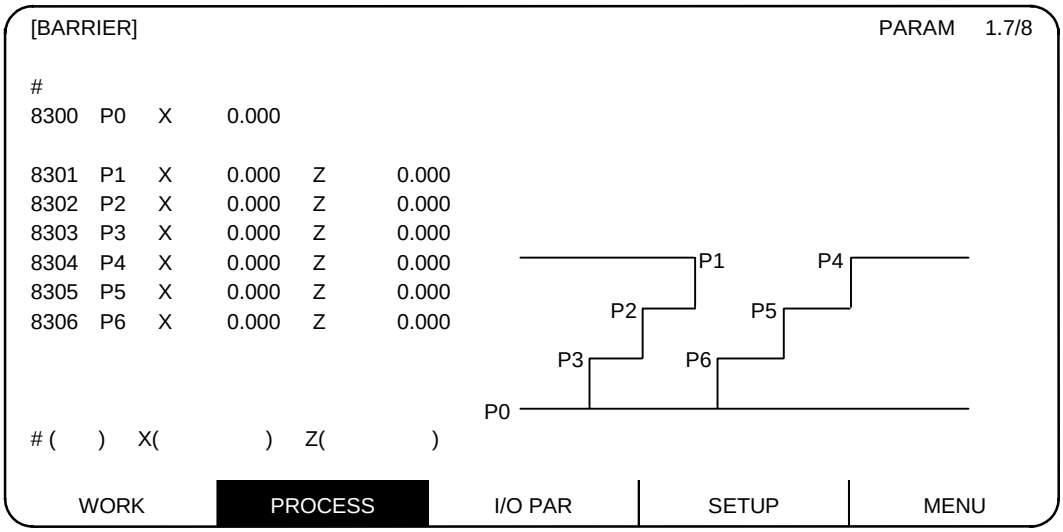
4.2.3 Axis Parameters



#	Item	Contents	Setup 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) is selected, the stored stroke limits I, II (or IIB) and IB will be invalid until the first zero point return is executed after the power is turned ON. 0: Stored stroke limit II valid (according to #8202) 1: Stored stroke limit II invalid Note: Temporary cancel of #8203 soft limit affects all the stored stroke limits.	0/1
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 (mm)

#	Item	Contents	Setup range (unit)
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 (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 (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 last positioning direction and distance for a G60 (uni-directional positioning) command.	±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 (select stored stroke limit II.) 1: Inhibits inside area (select stored stroke limit II B.)	0/1
8211	MIRR. IMAGE	Enable or disable the parameter mirror image function. 0: Disable 1: Enable	0/1

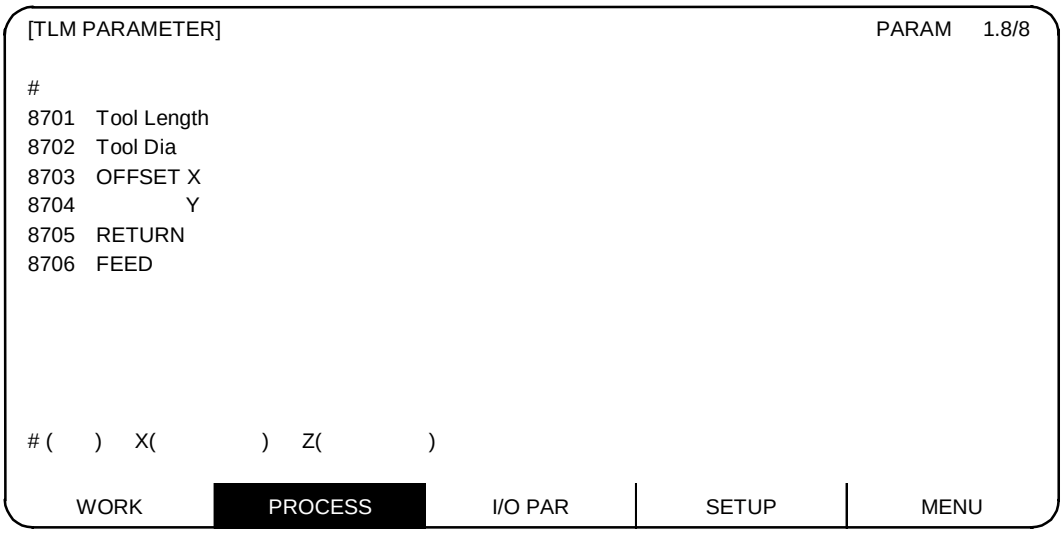
4.2.4 Barrier Data



#	Item	Contents	Setup range (unit)
8300	PO (For L system only)	Set the reference X-coordinates of the chuck and the tail stock barrier. Set the center coordinate (Radius value) of workpiece by the basic machine coordinate system.	±99999.999 (mm)
8301 8302 8303 8304 8305 8306	P1 P2 P3 P4 P5 P6 (For L system only)	Set the area of the chuck and tail stock barrier. (Radius value) Set the coordinate value from the center of workpiece for X-axis. Set the coordinate value by basic machine coordinate system for Z-axis.	±99999.999 (mm)
8310	Barrier ON (For L system only)	Select the validity of the chuck and tailstock barrier. 0: Invalid (Setting from special display unit valid) 1: Valid	0/1
8311 8312	P7 P8 (For L system only)	Set the area of the left spindle section. • X axis: Set the coordinate value from the workpiece center (P0). (radius value) • Z axis: Set the coordinates in the basic machine coordinate system.	±99999.999 (mm)
8313 8314	P9 P10 (For L system only)	Set the area of the right spindle section. • X axis: Set the coordinate value from the workpiece center (P0). (radius value) • Z axis: Set the coordinates in the basic machine coordinate system.	±99999.999 (mm)
8315	BARRIER TYPE (L) (For L system only)	Set the shape of the left chuck and tailstock barrier. 0: No area 1: Chuck 2: Tailstock	0/1/2

#	Item	Contents	Setup range (unit)
8316	BARRIER TYPE I (For L system only)	Set the shape of the right chuck and tailstock barrier. 0: No area 1: Chuck 2: Tailstock	0/1/2
8317	DELIV. AX. NAME (For L system only)	When the right chuck and tailstock barrier is movable, set the name of the delivery axis. When using the 2-system method and the delivery axis is an axis in the other system, designate the system as 1A, 1B or 2A, 2B. If the system is not designated as A and B, the set system will be used.	A/B/.. (axis address) 1A/1B/.. 2A/2B/.. (system designation) 0 (cancel)
8318	STOCK ANGLE (L) (For L system only)	Set the angle for the left tailstock end section. The angle will be interpreted as 90° if there is no setting (0).	0 to 180 (°) 0: 90° default
8319	STOCK ANGLE I (For L system only)	Set the angle for the right tailstock end section. The angle will be interpreted as 90° if there is no setting (0).	0 to 180 (°)

4.2.5 Tool Measurement Parameter



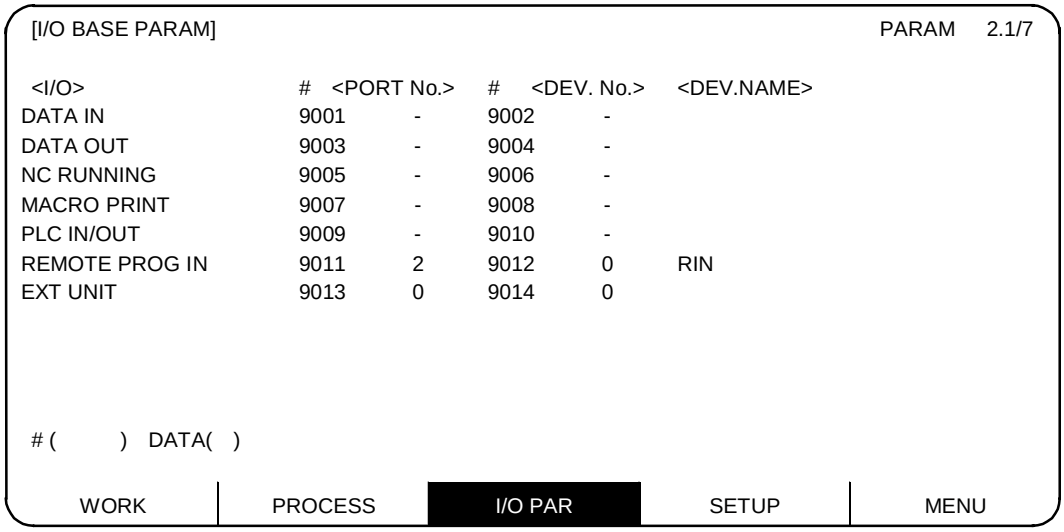
#	Item	Contents	Setup range (unit)
8701	Tool length	Set the length to the end of the touch tool.	±99999.999 (mm)
8702	Tool Dia	Set the spherical diameter of the touch tool end.	±99999.999 (mm)
8703	OFFSET X	Set the spindle center deviation amount from the touch tool center in the X axis direction.	±99999.999 (mm)
8704	Y	Set the spindle center deviation amount from the touch tool center in the Y axis direction.	±99999.999 (mm)
8705	RETURN	Set the return distance to contact the touch tool against the workpiece again.	±99999.999 (mm)
8706	FEED	Set the ederate when contacting the touch tool against the workpiece again.	1 to 60000 (mm/min)

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.

4.3.1 I/O BASE PARAM

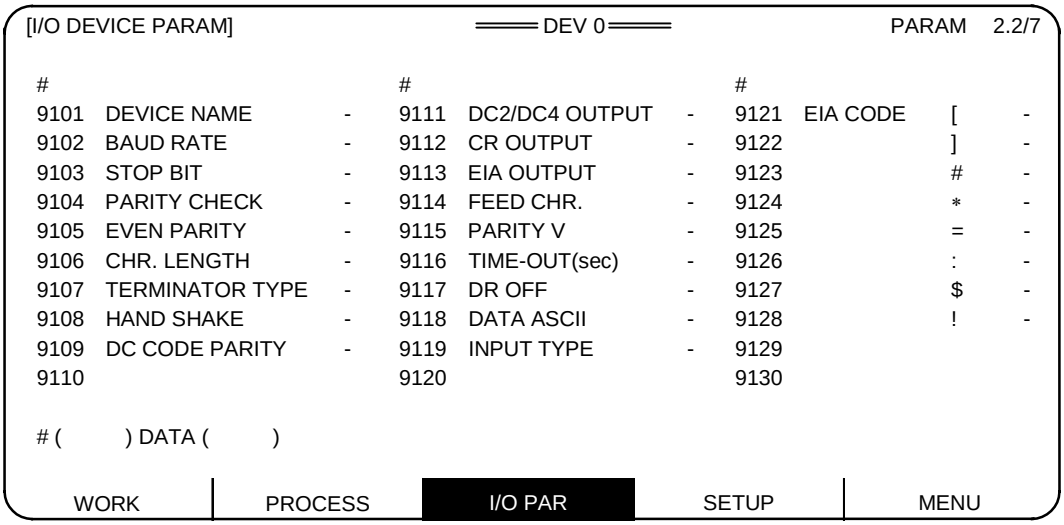


<I/O>	#	<PORT No.>	#	<DEV. No.> <DEV. NAME>
		Specify the board No. to which the serial input/output device is connected to 2. • Set “1” to use ch1. • Set “2” to use ch2.		Set the input/output device No. for each application. The device Nos. are 0 to 4 and correspond to the input/output device parameters. 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.
TAPE MODE	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	Specify the port for inputting remote programs.	9012	Specify the number of the device used to input remote programs.
EXT UNIT	9013	Specify the port for communication with an external unit.	9014	Specify the number of the unit used for communication with an external unit

#	Item	Contents	Setup range (unit)
9015	PORT NO. (tool ID)	Set the number of the port connected with the tool ID. (Either ch1 or ch2 can be used.) • Set “1” to use ch1. • Set “2” to use ch2.	1/2 (M64)
9016	DEV. NO. (tool ID)	Set the number of the input/output device to be used. (Any device No. can be used.)	0 to 4 (M64)

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>.



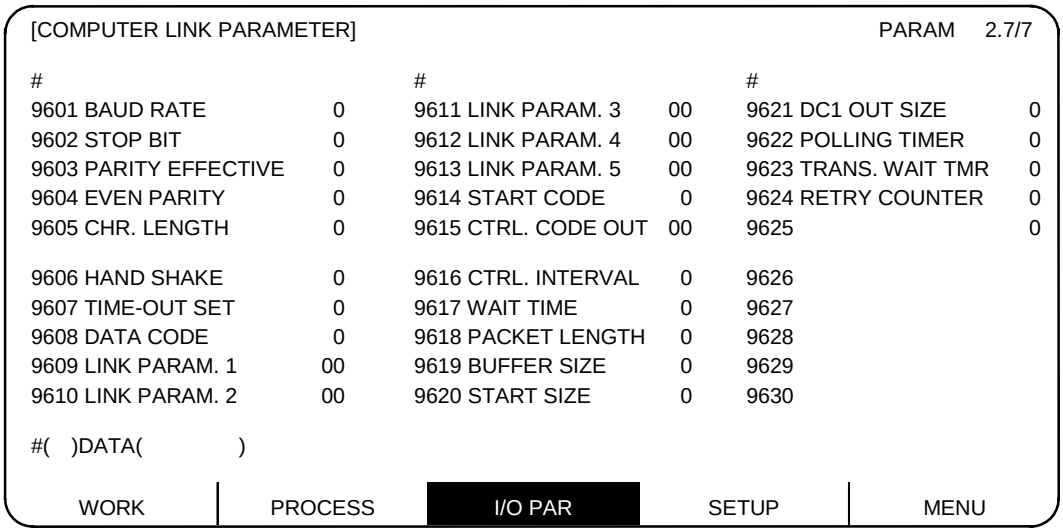
#	Item	Contents	Setup range (unit)
9101	DEVICE NAME	Set the device name corresponding to the device No. Set a simple name for quick identification.	Use alphabet characters, numerals and symbols to set a name within 3 characters.
9102	BAUD RATE	Set the serial communication speed.	0: 19200 (bps) 1: 9600 2: 4800 3: 2400 4: 1200 5: 600 6: 300 7: 150
9103	STOP BIT	Set the stop bit length used in the start-stop system.	1: 1 (bit) 2: 1.5 3: 2
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: 5 (bit) 1: 6 2: 7 3: 8
9107	TERMINATOR TYPE	The code to terminate data reading can be selected.	0 and 3: EOR 1 and 2: EOB or EOR

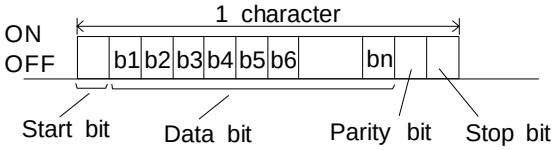
#	Item	Contents	Setup range (unit)
9108	HAND SHAKE	Specify the transmission control method. The method will be no procedure if a value except 1 to 3 is set.	1: RTS/CTS method 2: No procedure (No handshaking) 3: DC code method
9109	DC CODE PARITY	Specify the DC code when the DC code method is selected.	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	In data output mode, select the ISO or EIA code for data output. In data input mode, the ISO and EIA codes are identified automatically.	0: ISO code output 1: EIA code output
9114	FEED CHR.	Specify the length of the tape feed to be output at the start and end of the data during tape output.	0 to 999 (characters)
9115	PARITY V	Specify whether to check the parity of the No. of characters in block during data input. The No. of characters is factory-set so that the check is valid at all times.	0: Do not perform parity V check 1: Perform parity V check
9116	TIME-OUT	Set the time out time to detect an interruption in communication. Time out check will not be executed when set to 0 to 30 seconds.	0 to 30 (s)
9117	DR OFF	Specify whether to check the DR data at the data input/output.	0: DR valid 1: DR invalid
9118	DATA ASC II	0: Output in ISO/EIA code (Depends on whether #9113, #9213, #9313, #9413, or #9513 EIA output parameter is set up) 1: Output in ASC II code	0/1
9119	INPUT FORM	Specify the mode for input (collation). 0: Standard input (Data from the very first EOB is handled as significant information.) 1: EOBs following the first EOB of the input data are skipped until data other than EOB is input.	0/1

#	Item	Contents	Setup range (unit)
9121 9122 9123 9124 9125 9126 9127 9128	EIA CODE [ ] # * = : $!	When output with EIA code, data can be output using the alternate code in which the special ISO code not included in EIA is specified. Specify the codes which do not duplicate the existing EIA codes by hexadecimal for respective special codes.	0 to FF (hexadecimal)

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.	

4.3.3 COMPUTER LINK PARAMETER



#	Item	Contents	Setup range (unit)
9601	BAUD RATE	Specify the rate at which data is transferred.	0: 19200 (bps) 1: 9600 2: 4800 3: 2400 4: 1200 5: 600 6: 300 7: 110 8: 38400
9602	STOP BIT	Specify stop bit length used in start-stop mode. See “PARITY EFFECTIVE” in #9603. The number of characters is adjusted in output mode so that no problems occur if the parity check is enabled.	1: 1 2: 1.5 3: 2
9603	PARITY EFFECTIVE	This parameter is set when using a parity bit separately from the data bit.  Set this to match the input/output device specifications.	0: No parity bit used in I/O mode 1: Parity bit used in I/O mode
9604	EVEN PARITY	Specify whether even or odd parity is used when parity is used. This parameter is ignored when no parity is used.	0: Odd parity 1: Even parity
9605	CHR. LENGTH	Specify data bit length. See “PARITY EFFECTIVE” in #9603.	2: 7 3: 8

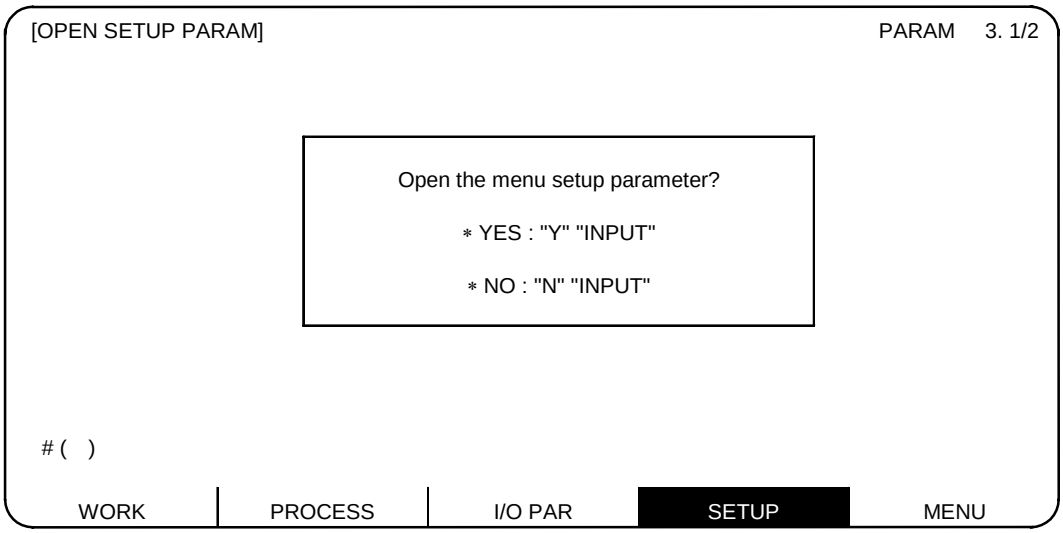
#	Item	Contents	Setup range (unit)
9606	HAND SHAKE	RS-232C transmission control mode DC control mode should be set for computer line B.	0: No control 1: RTS/CTS method 2: No handshaking 3: DC control mode
9607	TIME-OUT SET	Specify time-out time at which an interruption of data transfer during data input/output should be detected. If 0 is set, time infinity is specified.	0 to 999 (1/10s)
9608	DATA CODE	Specify the code to be used. See “PARITY EFFECTIVE” in #9603.	0: ASCII code 1: ISO code
9609	LINK PARAM. 1	Bit 1: DC1 output after NAK or SYN Specify whether to output the DC1 code after the NAK or SYN code is output. Bit 7: Enable/disable resetting Specify whether to enable resetting in the computer link.	0: Don't output the DC1 code. 1: Output the DC1 code. 0: Enable resetting in the computer link. 1: Disable resetting in the computer link
9610	LINK PARAM. 2	Bit 2: Specify the control code parity (even parity for the control code). Set the parity in accordance with the I/O device specifications. Bit 3: Parity V Specify whether to enable checking of parity V in one block in data input mode.	0: No control code parity added 1: Control code parity added 0: Disable 1: Enable
9611	LINK PARAM. 3	Not used	
9612	LINK PARAM.4	Not used	
9613	LINK PARAM.5	Not used	
9614	START CODE	Specify the code by which file data transfer begins at first. This parameter is used for a specific user, and set 0 in this parameter for normal operation.	0: DC1 (11H) 1: BEL (07H)

#	Item	Contents	Setup range (unit)
9615	CTRL. CODE OUT	Bit 0: NAK output Specify whether to send the NAK code to the host if a communication error occurs in computer link B. Bit 1: SYN output Specify whether to send the SYN code to the host if NC resetting or an emergency stop occurs in computer link B. Bit 3: DC3 output Specify whether to send the DC3 code to the host when communication ends in computer link B.	0: Do not output the NAK code. 1: Output the NAK code. 0: Do not output the SYN code. 1: Output the SYN code. 0: Do not output the DC3 code. 1: Output the DC3 code.
9615	CTRL. CODE OUT	Bit 0: NAK output Specify whether to send the NAK code to the host if a communication error occurs in computer link B. Bit 1: SYN output Specify whether to send the SYN code to the host if NC resetting or an emergency stop occurs in computer link B. Bit 3: DC3 output Specify whether to send the DC3 code to the host when communication ends in computer link B.	0: Do not output the NAK code. 1: Output the NAK code. 0: Do not output the SYN code. 1: Output the SYN code. 0: Do not output the DC3 code. 1: Output the DC3 code.
9616	CTRL. INTERVAL	Not used	
9617	WAIT TIME	Not used	
9618	PACKET LENGTH	Not used	
9619	BUFFER SIZE	Not used	
9620	START SIZE	Not used	
9621	DC1 OUT SIZE	Not used	
9622	POLLING TIMER	Not used	
9623	TRANS. WAIT TMR	Not used	
9624	RETRY COUNTER	Not used	

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.



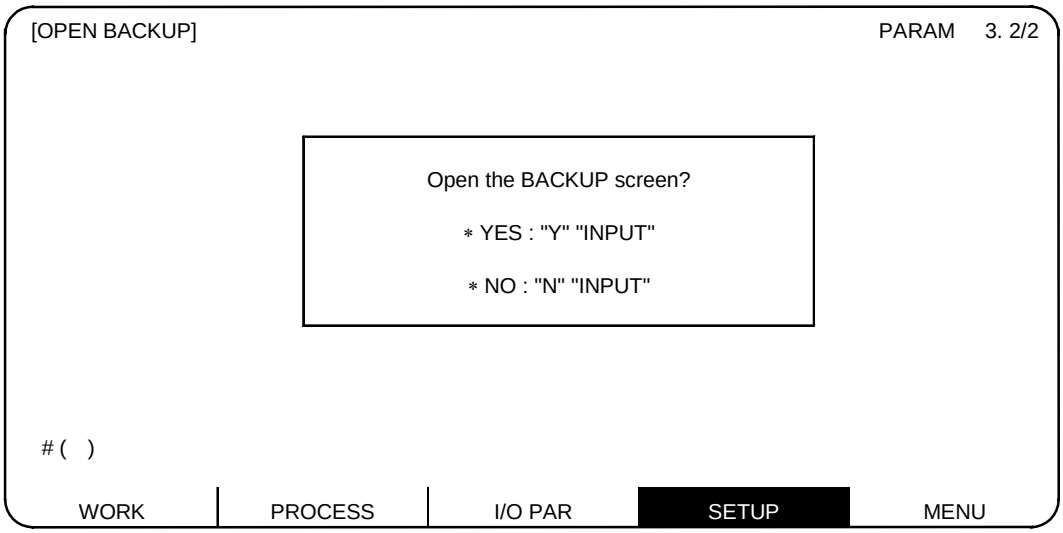
1) Select the setup parameter.
   Key-in "Y" in # ( ), and then press [INPUT] key.
   The normally hidden setup parameter menu will display when the menu changes over.
   The required menu can be selected to display and set the setup parameters.
2) Cancel the setup parameter selection.
   Key-in "N"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

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

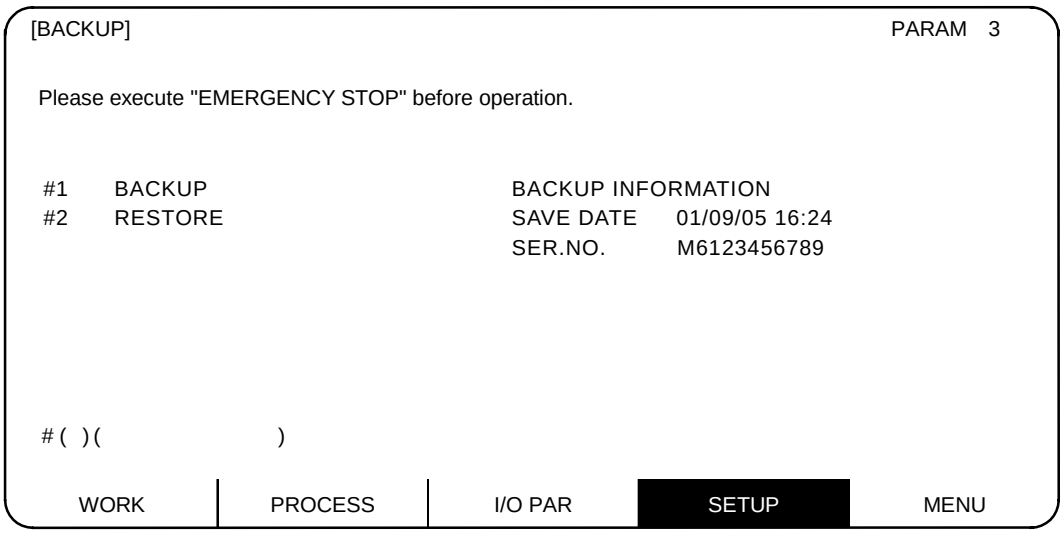
Refer to Alarm/Parameter Manual (BNP-B2201) for details on the setup parameters.
Always turn the power OFF after selecting the setup parameters.

4.5 BACKUP Screen

If the page key NEXT PAGE is pressed on the SETUP screen, the BACK UP screen will open.



Input "Y" in the setting area # ( ), and press the INPUT key. The BACKUP screen will open.



The date and time that the backup data was saved, and the serial No. are displayed on this screen.

4.5.1 Backup Operations

The parameters are backed up in the maintenance memory cassette with the following steps.

(1) Execute emergency stop.

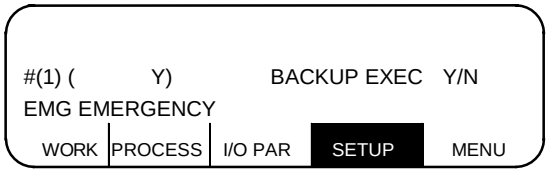
"EMG EMERGENCY" will appear at the operation message area.

(2) Input "1" at the setting area # ( ), and press the INPUT key.

"PLC STOP Y/N" will appear at the message area.

(3) Input "Y" at the setting area ( ), and press the INPUT key.

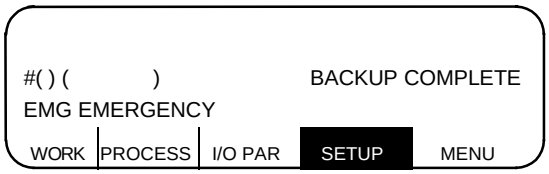
"BACKUP EXEC Y/N" will appear at the message area.



(4) Input "Y" at the setting area ( ), and press the INPUT key.

(a) The message "BACKUP EXECUTION" will appear while the parameters are being backed up.

(b) The message "BACKUP COMPLETE" will appear when backup is completed.



(Note 1) Other screens cannot be opened during this operation.
(Note 2) If the PLC is already stopped, step (3) is skipped.
The PLC will not start running even when the operations are completed.
(Note 3) If the INPUT key is pressed without inputting anything at the "~Y/N" display, the error "E01 SETTING ERROR" will occur.
(Note 4) Even if the PLC RUN/STOP changeover is prohibited (R2925/bit2 ON), the PLC will stop with step (3).

4.5.2 Restoration Operations

The parameters are restored from the maintenance memory cassette to the NC with the following steps.

(1) Execute emergency stop.

"EMG EMERGENCY" will appear at the operation message area.

(2) Input "2" at the setting area # ( ), and press the INPUT key.

"PLC STOP Y/N" will appear at the message area.

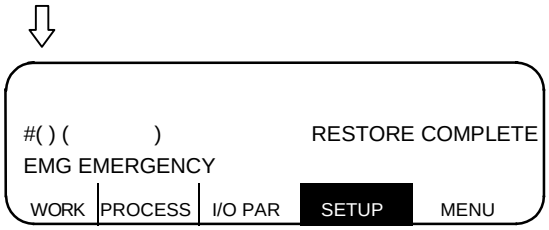
(3) Input "Y" at the setting area ( ), and press the INPUT key.

"RESTORE EXEC Y/N" will appear at the message area.

(4) Input "Y" at the setting area ( ), and press the INPUT key.

(a) The message "RESTORE EXECUTION" will appear while the parameters are being restored.

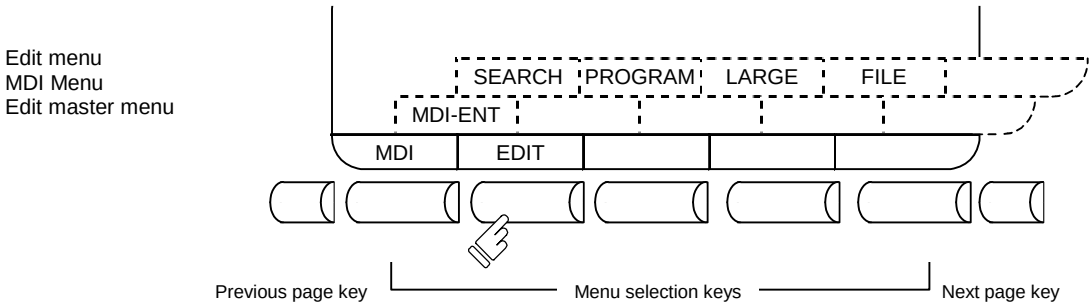
(b) The message "RESTORE COMPLETE" will appear when the restoration is completed.



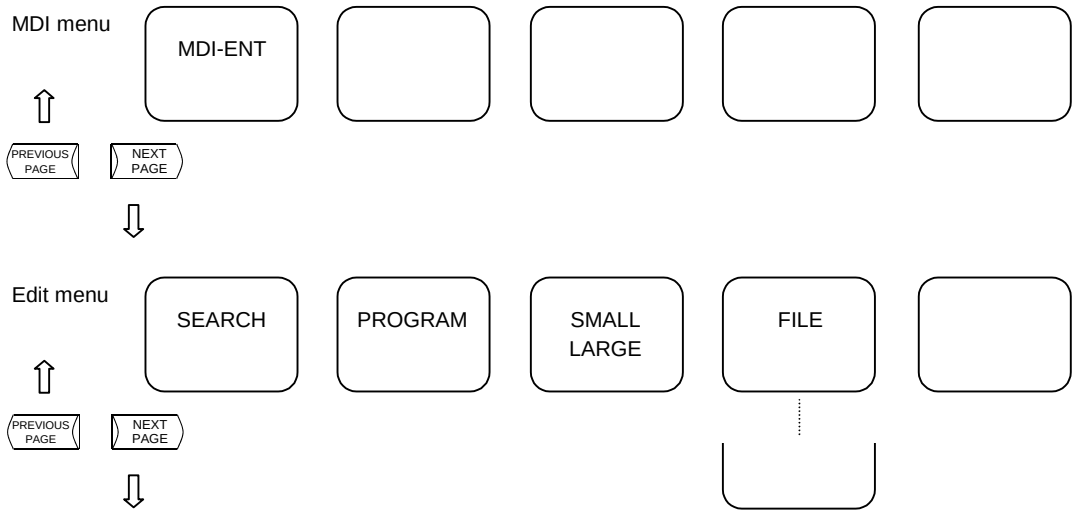
(Note 1) Other screens cannot be opened during this operation.
(Note 2) If the PLC is already stopped, step (3) is skipped.
The PLC will not start running even when the operations are completed.
(Note 3) If the INPUT key is pressed without inputting anything at the "~Y/N" display, the error "E01 SETTING ERROR" will occur.
(Note 4) Even if the PLC RUN/STOP changeover is prohibited (R2925/bit2 ON), the PLC will stop with step (3).

5. Program

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



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



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

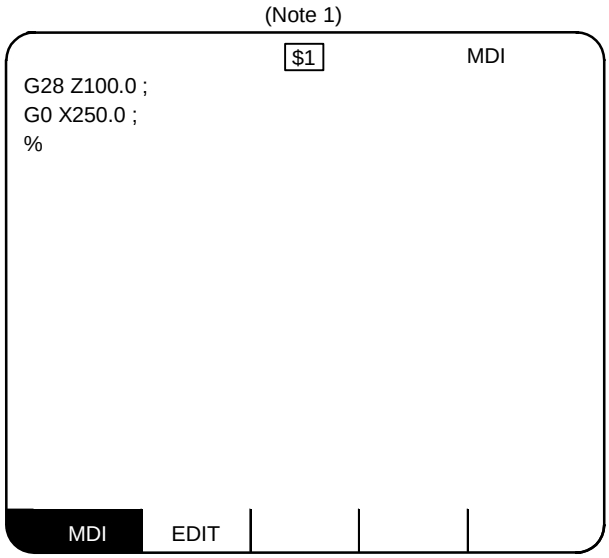
If a character string that matches a macro operator exists in the machining program (including a comment statement), it is automatically converted into the corresponding intermediate code during editing. This may cause a string different from that entered to be displayed during editing.

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

5.2 Menu Function

5.2.1 MDI Screen Menu Function

(1) Menu when [EDIT MDI] screen is selected



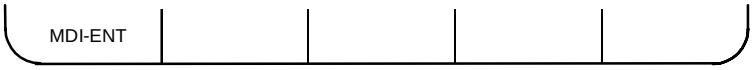
Menu	Function
MDI	Reverse display of MDI menu means that MDI screen is selected. MDI data can be set on the MDI screen.
Edit	Use this key to change the MDI screen to the EDIT screen.

(Note 1) When using the 2-part system, the system name of the currently selected system is displayed as $1 (system 1) and $2 (system 2). This is not displayed when using a 1-part system. (Only L system)

(Note 2) Whether to show or hide name of the selected system can be switched with the parameters.

#1050 MemPrg	Details
0, 2, 4, 6	The name of the selected system is not displayed.
1, 3, 5, 7	The name of the selected system is displayed.

(2) MDI screen extension operation menu



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

(Note 1) When using the 2-part system, the method for registering the MDI data as a machining program in the memory can be switched with the parameters.

#1050 MemPrg	#1285 ext21/bit0	Details
0, 4	-	The MDI data common for the systems is registered as a machining program common for the systems.
1, 5	OFF	The MDI data common for the systems is registered as a machining program for the selected system.
	ON	The MDI data common for the systems is registered as a machining program for the selected system. If the system is not selected, an empty (only EOR [%]) machining program is registered.
2, 6	-	The MDI data for the selected system is registered as a machining program common for the systems.
3, 7	OFF	The MDI data for the selected system is registered as a machining program for the selected system.
	ON	The MDI data for the selected system is registered as a machining program for the selected system. If the system is not selected, an empty (only EOR [%]) machining program is registered.

(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 5.3 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.

(Note 4) If the system is changed while correcting the MDI data, the corrections will be canceled.

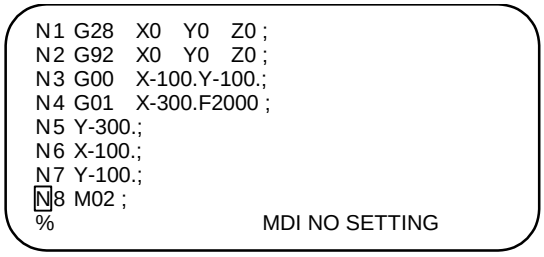
(4) Setting the MDI running start position

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.

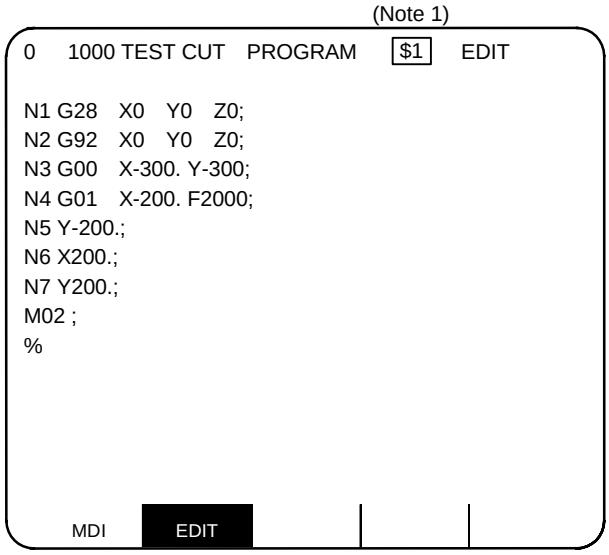


Press the INPUT key. ☞

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.

5.2.2 EDIT Screen Menu Function

(1) Menu when [EDIT MDI] screen is selected



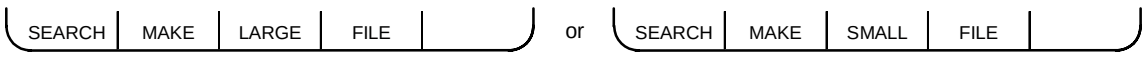
Menu	Function
Edit	Reverse display of EDIT menu means that EDIT screen is selected. Machining program can be set on the EDIT screen.
MDI	Use this key to change the EDIT screen to the MDI screen.

(Note 1) When using the 2-part system, the system name of the currently selected system is displayed as $1 (system 1) and $2 (system 2). This is not displayed when using a 1-part system. (Only L system)

(Note 2) Whether to show or hide name of the selected system can be switched with the parameters.

#1050 MemPrg	Details
0, 2, 4, 6	The name of the selected system is not displayed.
1, 3, 5, 7	The name of the selected system is displayed.

(2) EDIT screen extension operation menu



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.

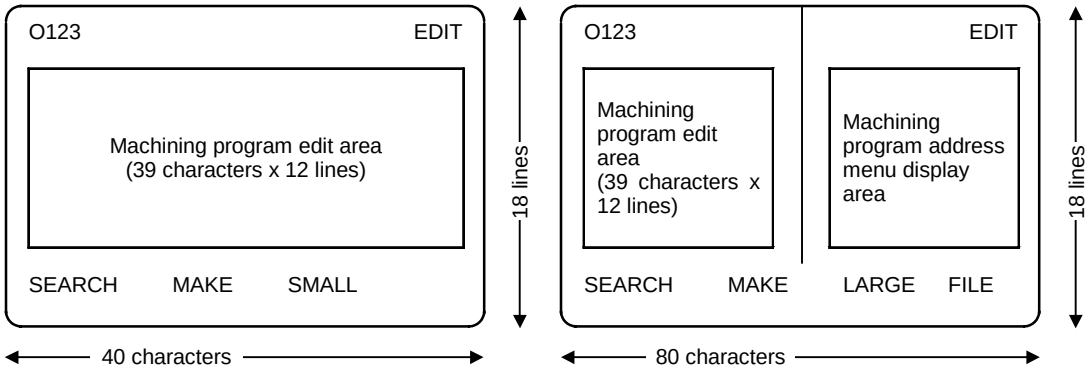
(3) Edit program call

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.

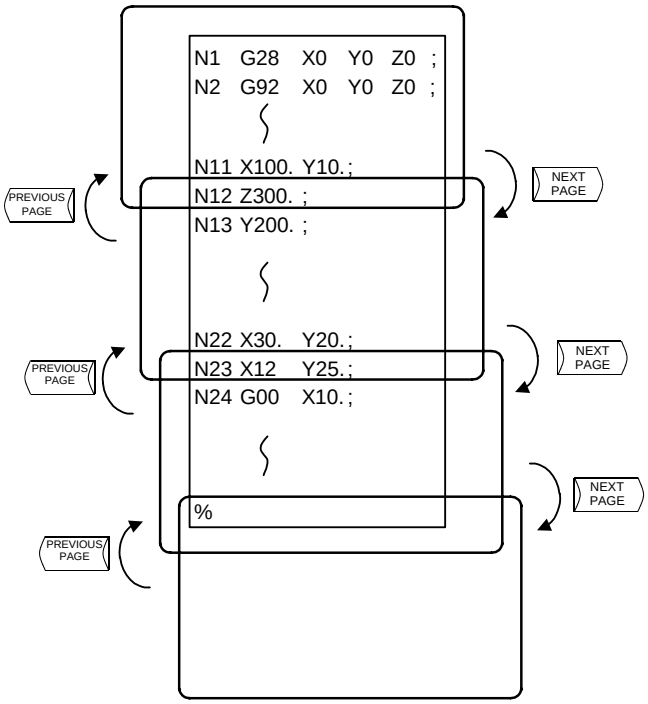
(Note 1) Switching the mode in the EDIT screen automatically changes the mode in the MDI screen.
(Note 2) During editing (while message "EDITING" is displayed on the lower right of the screen), menu keys ⟩LARGE⟩ and ⟩SMALL⟩ are disabled, i.e., pressing it does not change the mode. To change the mode, 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.
(Note 4) If the system is changed while editing the machining program, the edited details will be canceled.

5.3 Program Edit Operation

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

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 PREVIOUS PAGE or NEXT PAGE.
When the NEXT PAGE key is pressed, the data displayed at the screen bottom is moved to the screen top; when the PREVIOUS PAGE 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.

Press the NEXT PAGE key. ☞

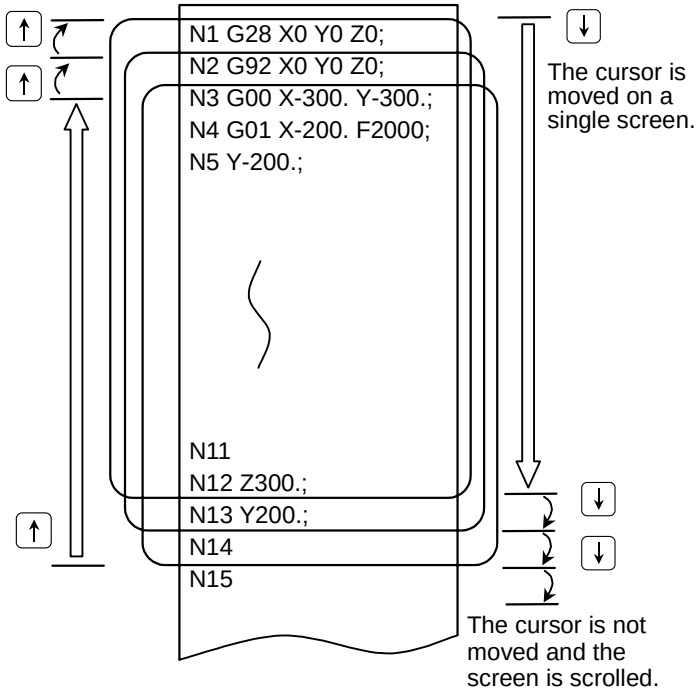
```
N1 G28   X0   Y0   Z0 ;
N2 G92   X0   Y0   Z0 ;
N3 G00   X-300. Y-300.;
N4 G01   X-200. F2000 ;
N5 Y-200.;
N6 X200.;
   ≀
N12 Z300.;
```

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

5.3.2 Data Display Update (One Line Scroll)

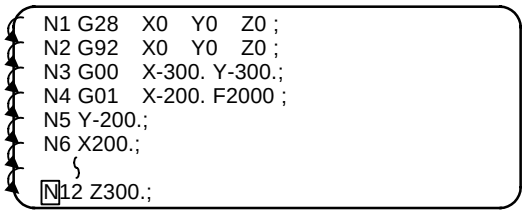
Data display on the screen can be updated in line units by using the ↑ or ↓ key.

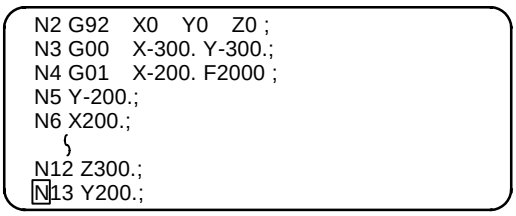
If the ↓ key is pressed when the cursor is placed at the screen bottom or if the ↑ key is pressed when the cursor is placed at the screen top, display is scrolled one line.



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.

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

```
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;
%
```

N7 Y200. ; block to

↓

G03 Y200. J100. ;

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

```
N7 Y200.;
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.

(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 word being both preceded and followed by space code.
3) When the data has been written into memory, the "EDITING" message disappears.

5.3.4 Data Insertion (SHIFT DELETE INS)

For example, let's try to insert data F5000 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.

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

(2) Press the SHIFT key, then DELETE INS key.

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

1) The characters to the right of the cursor are moved to the right (; in this case).
2) Data can be inserted in the position indicated by the cursor.

(3) Insert the data.
F5000

```
N7 G03 Y200.J100.F5000 ;
M02;
%                                EDITING
```

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 (; in this case) 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.
4) When data is entered by using the keys, the message "EDITING" is displayed.

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

```
N7 G03 Y200.J100.F5000 ;
M02;
%
```

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) When the data has been written into memory, the "EDITING" message disappears.

5.3.5 Deletion of One Character (DELETE INS)

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

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

```
N7 G03 Y200. J100.F5000;
M02;
%
```

(2) Press the DELETE INS key.

```
N7 G03 Y200. J100.F500␣;
M02;
%                                EDITING
```

1) The character 0 is deleted.
2) The cursor is automatically moved one column to the right.
3) When the key is pressed, 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) When the data has been written into memory, the "EDITING" message disappears.

5.3.6 Deletion of One Block (C.B CAN)

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

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

```
N1 G28 X0 Y0 Z0 ;
   ≀
N6 X200.;
N7 G03 Y200. J100. F500 ;
M02;
%
```

(2) Press the C.B CAN key.

1) Data in the entire block is deleted.
2) When the key is pressed, the message "EDITING" is displayed.

```
N1 G28 X0 Y0 Z0 ;
   ≀
N6 X200.;
M02;
%
                                EDITING
```

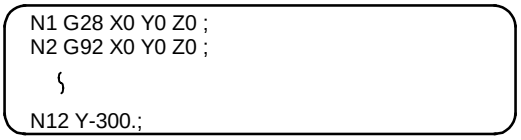
(3) After completion of correction, press the INPUT key.

1) The data in the block is deleted from memory.
2) The blocks following the deleted data block (M02; and % in this case) are moved forward for display.
3) When the data in the block has been deleted from memory, the "EDITING" message disappears.

```
N1 G28 X0 Y0 Z0 ;
   ≀
N6 X200.;
M02;
%
```

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 SHIFT key, then C.B CAN key.

1) The full screen becomes blank.
2) When the keys are pressed, the message "EDITING" is displayed.

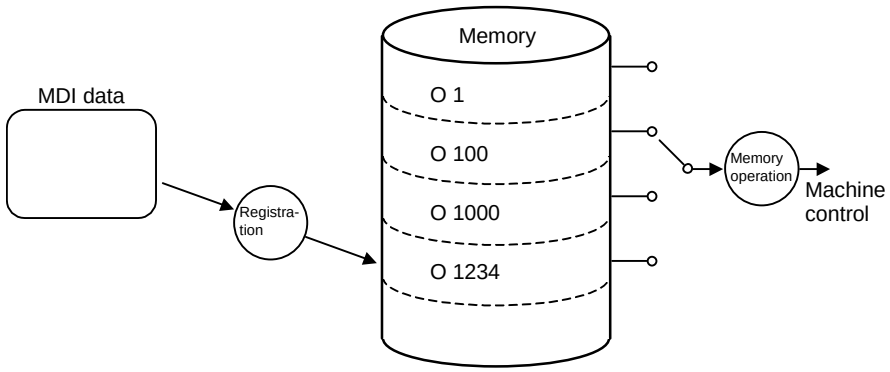
(2) Press the INPUT key.

1) The data displayed on the entire screen is deleted from memory.
2) Display is started at the block following the deleted data.
3) When the data has been deleted from memory, the "EDITING" message disappears.

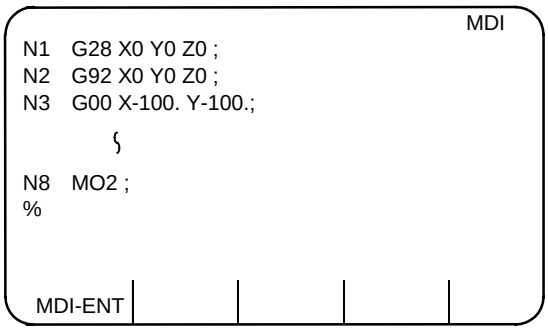
5.4 MDI Screen Extension Operation

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.

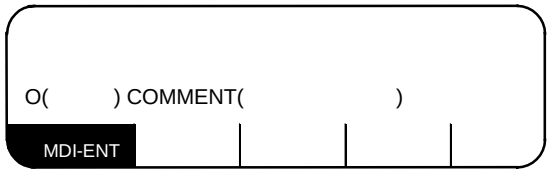


For example, assume that MDI data is set as shown in the right. The MDI data registration procedure in memory is described below:

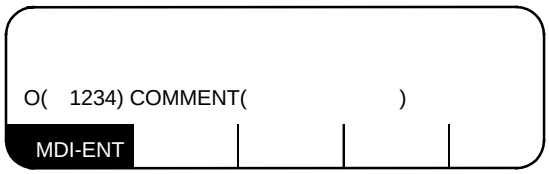


(1) Press the menu key MDI-ENT.

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

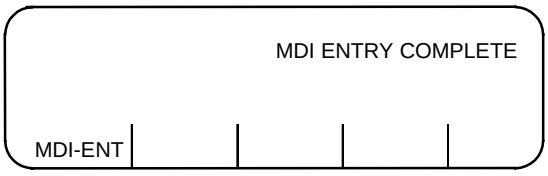


(2) Set the registered program number. A comment can also be set at the same time.
(Example)
O (1 2 3 4) COMMENT ( )



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

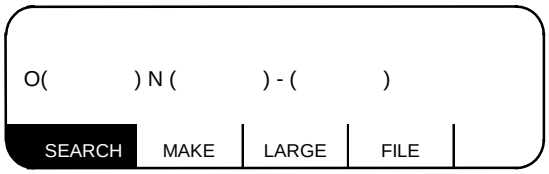
5.5 Edit Screen Extension Operation

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.

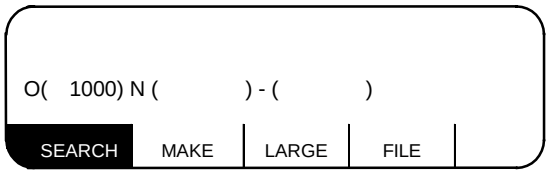
(1) Search for the program head
In the setting field, specify the program number of the program to be called.
The operating procedure is as follows:

(1) Press the menu key SEARCH.

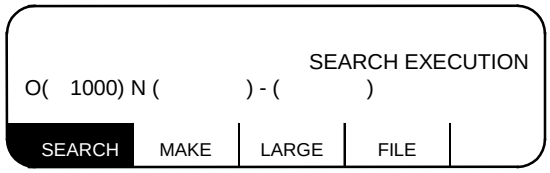


1) The setting area for "SEARCH" is displayed.

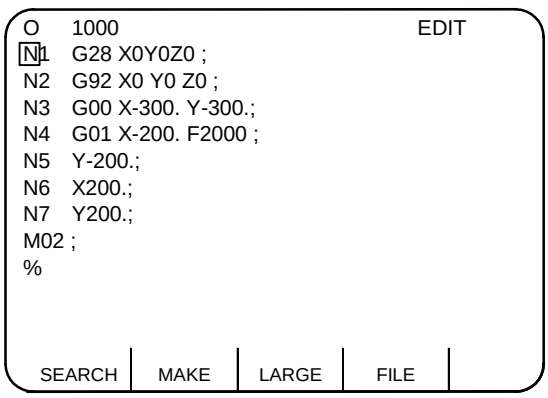
(2) Set the called program number.
(Example)
O (1 0 0 0) N ( ) - ( )



(3) Press the INPUT key.

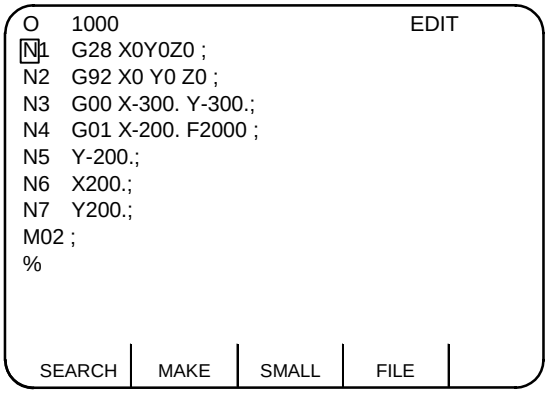


1) The message "SEARCH EXECUTION" is displayed during searching.
2) The specified program is displayed, beginning with top of the program.
3) The cursor is displayed at the top of the screen.
4) At normal completion of program head search, display of the setting area disappears and SEARCH menu display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



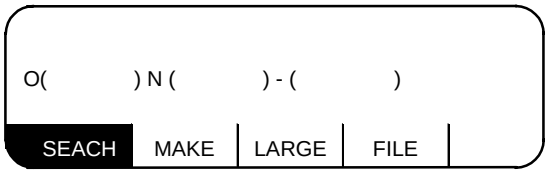
(2) Character string search

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.



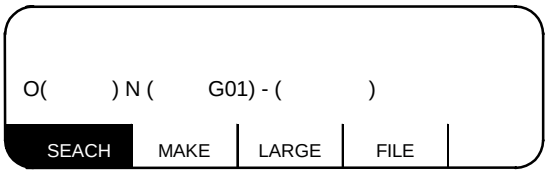
The operation procedure is described below:

(1) Press the menu key SEARCH.

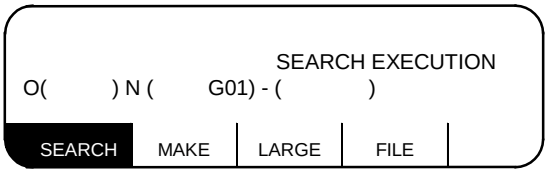


1) The setting area for "SEARCH" is displayed.

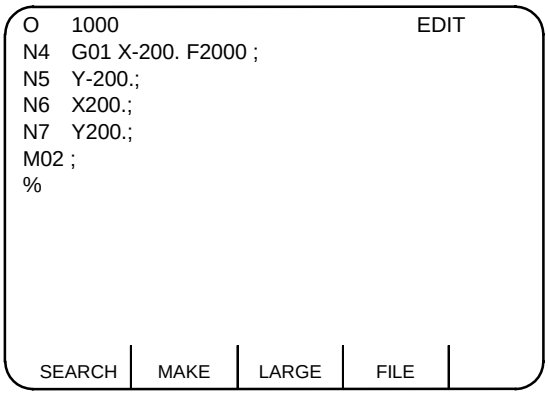
(2) Set the called program number and character string.
(Example)
O ( ) N (G 0 1) - ( )



(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.



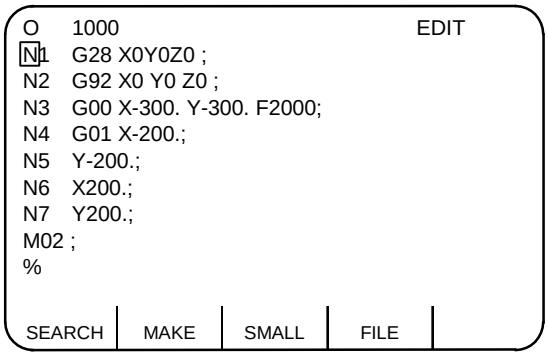
(Note 1) When a given character string is not found, a "NO CHARACTERS" message is displayed.
(Note 2) A string of up to 11 characters may be specified.
(Note 3) The specified character string is searched and identified in the specified number of character strings regardless of the preceding and subsequent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

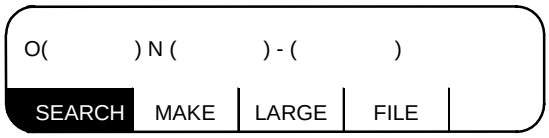
(3) Sequence number, block number search

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.



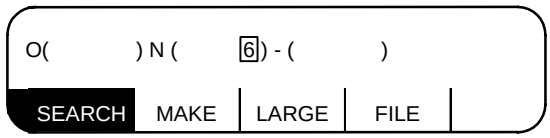
The operation procedure is described below.

(1) Press the menu key [SEARCH].

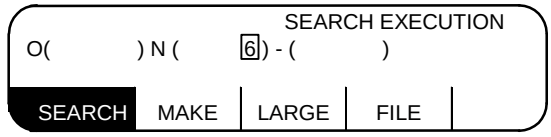


1) The setting area for "SEARCH" is displayed.

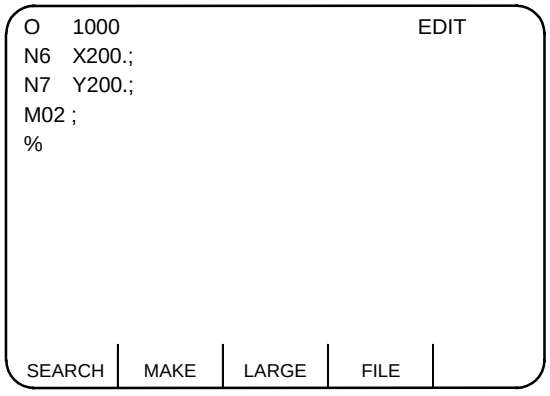
(2) Set the called program number, sequence number, and block number.
(Example)
O ( ) N (6) - ( )



(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.

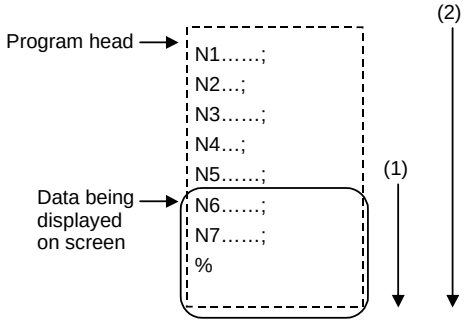


(Note 1) When a given N number is not found, an "NB NOT FOUND" message is displayed.
(Note 2) If a given program number is not found, a "PROG NOT FOUND" message is displayed.
(Note 3) The sequence number can be specified in a maximum of five digits.

(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"

(5) Precautions for 2-part system

When using the 2-part system, the methods for searching the program or block to be edited can be switched with the parameters.

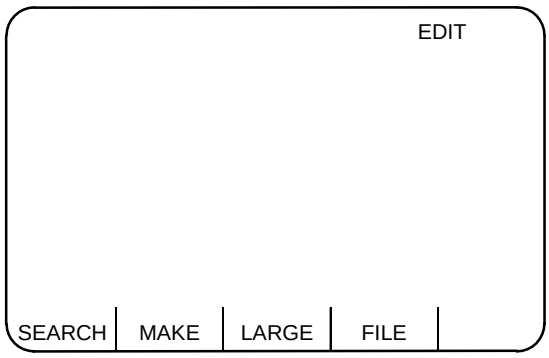
#1050 MemPrg	#1285 ext21/bit0	Details
0, 2, 4, 6	-	The machining program registered in the memory common for the systems is searched for with the designated program No., sequence No. and block No.
1, 3, 5, 7	OFF	Selected system: The machining program registered in the memory for each system is searched for with the designated program No., sequence No. and block No.
		System that is not selected: The searched machining program is held.
	ON	Selected system: The machining program registered in the memory for each system is searched for with the designated program No., sequence No. and block No.
		System that is not selected: The machining program registered in the memory for each system is searched for with the designated program No.

When using the 2-part system and the same number batch generation for all system programs is valid, the program is erased in the following manner according to whether programs are present for each system.

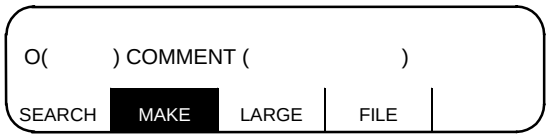
Presence of program		Operation
System 1	System 2	
Yes	Yes	The System 1 and System 2 machining programs are searched for simultaneously.
Yes	No	The System 2 machining program is newly created, and the System 1 and System 2 machining programs are searched for simultaneously. **(Note)** An error (E14) will occur if the number of programs is insufficient.
No	Yes	The System 1 machining program is newly created, and the System 1 and System 2 machining programs are searched for simultaneously. **(Note)** An error (E14) will occur if the number of programs is insufficient.
No	No	An error (E14) occurs.

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.

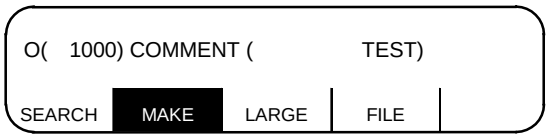


(1) Press the menu key [MAKE].



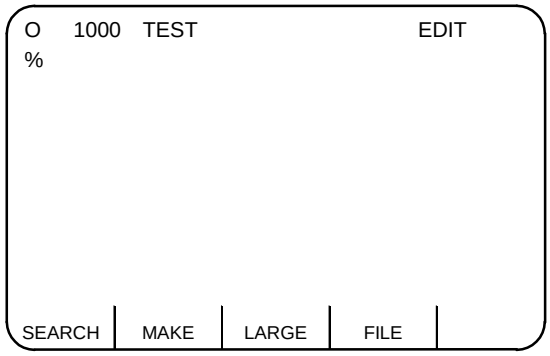
1) The setting area for "PROGRAM" is displayed.

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

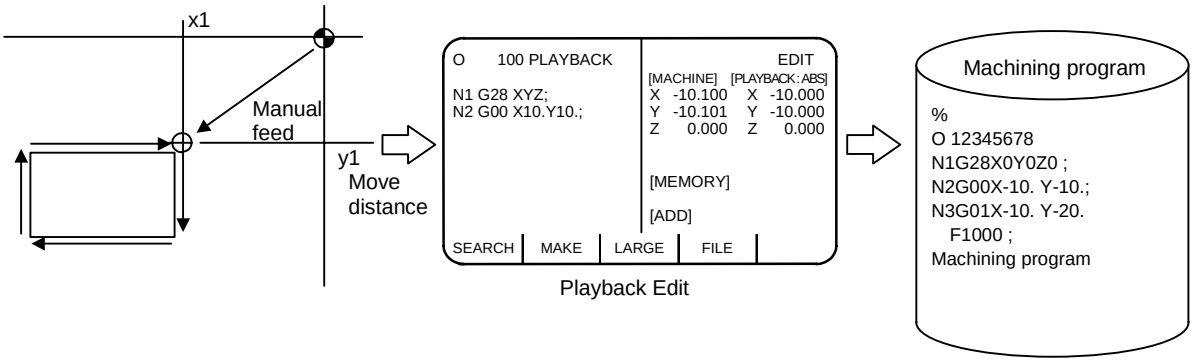
(Note 3) When using the 2-part system, the methods for registering a new program can be switched with the parameters.

#1050 MemPrg	#1285 ext21/bit0	Details
0, 2, 4, 6	-	The machining program is newly registered in the memory common for the systems.
1, 3, 5, 7	OFF	The machining program is newly registered in the memory for the selected system.
	ON	The machining program is newly registered in all system memories.

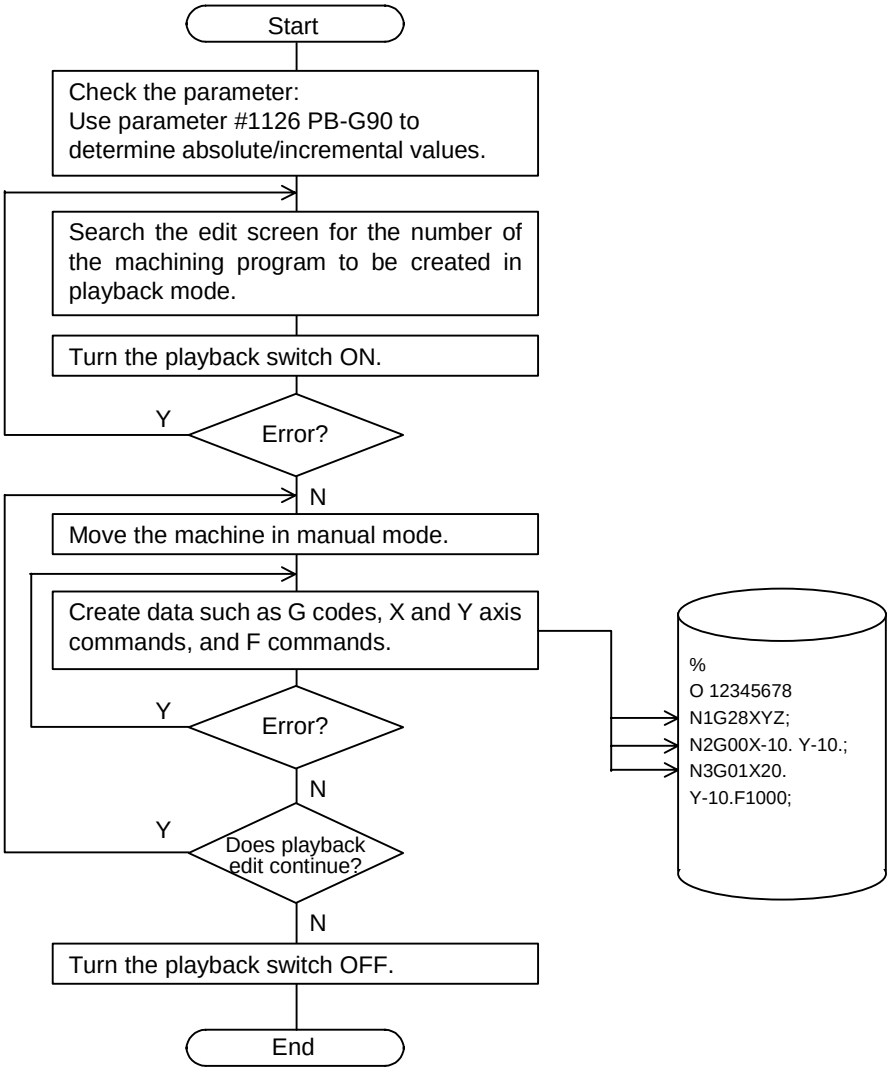
(Note 4) When using the 2-part system and the same number batch generation for all system programs is valid, an error (E11) will occur if the number being registered is already used in one of the systems.

5.6 PLAYBACK

The playback function enables creation of a program while trying sample machining by manual (handle or jog) feed or mechanical handle feed.
A machining program can be created with move distance data obtained by manual operation used as programmed command values.



Playback Edit



Machining program creation flowchart in playback mode

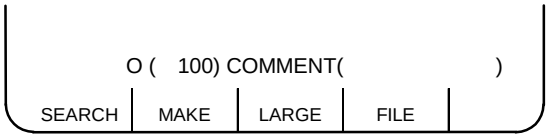
5.6.1 Playback Operation

(1) PLAYBACK screen

(a) Creating a program and editing it in playback mode

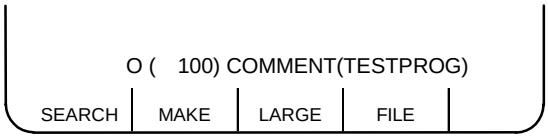
(1) Create a program:

Press the [EDIT MDI] key, then press menu keys [EDIT] and [MAKE].



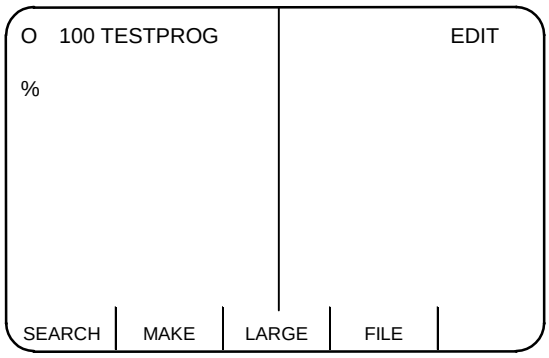
The setting area for "PROGRAM" is displayed.

Set the program number and comment in the data setting area.
(Example)
O (100)
COMMENT (TESTPROG)



Press the [INPUT] key.

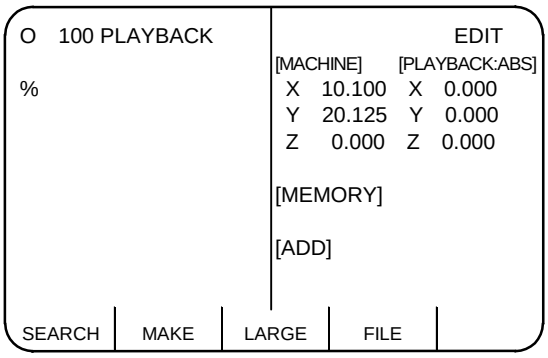
The specified program number and comment are displayed on the upper part of the screen, and one character data "%" is automatically stored in memory.



(2) Display the PLAYBACK screen:

Press the playback switch prepared on the machine side.

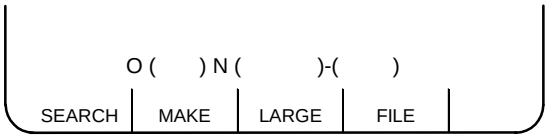
Because no program has been made, only "%" is displayed on the left side on the screen. The [MEMORY] field on the right side is blank.



(b) Editing a stored program in playback mode

(1) Display the EDIT screen.

Press the [EDIT MDI] key, then press menu keys ⟩EDIT⟩ and ⟩SEARCH⟩.

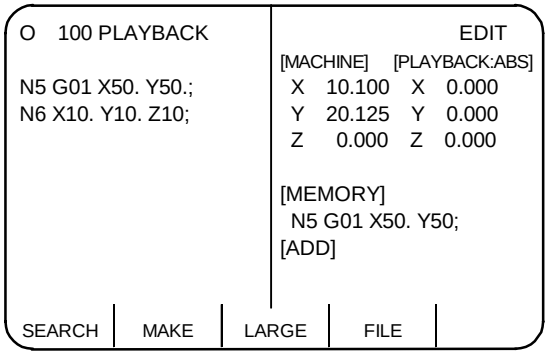


The setting area for "SEARCH" is displayed.

Set the numbers of the program and sequence to call in the data setting area, then press the [INPUT] key.

(Example) O (100) N (5) - ( )

Press the playback switch prepared on the machine side.



1) The specified program is searched from the beginning of the block containing the specified character string, then the program is displayed with the block placed on the top.
2) A cursor is placed on the top of the character string displayed.
3) Program editing starts with the block next to the specified one. The specified block is displayed in the [MEMORY] field.
4) Another cursor is displayed in the [ADD] field, allowing the program to be edited.

In either creating and editing a new program or editing a stored program in playback mode, select the PROGRAM screen and perform editing on the screen. Editing in playback mode is performed using the [ADD] field displayed on the right side on the screen. This therefore prevents the machining program displayed on the left half on the screen from being edited.
The cursor displayed on the left screen indicates the block displayed in the [MEMORY] field on the right screen.

(c) Invoking and editing an another program in playback mode

Press menu key ⟩SEARCH⟩ again, then repeat the operation described under item (2) .

(2) Playback editing

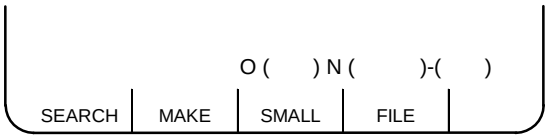
1) Set the parameter to specify whether to perform playback editing in absolute or incremental mode. To edit with absolute values, set #1126 PB-G90 to 1, and to edit with incremental values, set to 0. If the incremental mode is selected, INC is displayed after PLAYBACK on the screen. If the absolute mode is selected, ABS is displayed.

2) Select the EDIT screen.

Press the [EDIT MDI] key, then press menu key [EDIT].

Select "SEARCH".

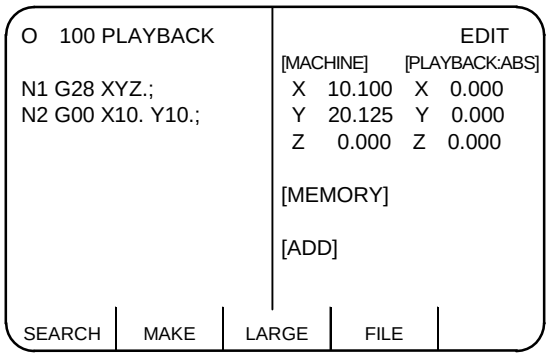
Press menu key [SEARCH].



Specify the program number and playback mode.

Specify the numbers of the program, sequence and block to be edited in playback mode, then press the [INPUT] key.
(Example) O (100) N ( ) - ( )

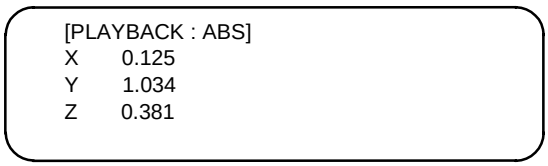
Press the playback switch prepared on the machine side.



This operation displays the PLAYBACK screen on the right half. A data insertion position can be selected by operating the cursor keys. For the details, see Section 5.6.2, "Edit Operation".

3) Move the machine in manual mode.

Move the machine from the work origin to the target position in handle or jog feed mode.



4) Convert the playback move distance into machining program data.

Enter the necessary data, such as sequence number and G code.
(Example)
N 1 0 G 0 0

```
[PLAYBACK : ABS]
X      0.125
Y      1.034
Z      0.010

[MEMORY]

[ADD]      N10G00 □
```

Press axis address keys such as X and Y.
(Example) X Y

```
[PLAYBACK : ABS]
X      0.125
Y      1.034
Z      0.010

[MEMORY]

[ADD]      N10G00X0.125Y1.034 □
```

1) When an axis address key is pressed, the playback move distance is displayed after the corresponding axis address.
2) If an axis address key is pressed while the playback counter is operating, playback data at that time is displayed.

Press the INPUT key.

```
[PLAYBACK : ABS]
X      0.125
Y      1.034
Z      0.010

[MEMORY]    N10G00X0.125Y1.034;

[ADD] □
```

5) End playback editing.
Turn the playback switch OFF to end playback editing; the screen returns to the normal editing screen.

(3) Notes on playback operation

1) The number of characters specified in the [ADD] field must not exceed 96.
2) If an EOB (;) is omitted at the end of the program created in the [ADD] field, it is automatically appended when the INPUT key is pressed.
3) Blocks can be delimited by inserting an EOB (;) between X and Y.
4) If an incorrect data is entered, the error message is displayed when the INPUT key is pressed. (See the operation messages.)
5) Do not edit macro statements in playback mode; otherwise, for example, if an attempt is made to input "XOR", input of the X may play back the X.
6) If one of the following items is operated during playback editing, another program may be called or the state where no program has been called may occur:

SEARCH ERASE CONDENSE PROGRAM NO. CHANGE

(4) Playback counter display

Operation of the playback counter may depend on the control unit mode.

	#1126 PB-G90=0	#1126 PB-G90=1
Counter display at start of playback	Displays 0.	Displays the current value (2) (added by a manual interrupt value if any).
Setting by position data [PLAYBACK: ***] X 10.002 [ADD] G01X10. ; INPUT ☞	[PLAYBACK: INC] X 0.002 MEMORY] G01X10. ; [ADD] The difference between an axis command value and playback counter remains in the playback counter.	[PLAYBACK: ABS] X 10.002 [MEMORY] G01X10. ; [ADD] The playback counter is not changed and the move distance is accumulated.
Setting G92 (counter preset) [PLAYBACK: ***] X 20.000 [ADD] G92X10. ; INPUT ☞	[PLAYBACK: INC] X 0.000 [MEMORY] G92X10. ; [ADD] Regardless of the axis command value following G92, the playback counter is cleared to 0.	[PLAYBACK: ABS] X 10.000 [MEMORY] G92X10. ; [ADD] The axis command value following G92 is set in the playback counter.

(5) Coordinates to be stored

1) A coordinate value is stored in memory with a decimal point in playback mode. The trailing 0s are omitted.

   (Example) Playback counter Memory
   X 0.000 → X0
   X 10.000 → X10.

2) The No. of digits in the axis command value during playback will depend on the input unit (#1015 cunit) for each axis.

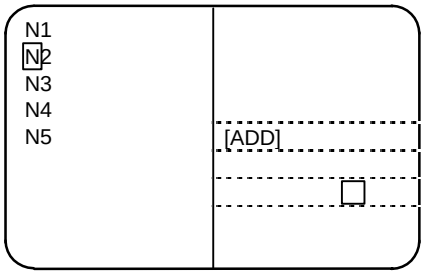
5.6.2 Edit Operation

(1) Moving the cursor

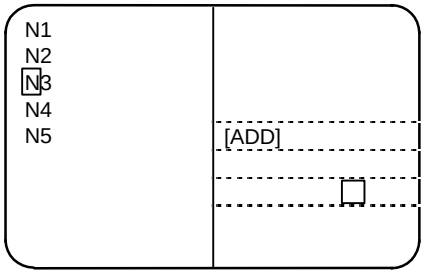
The block insertion position or deletion block can be specified by moving the cursor vertically on the left side on the screen.

(a) Moving the cursor down

Move the cursor in the [ADD] field down to the third line. (↓)



Press the ↓ key again.

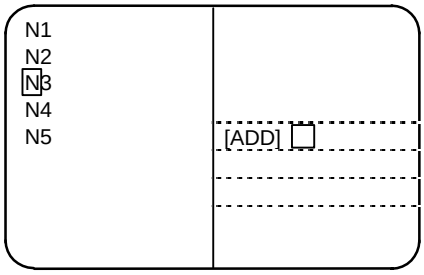


This moves the cursor on the left side on the screen down.

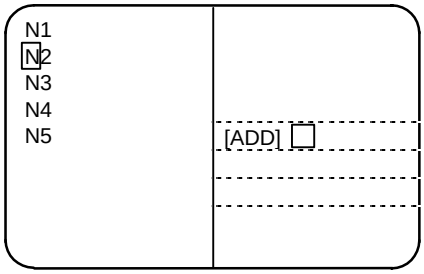
When the cursor key ↓ is further pressed with the cursor located at the bottom of the data field, data scrolls up one line each time. The cursor remains on the bottom.

(b) Moving the cursor up

Move the cursor in the [ADD] field up to the first line. (↑)



Press the ↑ key again.



This moves the cursor on the left side on the screen up.

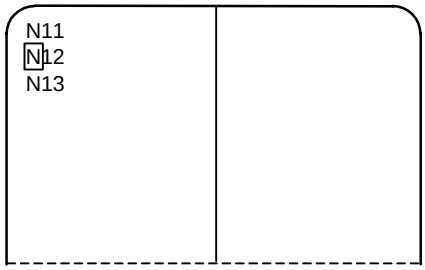
When the cursor key ↑ is further pressed with the cursor placed at the top of the data field, data scrolls down one line each time; previous block data is displayed at the top.

(2) Insertion of block

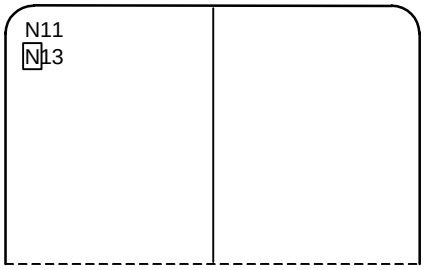
A block can be inserted following the block specified by the cursor on the left side on the screen.

(3) Deletion of block

Move the cursor to the block to be deleted. (↑ ↓)



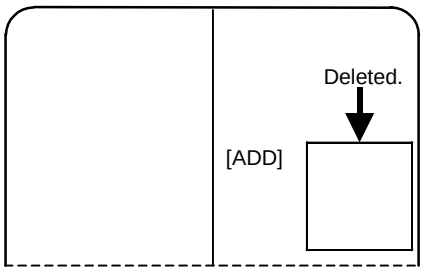
Press the SHIFT and C.B CAN keys at the same time.



The N12 block is deleted, and the updated data is written in memory.

(4) [ADD] Program deletion (C.B CAN)

The program being created in the [ADD] field on the right side of the screen is completely deleted. The cursor automatically returns to the head of the [ADD] field.



5.6.3 Limitations

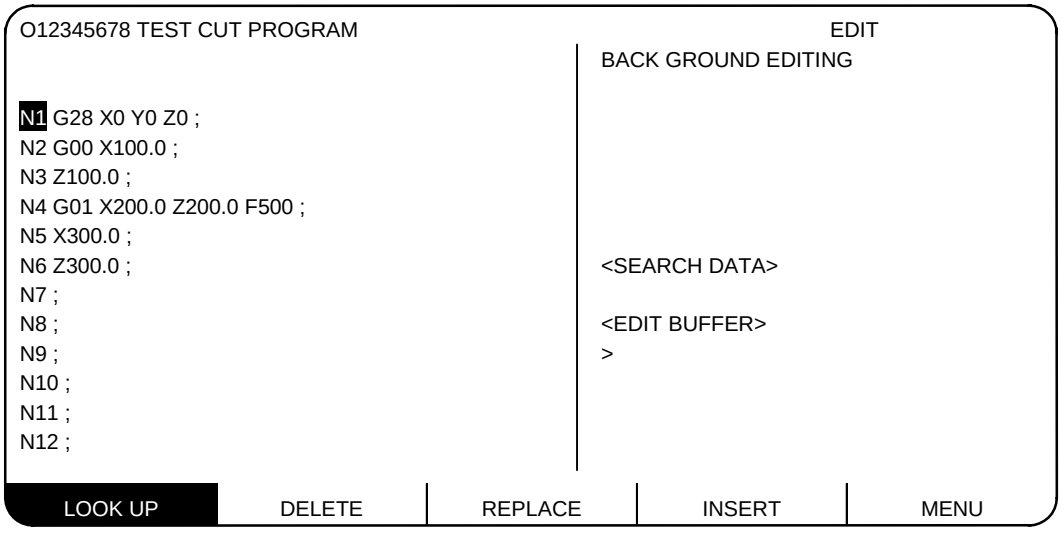
(1) Playback editing is disabled in the machine lock state.
(A move distance during machine lock is ignored.)
(2) The program that is running under automatic operation cannot be edited in playback mode.
(Generally, programs that are running under automatic operation cannot be edited.)
(3) A subprogram used in the fixed cycle cannot be edited in playback mode.
(Generally, subprograms used in the fixed cycle cannot be edited.)
If the playback switch is set to ON on the SEARCH screen, an error results.)
(4) While message "EDITING" is displayed, playback editing is disabled.
If the playback switch is set to ON, an operation error results.
(5) Playback editing is disabled in large-size mode.
If the playback switch is set to ON, an operation error results.

5.7 Word Editing

In addition to the conventional editing function, program editing in word units can be selected.
The word editing function allows deletion, replacement, insertion, etc., of the program in word units, enabling concise creation of programs.

(Note 1) Word editing is valid when SETUP PARAMETER "#1139 edtype" is set to 2.
(Note 2) Use the word editing function with #1050 MemPrg = 0 (system common program control). If #1050 MemPrg is set to a value other than 0, the functions may be limited as shown below.

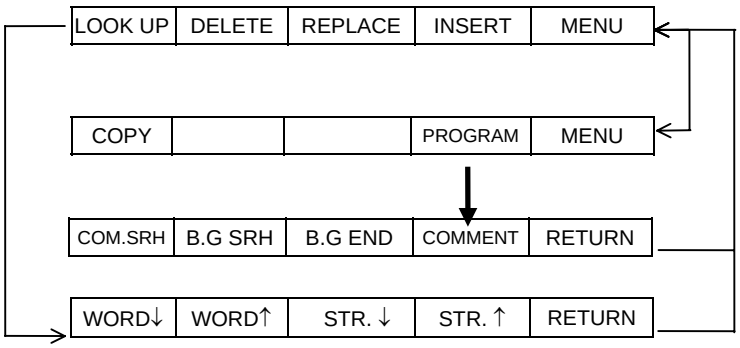
#1050 MemPrg	Display	Editing, cursor movement	Menu operations
0	Enabled	Enabled	Enabled
2, 4, 6	Disabled	Enabled	Enabled
1, 3, 5, 7	Disabled	Disabled [error (E05)]	Disabled [error (E74)]



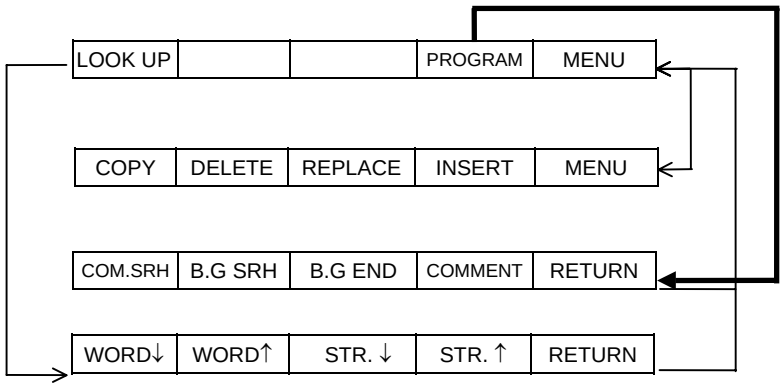
(1) EDIT Screen Operation Menu

The operation menu format changes according to bit 6 of the SETUP PARAMETER "#1227 aux11/bit6".

When the bit is OFF (menu type 1)



When the bit is ON (menu type 2)



Menu	Function
SEARCH	This changes the menu for selecting the search direction.
DELETE	The word at the cursor position can be deleted. (The deleted word is set in the EDIT BUFFER area.)
REPLACE	The word at the cursor position can be replaced with the data in the EDIT BUFFER area. (The EDIT BUFFER area data is not cleared.)
INSERT	A word in the EDIT BUFFER area can be inserted in the location immediately after the word at the cursor position. (The EDIT BUFFER area data is not cleared.)
COPY	The word at the cursor position can be copied into the EDIT BUFFER area.
PROGRAM	The menu changes to the one for searching the program. The searched program and a list of programs are displayed.
COM. SRH	The program Nos., sequence Nos., and block Nos. for carrying out automatic operation can be searched from the machining programs registered in the NC memory.
B.G SRH	The program Nos. sequence Nos. and block Nos. for background editing can be searched. If a program No. not registered in the NC memory is set, a new machining program will be registered.
B.G-END	This quits the background editing function.
COMMENT	An outline of the machining program functions, specifications, applications, etc., can be set as a comment.
RETURN	This returns to the top menu.
WORD↓	This searches in the downward direction. The word matching the search data is searched, and the cursor moves to that word. (The search data is not cleared.)
WORD↑	This searches in the upward direction. The word matching the search data is searched, and the cursor moves to that word. (The search data is not cleared.)
STR. ↓	This searches in the downward direction. The character string matching the search data is searched, and the cursor moves to that word. (The search data is not cleared.)
STR. ↑	This searches in the upward direction. The character string matching the search data is searched, and the cursor moves to that word. (The search data is not cleared.)

(2) Foreground/Background Editing Explanation

(a) In the background editing mode

1) The background editing mode lasts from the BG search to the BG quit.
2) "BACKGROUND EDITING" is displayed on the screen.
3) Program indexing is carried out if the INPUT key is pressed during background editing.
4) Even during program execution, programs besides the one in execution can be edited.
5) If an operation search is commanded from the EDIT screen during background editing, the background editing mode is quit.
6) During background editing, programs not in the background editing mode can be externally searched, searched & started, or operation searched from a screen besides the EDIT screen, and the background editing mode will not quit.
   Note that background editing mode will quit if a program in the background editing mode is externally searched, searched & started, or operation searched from a screen besides the EDIT screen.

(Note) A BG search is not possible for programs in an operation search or programs in operation. (The error message "E190 FORE EDITING")

(b) In the foreground editing mode

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

(Note1) "EDIT POSSIBLE" is displayed on the screen when editing is possible, "EDIT IMPOSSIBLE" is displayed when editing is not possible.

(Note2) "EDIT IMPOSSIBLE" is displayed in the fixed cycle mode during feed hold or single block stop.

(c) In modes besides the foreground editing mode

1) When the display request (Y23C) is ON during program operation, the program in operation is displayed on the left side of the screen.

5.7.1 Handling of the Various Keys During Word Editing

Various keys during word editing

Key data	Edit area (left side)	Edit buffer (right side)	Details
Cursor keys (↑, ↓, ←, →)	○	×	→: This key moves the cursor to the next word in the order direction. ←: This key moves the cursor to the previous word in the opposite direction of the order. ↓ : This key moves the cursor to the head word of the next block. ↑ : This key moves the cursor to the head word of the previous block.
Page keys	○	×	NEXT: This key changes the screen to the next page in one screen units, and moves the cursor to the head word. BACK: This key changes the screen to the previous page in one screen units, and moves the cursor to the head word.
DEL	○	×	This key functions the same as the "DELETE" menu key.
INS	○	×	This key functions the same as the "INSERT" menu key.
Alphabetic keys, numeric keys, symbol keys (0 to 9, A to Z, etc.)	×	○	These keys input characters in the edit buffer/search data. The edit buffer/search data is cleared at the alphabetic, numeric, or symbol key input.
C. B	×	○	This key deletes the last character input in the edit buffer and search data. (This key functions the same as the Back Space key.)
CAN	×	×	Invalid
INPUT	○	×	When the cursor is at the head of the block: That block is searched. The operation starts from the designated block. When the cursor is at a position besides the head of the program block: The top of the program is searched. Operation starts from the top of the program.
CALC	×	×	Invalid

Word character judgment method

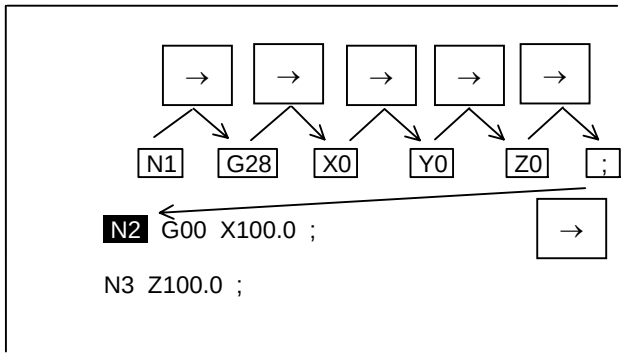
(1) Data with any of the following head characters are handled as words.
- A to Z
- ( ), # / ! % ; [ ]

(2) Macro statements are handled as word characters.
Examples of macro statements: GOTO, DO, WHILE, IF, OR, XOR, etc.

5.7.2 Searching Word Units

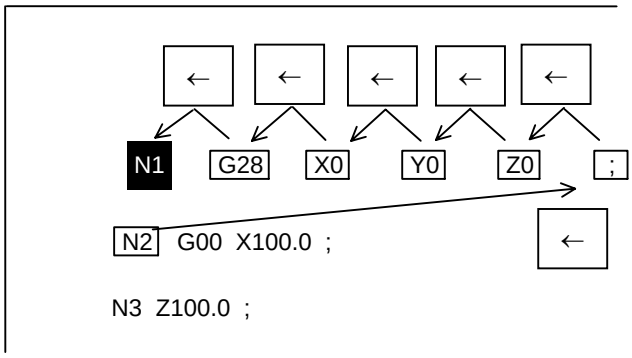
(1) → key

This key moves the cursor to the next word in the order direction.



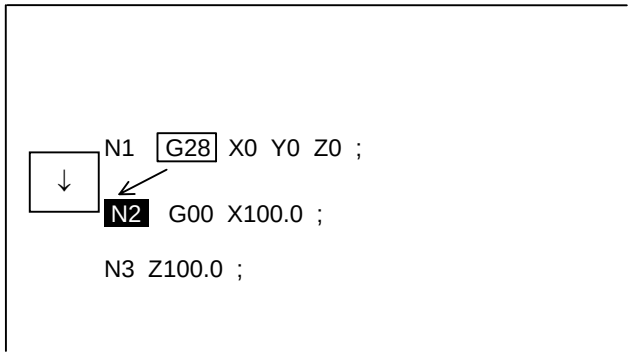
(2) ← key

This key moves the cursor to the previous word in the opposite direction of the order.



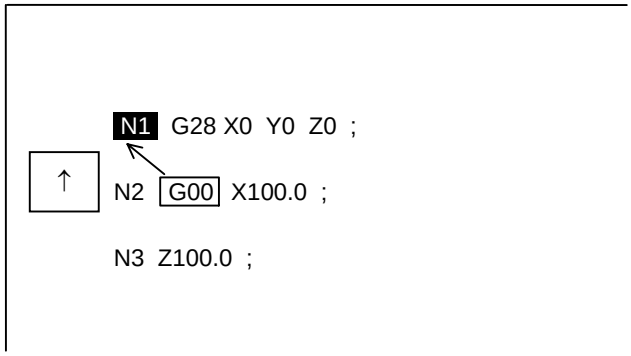
(3) ↓ key

This key moves the cursor to the head word of the next block.



(4) ↑ key

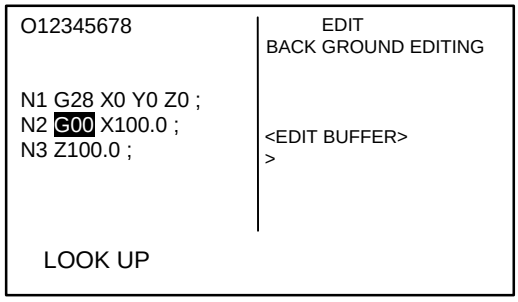
This key moves the cursor to the head word of the previous block.



5.7.3 Word Search

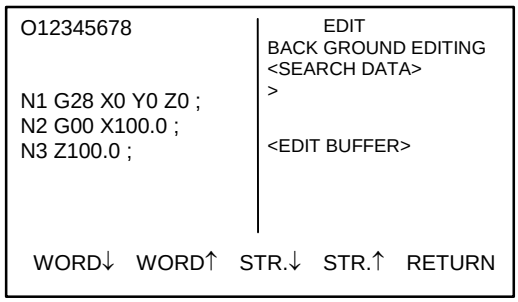
The word matching the search data is searched, and the cursor moves to the head of that word. (The search data is not cleared.)

Use the [WORD↓] and [WORD↑] keys in the word search.



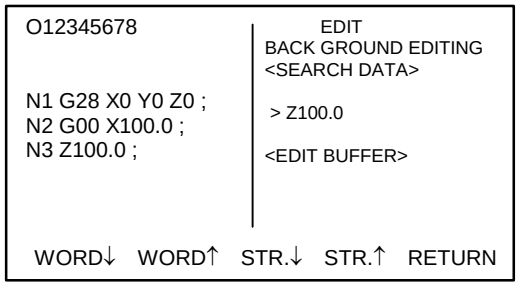
Press the menu key [SEARCH]. ☞

1) The cursor moves to <SEARCH DATA>. (The edit data is not cleared.)

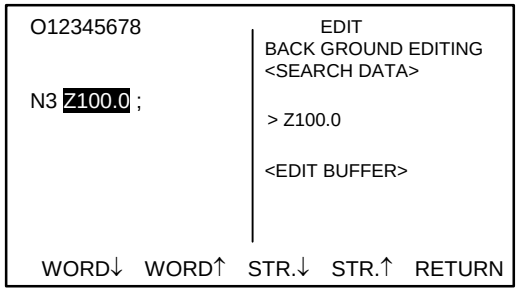


Key input the word data to be searched. ☞
(Ex.) [Z] [1] [0] [0] [.] [0]

1) Up to 11 characters of search data can be designated.



Select a search in the upward or downward direction. ☞
(Ex.) [WORD↓]



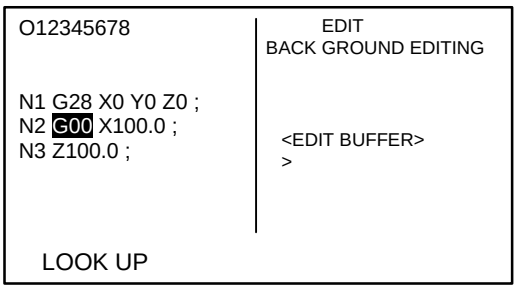
1) The message "SEARCH EXECUTION" appears during the search.
2) Words matching the search data are searched, starting from the word at the cursor position.
3) The cursor moves to the top of the word that was searched.
4) The search data is not cleared.
5) The screen returns to the 1st menu after the search is finished. (The search data is not cleared.)
6) The screen returns to the 1st menu if the menu key [RETURN] is pressed.

(Note 1) The message "NO CHARACTERS" appears on the screen if the designated word cannot be found.
(Note 2) If a word character is input in the search data after the menu key is pressed, the character will be input after the search data buffer is cleared.
(Note 3) The search data is valid until ; (EOB). Only one block can be searched at a time.
(Note 4) The [C.B CAN], [SHIFT], and [DELETE INS] keys are invalid while the search menu is displayed.

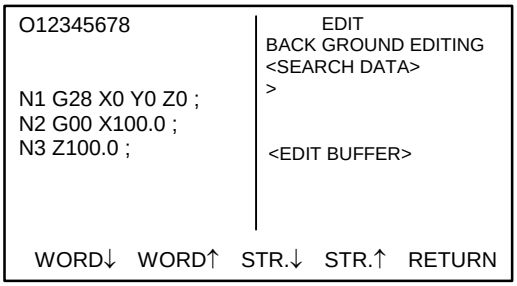
5.7.4 Character String Search

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

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

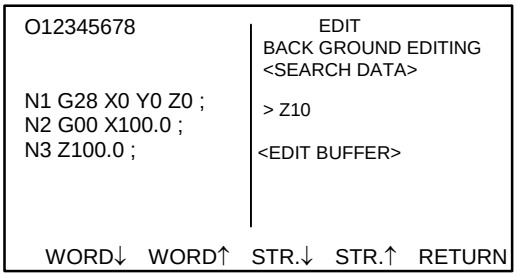


Press the menu key [SEARCH].



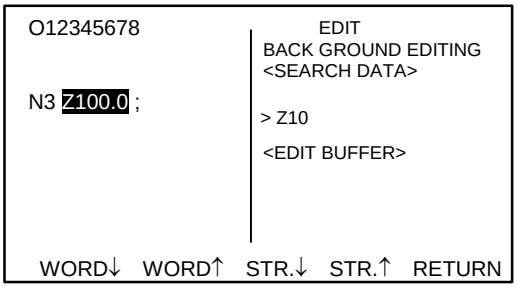
1) The cursor moves to <SEARCH DATA>.

Key input the character string data to be searched.
(Ex.) [Z] [1] [0]



1) Up to 11 characters of search data can be designated.

Select a search in the upward or downward direction.
(Ex.) [STR.↓]



1) The message "SEARCH EXECUTION" appears during the search.
2) Character strings matching the search data are searched, starting from the character string at the cursor position.
3) The cursor moves to the top of the word that was searched.
4) The search data is not cleared.
5) The screen returns to the 1st menu after the search is finished.
6) The screen returns to the 1st menu if the menu key [RETURN] is pressed.

(Note 1) The message "NO CHARACTERS" appears on the screen if the designated character string cannot be found.

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

(Note 3) Macro statements are not handled as 1 word of data during a character string search, so the operation differs from that of normal character string searches. For example, if the character "GO" is designated for [GOTO], and a character string search is executed, the cursor will appear at the [GOTO] position.

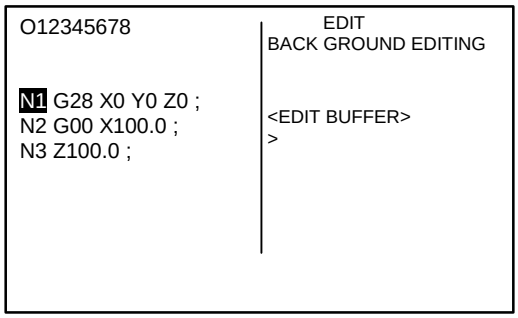
(Note 4) If a word character is input in the search data after the menu key is pressed, the character will be input after the search data buffer is cleared.

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

(Note 6) The [C.B CAN], [SHIFT], and [DELETE INS] keys are invalid while the search menu is displayed.

5.7.5 Deleting Words

The word at the cursor position can be deleted.

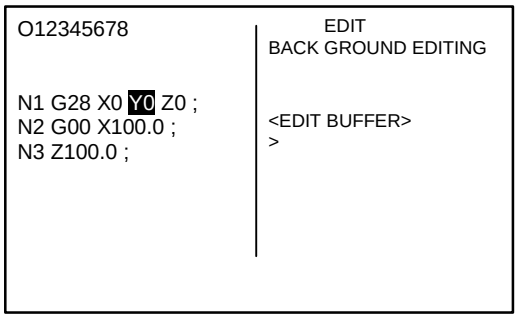


Move the cursor to the word to be deleted.

→ ← ↑ ↓

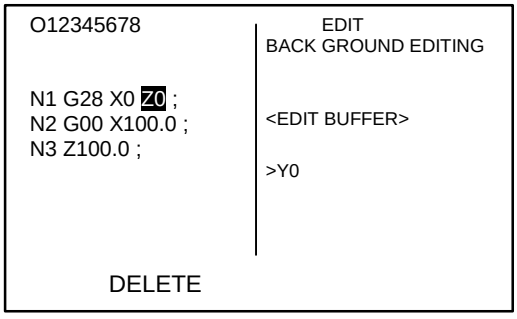
NEXT PAGE PREVIOUS PAGE

☞



Press the menu key DELETE.

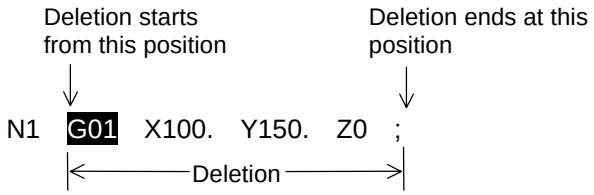
☞



1) The word at the cursor position is deleted.
2) The cursor moves to the next word.
3) The deleted word is set in the "EDIT BUFFER" area.

5.7.6 Deleting Lines

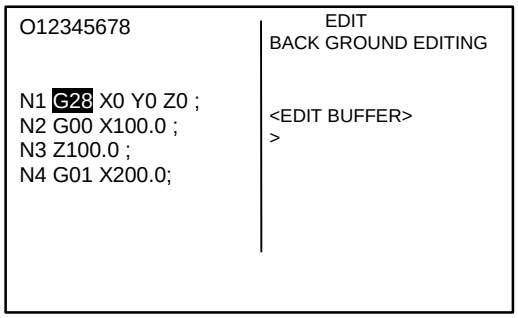
The line from the current cursor position to [EOB] (;) is deleted.



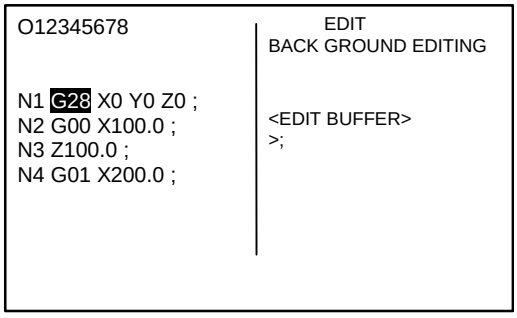
Move the cursor to the head word of the line to be deleted.

[→] [←] [↑] [↓]

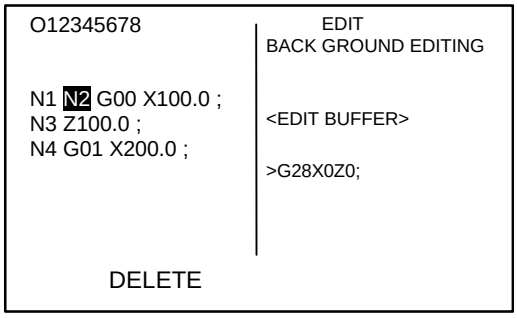
[NEXT PAGE] [PREVIOUS PAGE]



Key input [EOB].



Press the menu key [DELETE].



1) The line from the word at the cursor position to EOB (;) is deleted.
2) The cursor moves to the head word of the next line.
3) The deleted line is set in the "EDIT BUFFER" area.

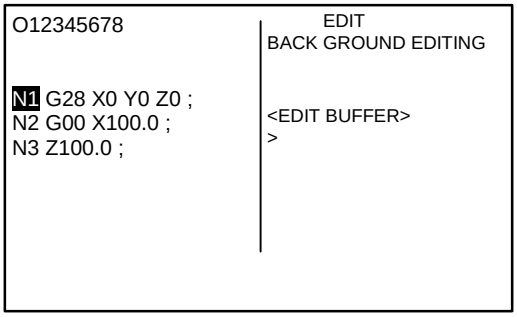
(Note 1) Only the EOB (;) key input in the EDIT BUFFER area is valid.

(Note 2) Up to 96 characters of the deleted line, starting from the head word, are set in the EDIT BUFFER area.

(Note 3) After the line is deleted, the deleted words (lines) will be added into the EDIT BUFFER area every time deleting operation is carried out.
Up to 96 characters can be stored in the EDIT BUFFER, so the other characters will be ignored.

5.7.7 Replacing Words

The word at the cursor position can be replaced with a word in the EDIT BUFFER area data.
(The EDIT BUFFER area data is not cleared.)

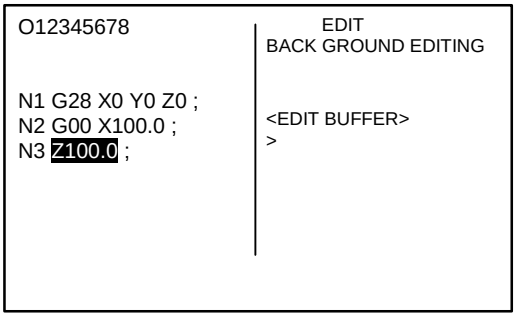


Move the cursor to the word to be replaced.

[→] [←] [↑] [↓]

[NEXT PAGE] [PREVIOUS PAGE]

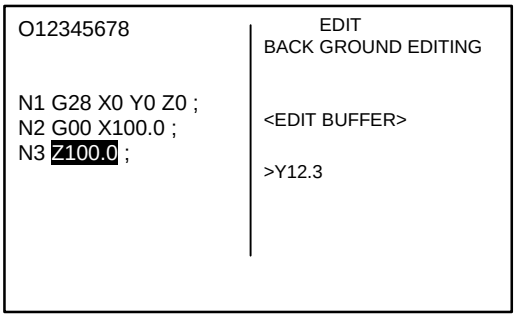
☞



Key input the word to be replaced into the EDIT BUFFER area.

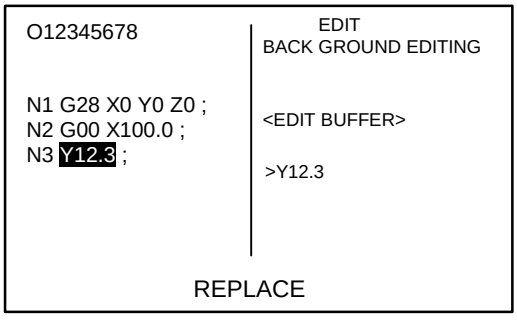
(Ex.) [Y] [1] [2] [.] [3]

☞



Press the menu key [REPLACE].

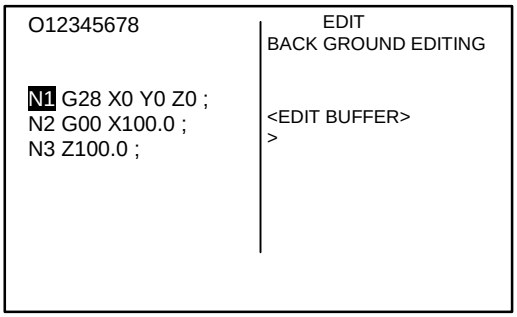
☞



1) The word at the cursor position is replaced with the EDIT BUFFER area data.
2) The cursor appears at the word that was replaced.
3) The EDIT BUFFER area data is not cleared.

5.7.8 Inserting Words

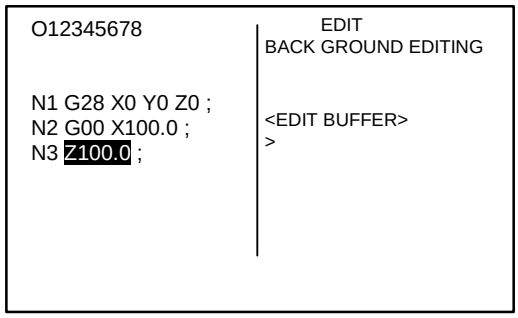
(1) A word in the EDIT BUFFER area can be inserted in the location immediately after the word at the cursor position.
(The EDIT BUFFER area data is not cleared.)



Move the cursor to the word immediately before the position of the word to be inserted.

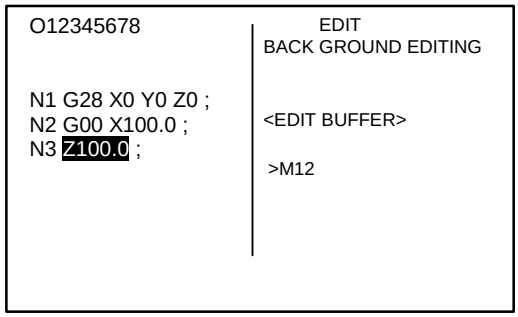
→ ← ↑ ↓

NEXT PAGE PREVIOUS PAGE

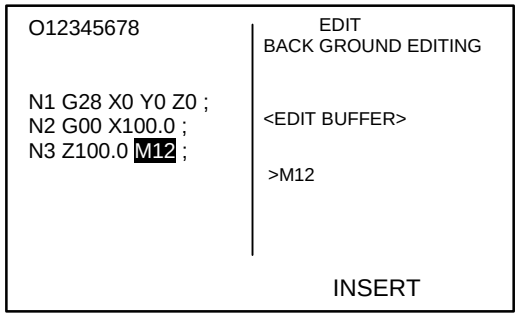


Key input the word to be inserted into the EDIT BUFFER area.

(Ex.) M 1 2



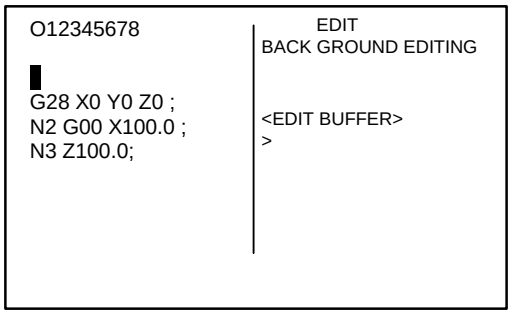
Press the menu key INSERT.



1) The EDIT BUFFER area data is inserted immediately after the word at the cursor position.
2) The cursor moves to the word that was inserted.
3) The EDIT BUFFER area data is not cleared.

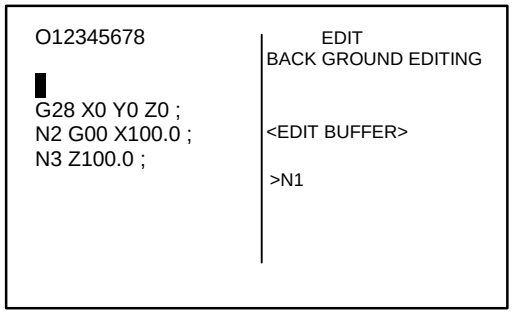
(2) A Word in the EDIT BUFFER can be inserted before the head word of the program
(The EDIT BUFFER area data is not cleared.)

Move the cursor to the empty line at the top of the program.
[→] [←] [↑] [↓]
[NEXT PAGE] [PREVIOUS PAGE]

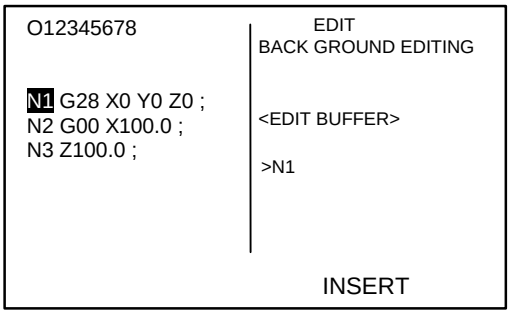


1) Deletion and replacement operations are ignored in this state.
The word insert function will be canceled if the [NEXT PAGE] key is pressed.

Key input the word to be inserted into the EDIT BUFFER area.
(Ex.) [N] [1]



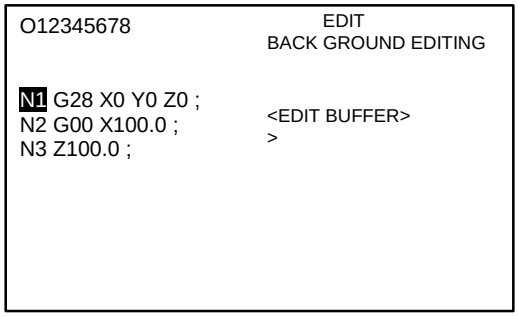
Press the menu key [INSERT].



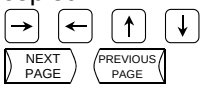
1) The EDIT BUFFER area data is inserted at the top of the program.
2) The cursor moves to the word that was inserted.
3) The EDIT BUFFER area data is not cleared.

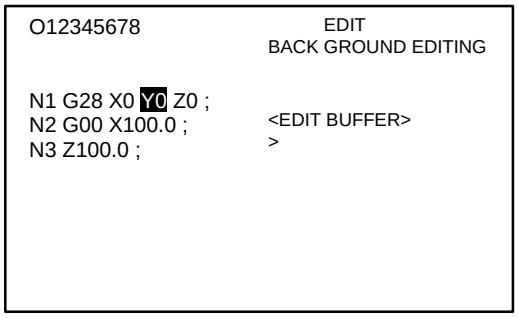
5.7.9 Copying Words

The word at the cursor position can be copied into the EDIT BUFFER area.

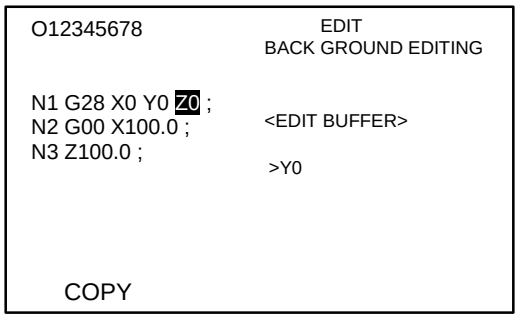


Move the cursor to the word to be copied.





Press the menu key [COPY].

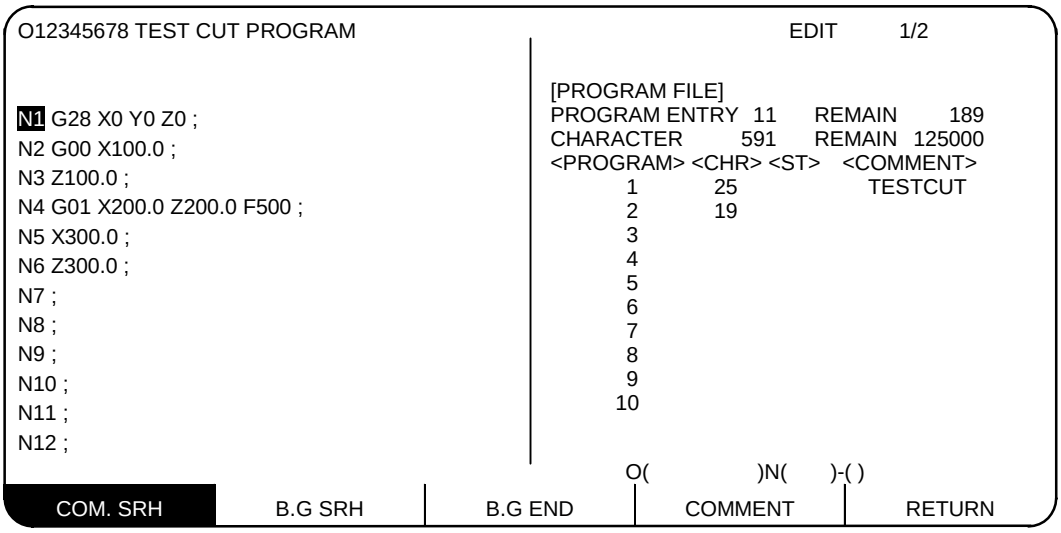


1) The word at the cursor position is set in the EDIT BUFFER area.
2) The cursor moves to the next word.

(Note 1) "%" cannot be copied.

5.7.10 Program

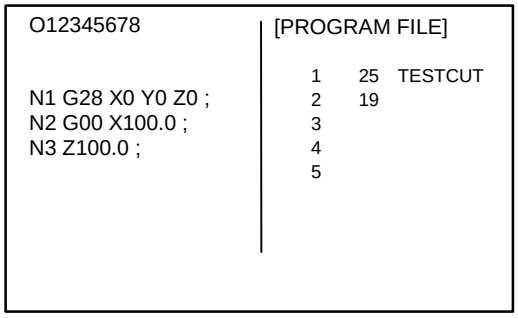
When the menu key PROGRAM is pressed, the searched program appears on the left side of the screen, and a list of programs registered in the memory appears on the right side of the screen. The operation search menu (COM.SRH) is highlighted, and the setting area is displayed.



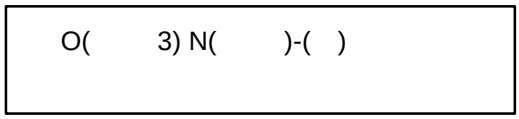
Display item	Details	
012345678	This item displays the program No. that was searched.	
N1 G28 X0 Y0 Z0 ; N2 G00 X100.0 ; N3 Z100.0 ; N4 G01 X200.0 Z200.0 F500 ; N5 X300.0 ; N6 Z300.0 ; . .	This item displays the program that was searched.	
PROGRAM ENTRY and REMAIN	The No. of programs already registered as user machining programs appears in the PROGRAM ENTRY column. The remaining No. of programs that can be registered appears in the REMAIN column. The total of the No. of registered programs and the remainder is the max. No. of programs. This figure is determined by the specifications.	
CHARACTER and REMAIN	The No. of characters already registered as user machining programs appears in the CHARACTER column. The remaining No. of characters that can be registered appears in the REMAIN column. The value in the REMAIN column is displayed in 250-character units.	
<PROGRAM> <CHR> <ST> <COMMENT>	<PROGRAM>	This item shows the Nos. of the machining programs already registered. The Nos. are displayed in order from the smallest No. The display range is from 1 to 99999999.
	<CHR>	This item shows the No. of characters registered in the memory for the corresponding machining program No.
	<ST>	This item shows the machining program status.
	<COMMENT>	An outline of the machining program functions, specifications, applications, etc., can be displayed as a comment in this item. The comment can be set with up to 18 alphanumeric and symbol characters.

5.7.11 Deleting Programs

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

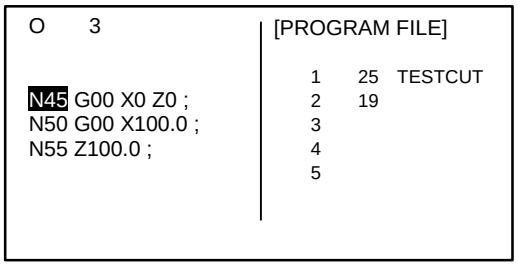


Set the No. of the program to be deleted.
(Ex.)
O (3) N ( ) – ( )



Press DELETE INS key.

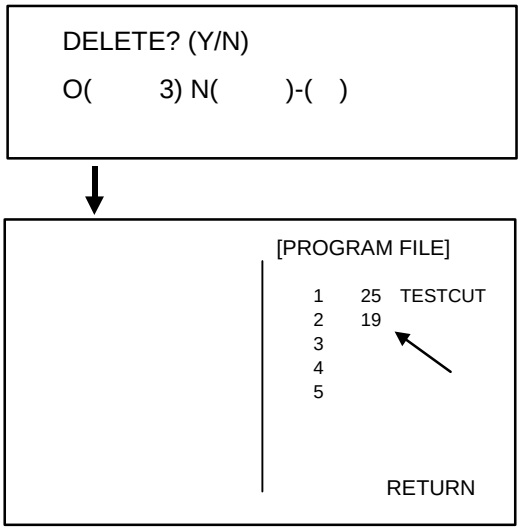
1) The list of programs is updated.
2) The message "DELETE? (Y/N)" appears.



Press Y Key.

The program is deleted when INPUT key is pressed.

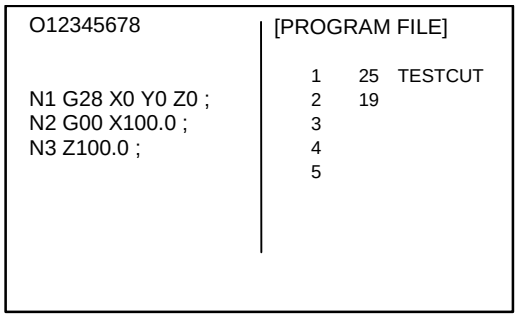
1) Deletion is started.
2) When the designated program No. is found, that program is deleted.
3) The screen returns to the WORD EDIT screen (1st menu) if the menu key RETURN is pressed.



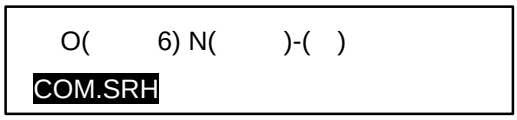
(Note 1) Even if there is data in the N ( ) – ( ) area during program deletion, it will be ignored.
(Note 2) If a sub-program is called from the main program currently being executed, deletion will still be possible as long as the sub-program is not executed. However, the operation of the program being executed cannot be assured. An error will occur if deletion is designated for a sub-program being executed.
(Note 3) Batch deletion of a setting area is not possible.
(Note 4) The list of programs is updated when a program is deleted, but the program display area (left side) is not.

5.7.12 Newly Creating Programs

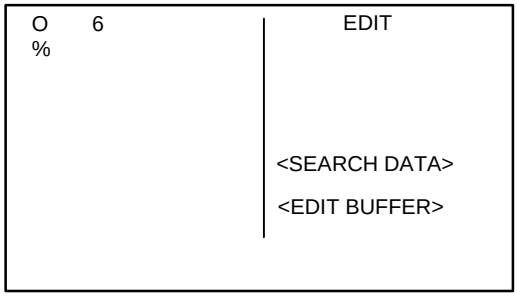
Programs to carry out automatic operation can be created and stored in the memory.



Set the No. of the program to be registered.
(Ex.)
O (6) N ( ) – ( )



Press SHIFT and DELETE INS keys.



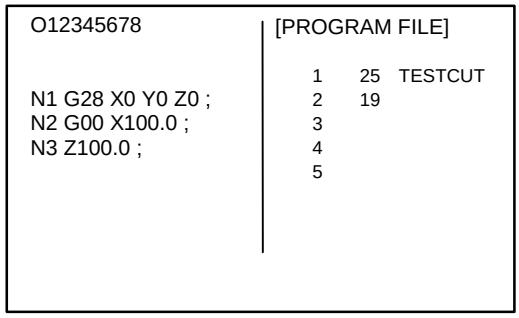
1) The list of programs is updated.
2) Program creation is enabled.
3) The screen returns to the WORD EDIT screen (1st menu) if the menu key RETURN is pressed.

(Note 1) Even if there is data in the N ( ) – ( ) area when newly creating the program, it will be ignored.
(Note 2) The SHIFT and DELETE INS keys are only valid when the COM.SRH menu is highlighted.
(Note 3) An operation search will result if the set program No. has already been registered.

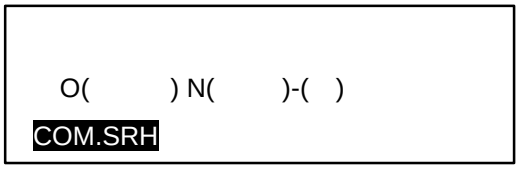
5.7.13 Operation Search

Calling a program

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

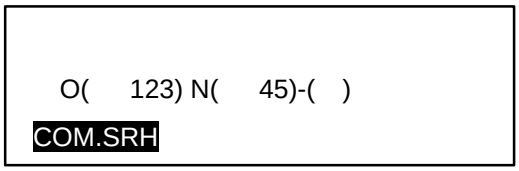


Press the menu key [COM-SRH].

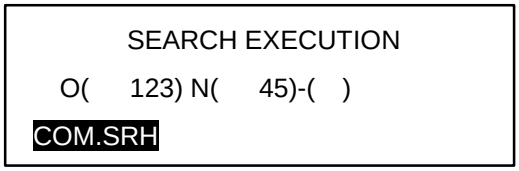


1) The setting area for "COM. SRH" is displayed.

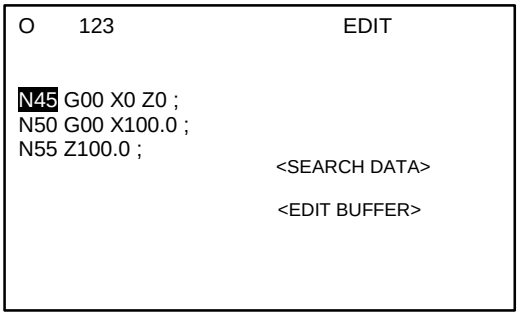
Set the No. of the program to be called.
Set the sequence No. and block No. if required.
(Ex.)
O (1 2 3) N (4 5) – ( )



Press [INPUT] key.



1) The search starts.
2) When the designated program Nos., sequence Nos., and block Nos. are found, that program is displayed, and the screen returns to the WORD EDIT screen (1st menu)



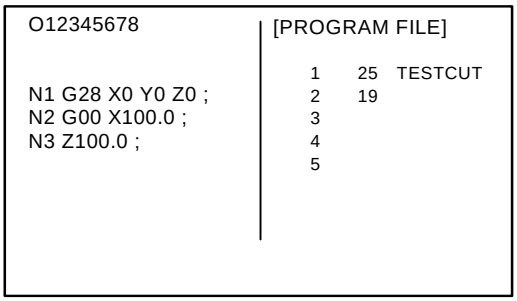
(Note 1) The search will not be executed when the [INPUT] key is pressed if only the N No. and B No. have been input.
Always set the program No. before searching.

(Note 2) A program deletion operation will be carried out if the program No. is input when the setting area is displayed and the [C.B CAN] key is pressed.

5.7.14 B. G Search

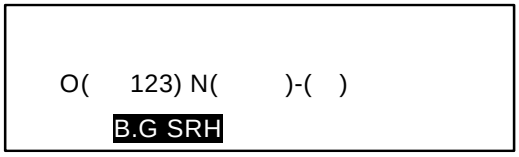
Calling the program

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



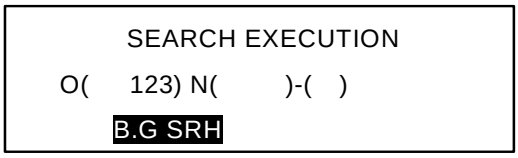
Set the No. of the program to be background edited.
(Ex.)
O (1 2 3) N ( ) – ( )

☞

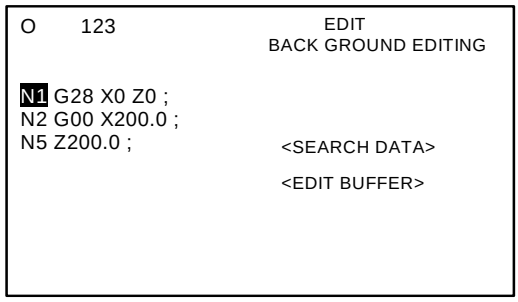


Press INPUT key.

☞



1) The message "SEARCH EXECUTION" appears during the search.
2) The designated program is displayed, and the screen returns to the WORD EDIT screen (1st screen).
If the designated program No. does not exist at this time, a new program creation operation will result.



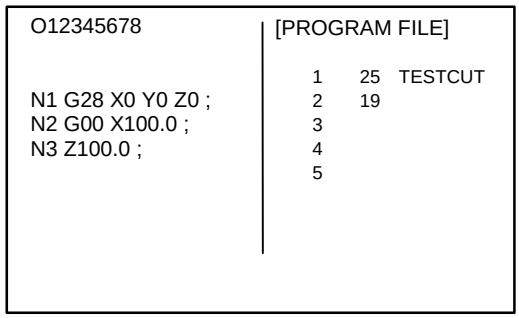
(Note 1) A program deletion operation will be carried out if the program No. is input when the setting area is displayed and the C.B CAN key is pressed.

5.7.15 B. G Quit

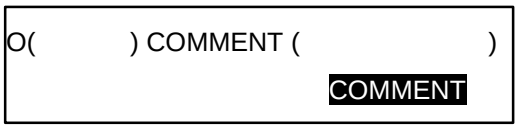
The "B.G-END" menu is used to quit the function after carrying out background editing.
If a running program is displayed on the EDIT screen, changeover to that program display will not occur unless the "B.G-END" menu is pressed and the background editing is canceled.
(The button does not have to be specially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

5.7.16 Comments

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



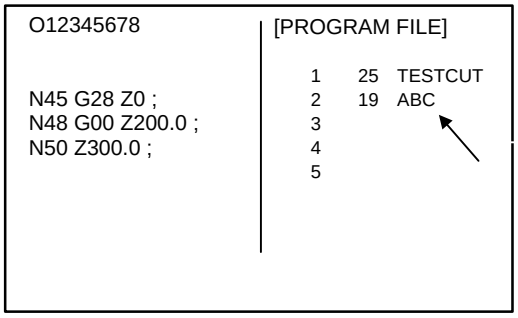
Press the menu key [COMMENT].



1) The "COMMENT" setting area is displayed.

Set the No. of the program to which the comment will be set.
(Ex.)
O (2) COMMENT (A B C)

Press [INPUT] key.

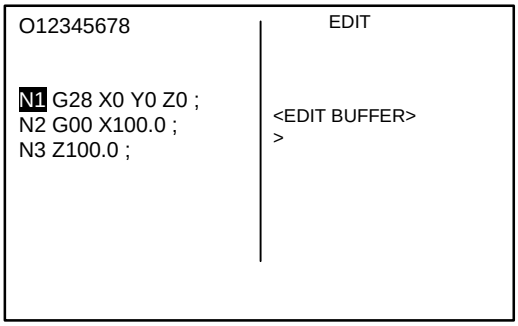


1) The comment is set to the designated program No.
When the designated program No. does not appear on the screen, the page with designated program No. will be displayed.
Press [INPUT] key again to set the comment.

(Note 1) A program deletion operation will be carried out if the program No. is input when the setting area is displayed and the [C.B CAN] key is pressed.

5.7.17 Setting the Program Operation Start Position

After setting a program for memory operation, the operation can be started from the designated block in the program by designating the starting block. The operation start position in normal word editing is set at the head block of that program. To change this start position, move the cursor to the head of the required starting position block, and press the [INPUT] key.

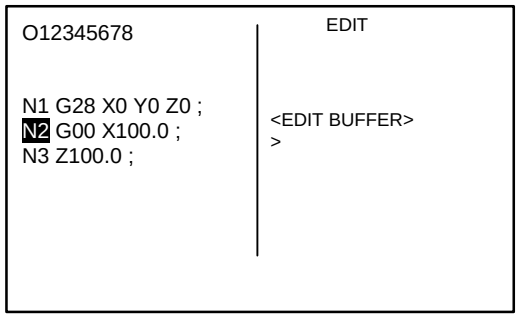


Move the cursor to the head of the required starting position block.

[↑] [↓]

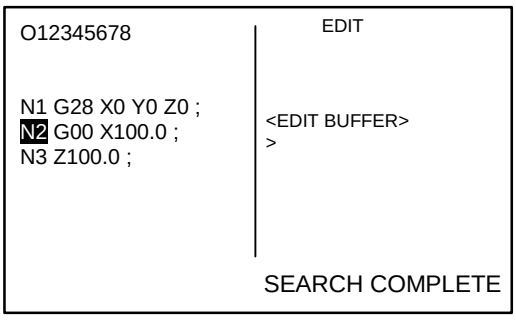
[NEXT PAGE] [PREVIOUS PAGE]

☞



Press the [INPUT] key.

☞



1) The "SEARCH COMPLETE" message appears, and operation from the designated block is enabled.

(Note 1) If the [INPUT] key is pressed when the cursor is at a position besides the head of the block, the top of the program will be searched. The program will also be displayed from the top.

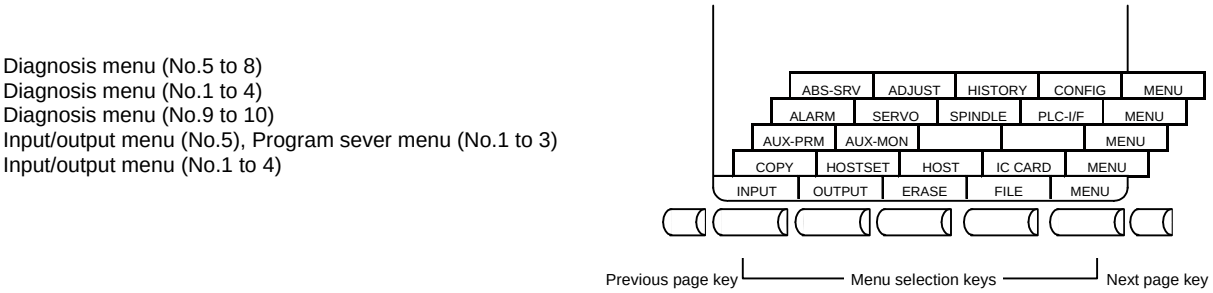
(Note 2) Up to 96 characters can be key-input in the EDIT BUFFER area.

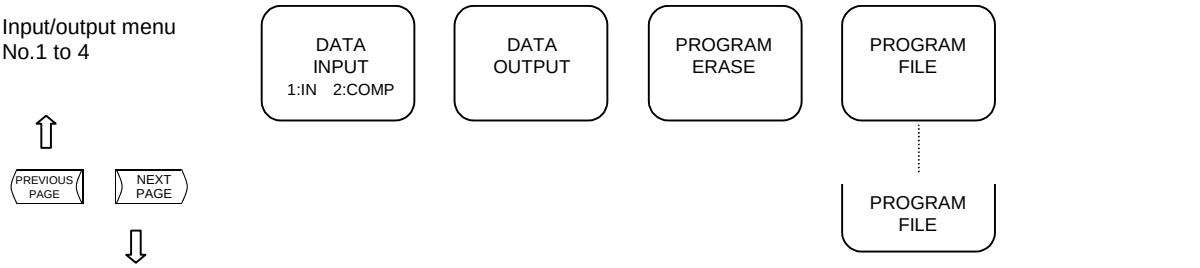
⚠ Caution

⚠ If a block in the program is set as 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 and commands.

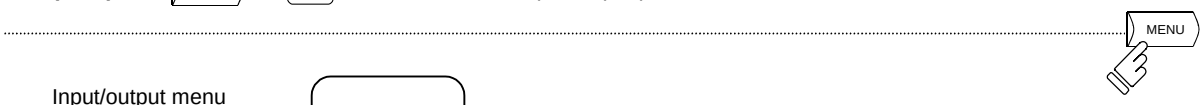
6. Data In/Out

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

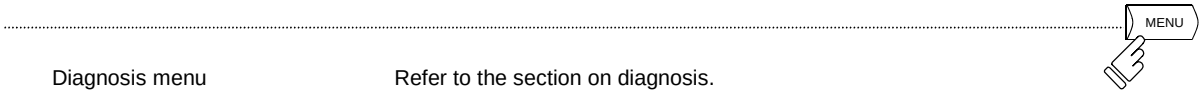




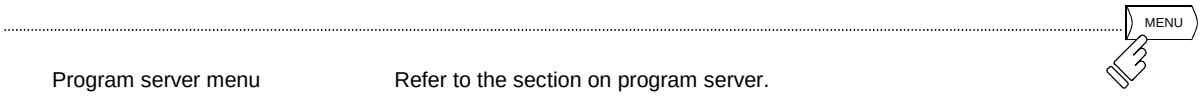
(Note) Use [I/O PARA] of [TOOL PARAM] to define the data input/output parameter.



Input/output menu No.5



Diagnosis menu — Refer to the section on diagnosis.



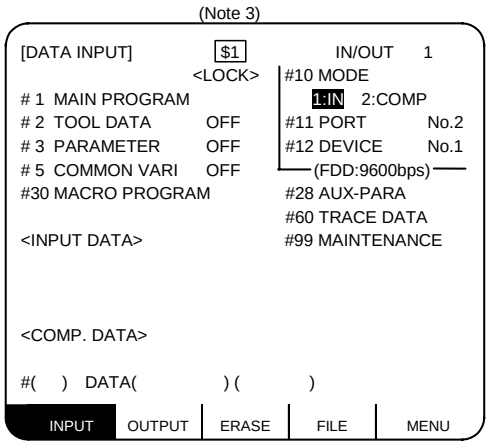
Program server menu — Refer to the section on program server.

(Note) When connected to MELSEC GPPQ/GPPW or GOT, the RS-232C communication port is used constantly, so the input/output device cannot be used.
Thus, if input/output operations are attempted when the parameters are set to MELSEC GPPQ/GPPW or GOT connection, the "E60 IOP ERR-2" error will occur.

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, common variables and history data.



#	Item	Explanation
1	MAIN PROGRAM **(Note 1)**	The types of data that can be input are indicated. (1) Used when inputting the machining program.
2	TOOL DATA	(2) Used when inputting the tool data.
3	PARAMETER	(3) Used when inputting the parameter data.
5	COMMON VARI	(4) Used to input a common variable.
28	AUX-PARA	(5) Used to input an auxiliary axis parameter data.
30	MACRO PROGRAM	(6) Used to input a macro program.
60	TRACE DATA	(7) Used to input history data.
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	PORT NO.	The I/O port number and device number required to input data are set.
12	DEVICE NO.	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 4 "Registration/Editing of Fixed Cycle Program".

(Note 2) The screen cannot be changed during data input.

(Note 3) When using the 2-part system, the system name of the currently selected system is displayed as $1 (system 1) and $2 (system 2). This is not displayed when using a 1-part system. (Only L system)

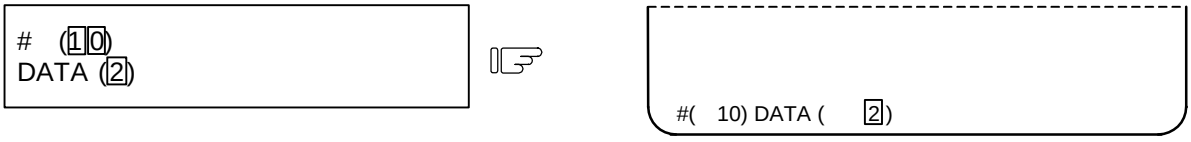
(Note 4) Whether to show or hide name of the selected system can be switched with the parameters.

#1050 MemPrg	#1286 ext22/bit0	Details
0, 2, 4, 6	-	The name of the selected system is not displayed.
1, 3, 5, 7	OFF	The name of the selected system is displayed.
	ON	The name of the selected system is not displayed.

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.



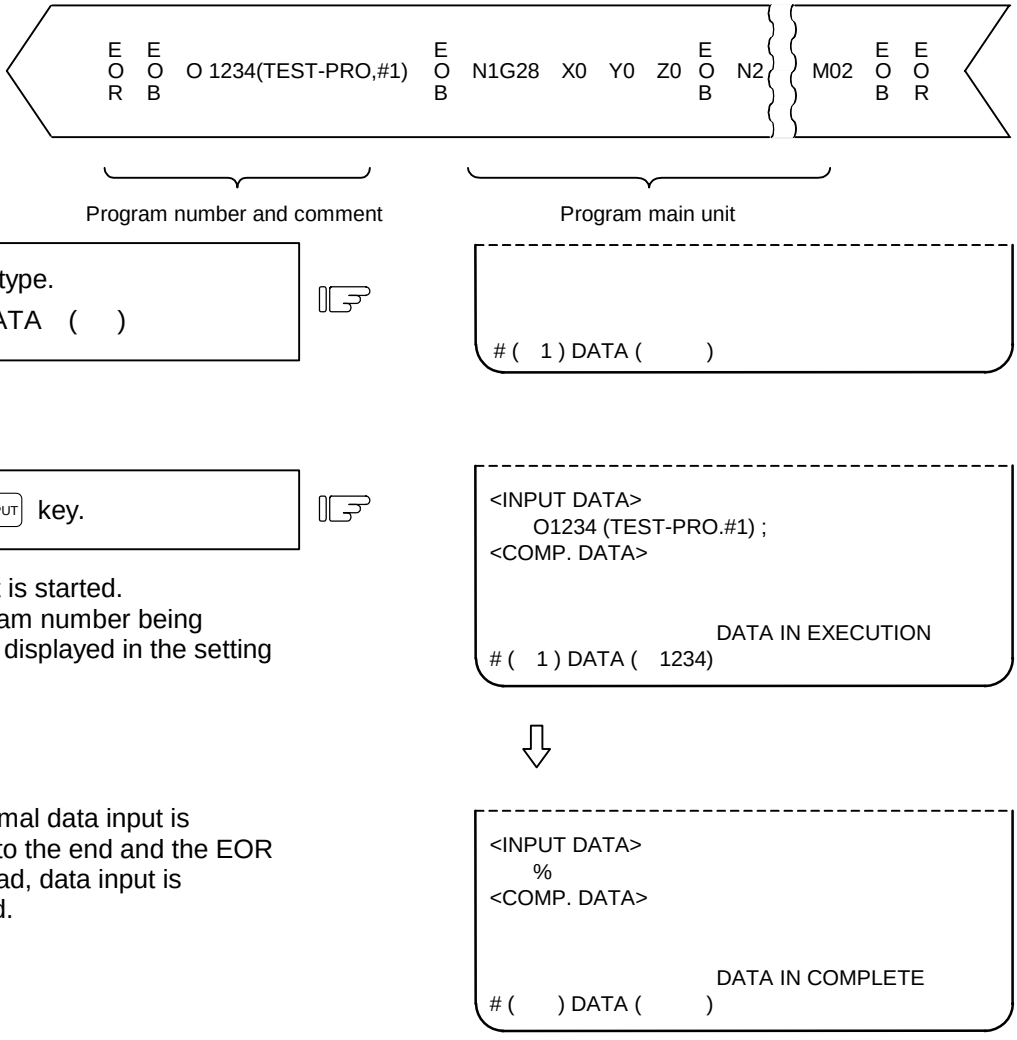
Press the [INPUT] key.

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.

6.1.2 Machining Program Input

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

(1) When inputting the program number output onto tape.
The program can be input simply by specifying machining program data type #1. If the program number is specified, the number on tape takes precedence over that number.



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

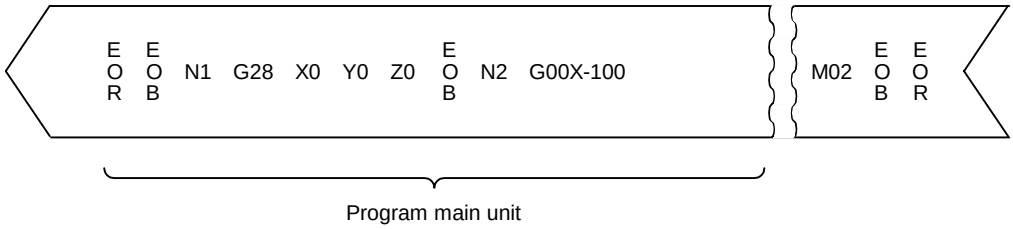
2) When normal data input is executed to the end and the EOR code is read, data input is terminated.

⚠ CAUTION

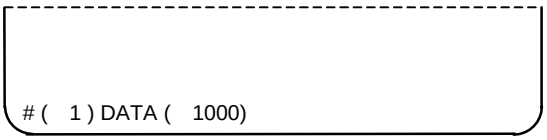
⚠ " ; " "EOB" and " % " "EOR" are explanatory notations. The actual codes are "Line feed" and "%" for ISO, and "End of Block" and "End of Record" for EIA.

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

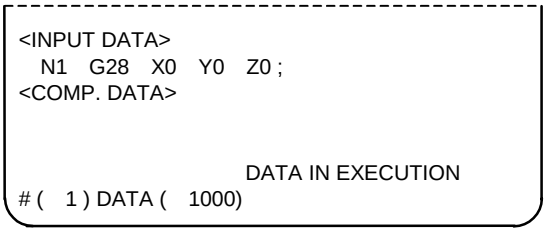
(2) When program number does not exist on tape
Specify machining program data type #1 and the program number to be registered.



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

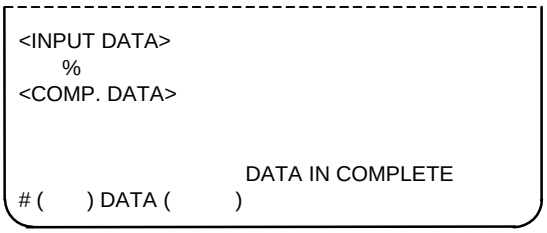


Press the INPUT key.



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

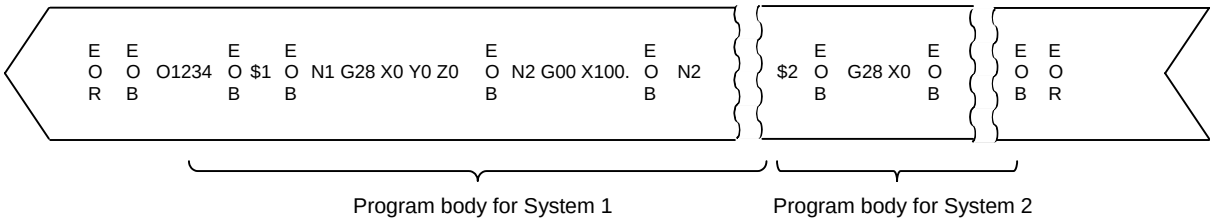
3) When normal data input is executed to the end and the EOR code is read, data input is terminated.



(3) Precaution for using 2-part system
When using the 2-part system, the machining program input operation can be switched with the parameters.

#1050 MemPrg	#1285 ext21/bit0	#1286 ext22/bit0	Details
0, 2, 4, 6	-		The input machining program is registered in the memory common for the systems.
1, 3, 5, 7	OFF	OFF	The input machining program is registered in the memory for the selected systems.
	OFF	ON	The input machining program is registered in the memory for the selected systems. If the system is not selected, an empty (only EOR [%]) machining program is registered in the memory.
	ON	OFF	The input machining program is separated into machining programs for each system at the $ mark, and is registered in the memory for each system.
	ON	ON	The input machining program is separated into machining programs for each system at the $ mark, and is registered in the memory for each system. If the system does not have a $ mark, an empty (only EOR [%]) machining program is registered in the memory.

(Example) When parameter "#1050 MemPrg" is set to 1, 3, 5 or 7, and "#1286 ext22 bit0" is set to 1, the file (tape) delimited into systems with the $ mark is separated and input for each system.



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 ( )

```
# ( 2 ) DATA (        )
```

Press the INPUT key.

```
<INPUT DATA>
    G10  L10  P···;
<COMP. DATA>

                        DATA IN EXECUTION
# ( 2 ) DATA (        )
```

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

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

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

(2) When an error occurs during offset tape input:
If an error occurs during offset tape 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.

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 ( )


```
# ( 3 ) DATA (        )
```

Press the INPUT key.


```
<INPUT DATA>
    P5  N1    L···;
<COMP. DATA>


                        DATA IN EXECUTION
# ( 3 ) DATA (        )
```

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.

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


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

(2) Input parameter skip operations

1) Skipping N No. data not found in the specifications
   When "S" is set in the second setting area during data input or compare, and the INPUT key is pressed, an error will not occur even if parameter No. (N No.) data not found in the specifications is input. Data input and compare of that number will be skipped.
   (With normal input and compare, an error will occur if N No. data not found in the specifications is input, and data input/compare will be stopped.)

2) Skipping axis data not found in the specifications
   When "S" is set in the second setting area during data input or compare, and the INPUT key is pressed, an error will not occur even if data with a different number of axes is input. Axis parameters not found in the specifications will be skipped during the input and compare.

3) Skipping spindle data not found in the specifications
   When "S" is set in the second setting area during data input or compare, and the INPUT key is pressed, an error will not occur even if data with a different number of spindles is input. Spindle parameters not found in the specifications will be skipped during the input and compare.

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 ( )

```
# ( 5 ) DATA (      )
```

Press the [INPUT] key.

```
<INPUT DATA>

<COMP. DATA>


                    DATA IN EXECUTION
# ( 5 ) DATA (      )
```

1) Read of the data 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 tape end code % (EOR) is read, data input ends with message "DATA IN COMPLETE" displayed.

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


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

6.1.6 Inputting History Data

History data that has been output by history data output operation can be input.
History data input operation is performed with DATA IN/OUT 1 screen.

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

(1) Inputting history data

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


```
# ( 60 ) DATA (          )
```

Press the [INPUT] key.


```
<INPUT DATA>

<COMP. DATA>


                        DATA IN EXECUTION
# ( 60 ) DATA (          )
```

1) Read of the data 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 tape end code % (EOR) is read, data input ends with message "DATA IN COMPLETE" displayed.

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


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

6.1.7 Inputting Waveform Data

Waveform data that has been output by waveform data output operation can be input.

(1) Inputting waveform data

Set the data type to waveform data.
(22) DATA ( )

```
# ( 22 ) DATA (         )
```

Press the [INPUT] key.

```
<INPUT DATA>

<COMP. DATA>


                          DATA IN EXECUTION
# ( 22 ) DATA (         )
```

1) Read of the data 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 tape end code % (EOR) is read, data input ends with message "DATA IN COMPLETE" displayed.

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


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

(Note 1) If input operations are attempted while the waveform is displayed, the operation message "V-ANALYZER EXEC." will appear, and the waveform display data will not be input.

(Note 2) If input operations are attempted while the Visual analyzing function is invalid, the error "E01 SETTING ERROR" will occur.

6.1.8 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 ( )


```
# ( 28 ) DATA (        )
```

Press the [INPUT] key.


```
<INPUT DATA>
<COMP. DATA>
                          DATA IN EXECUTION
# ( 28 ) DATA (        )
```

1) Read of the data 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 tape end code % (EOR) is read, data input ends with message "DATA IN COMPLETE" displayed.

```
<INPUT DATA>
      %
<COMP. DATA>
                          DATA IN COMPLETE
# (      ) DATA (        )
```

(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.

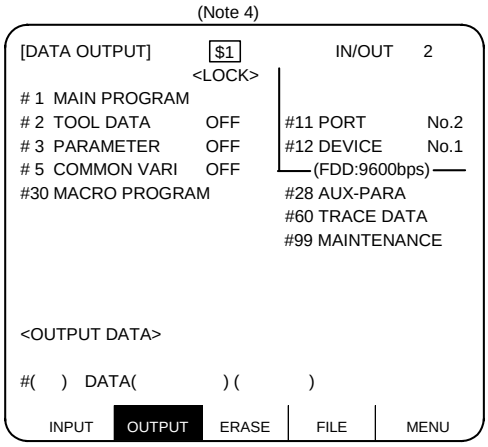
#7 ATU in input data	#7 ATU set in NC	Auto-tuning parameter input
Auto-tuned (0 or 1)	Auto-tuned (0 or 1)	Not input
Not auto-tuned (2)	Auto-tuned (0 or 1)	Input **(Note 3)**
Auto-tuned (0 or 1)	Not auto-tuned (2)	Input **(Note 3)**
Not auto-tuned (2)	Not auto-tuned (2)	Input **(Note 3)**

(Note 3) Which parameters can be input depends on the setting of "#7 ATU" parameter. Refer to "6.2.7 Outputting Auxiliary Axis Parameter Data" for details on auto-tuning target parameters.

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	MAIN PROGRAM **(Note 1)**	The types of data that can be output are as follows. (1) Used to output a machining program.
2	TOOL DATA	(2) Used to output tool data.
3	PARAMETER	(3) Used to output parameter data.
5	COMMON VARIABLE	(4) Used to output a common variable.
28	AUX-PARA	(5) Used to output an auxiliary axis parameter data.
30	MACRO PROGRAM	(6) Used to output a macro program.
60	TRACE DATA	(7) Used to output history data.
11	PORT 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.
12	DEVICE NO.	

(Note 1) To output a fixed cycle program, set the parameter.
Refer to the Appendix "Registration and editing of 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" in Section 6.7.

(Note 3) The screen cannot be changed during data output.

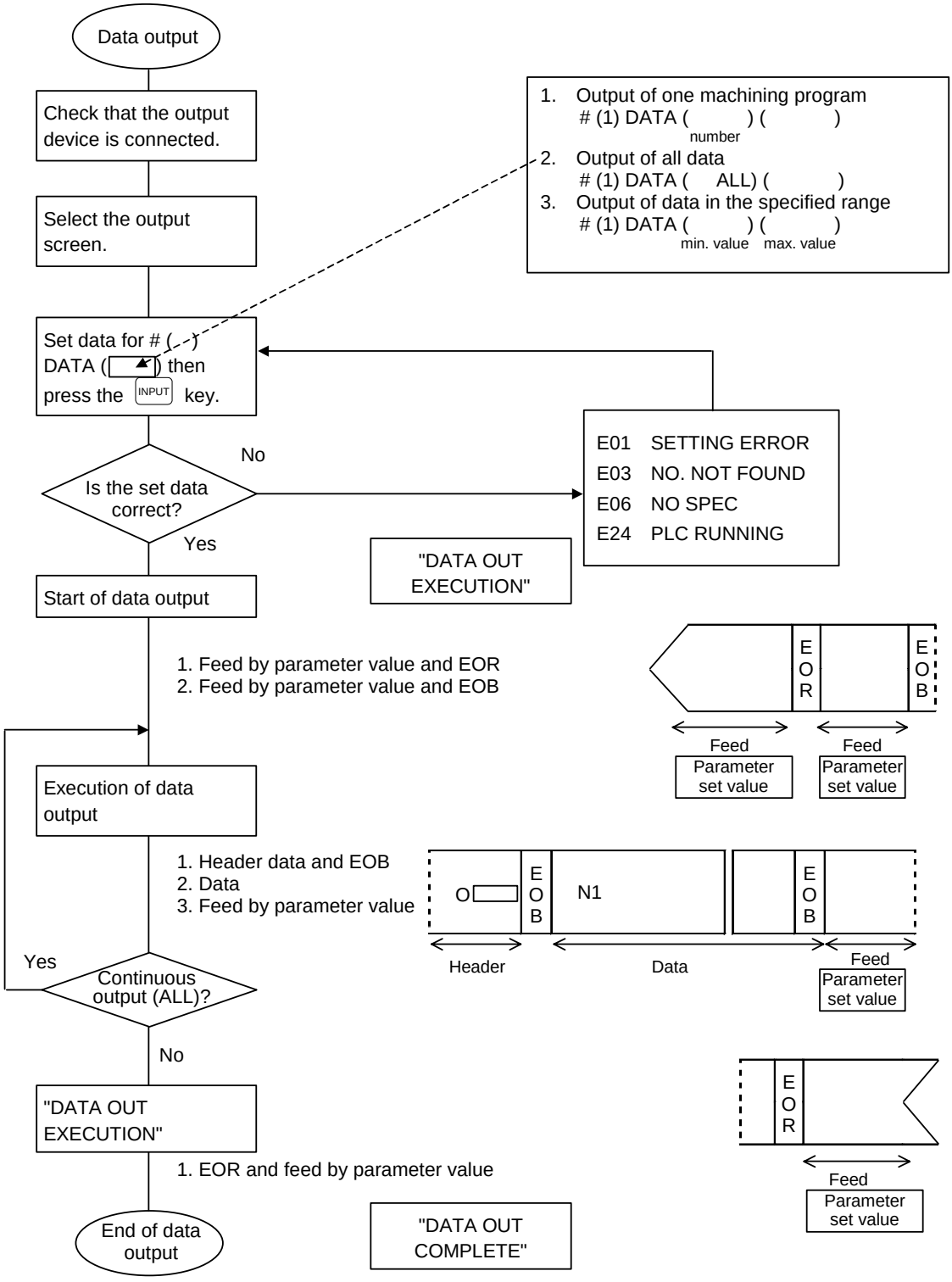
Output inhibit condition / Output method	Data protection key on Machining programs, tool data, parameters	Edit lock B on Machining programs 8000 to 9999	Edit lock C on Machining programs 9000 to 9999
Specifying individual machining programs	No data is output.	Machining programs O8000 to 9999 are not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

(Note 4) When using the 2-part system, the system name of the currently selected system is displayed as $1 (system 1) and $2 (system 2). This is not displayed when using a 1-part system. (Only L system)

(Note 5) Whether to show or hide name of the selected system can be switched with the parameters.

#1050 MemPrg	#1286 ext22/bit0	Details
0, 2, 4, 6	-	The name of the selected system is not displayed.
1, 3, 5, 7	OFF	The name of the selected system is displayed.
	ON	The name of the selected system is not displayed.

Operation procedure for outputting data



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 (1 0 0 0)

(1) DATA (1000) ( )

Press the INPUT key.

<OUTPUT DATA>
O1000(TEST);
DATA OUT EXECUTION
(1) DATA (1000) ( )

1) Data output is started.

<OUTPUT DATA>
%
DATA OUT COMPLETE
( ) DATA ( ) ( )

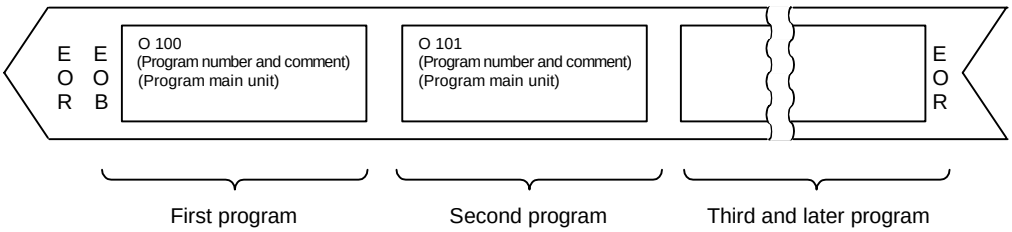
(2) When all machining programs are output
To output all machining programs registered in memory in batch, specify machining program data type #1 and "ALL" in DATA ( ).

Set 1 in # ( ) and "ALL" in DATA ( ).
(1) DATA (A L L)

(1) DATA (ALL) ( )

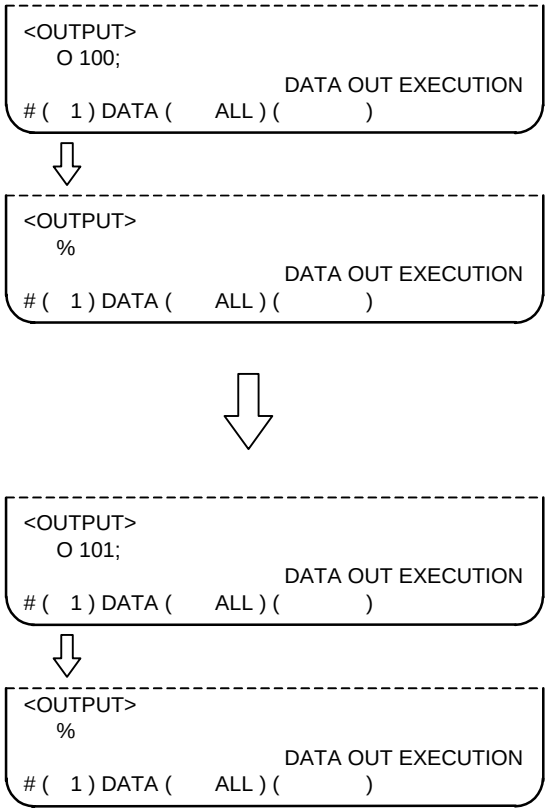
Press the INPUT key.

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)

<OUTPUT DATA>
(1) DATA (9000) (9999)

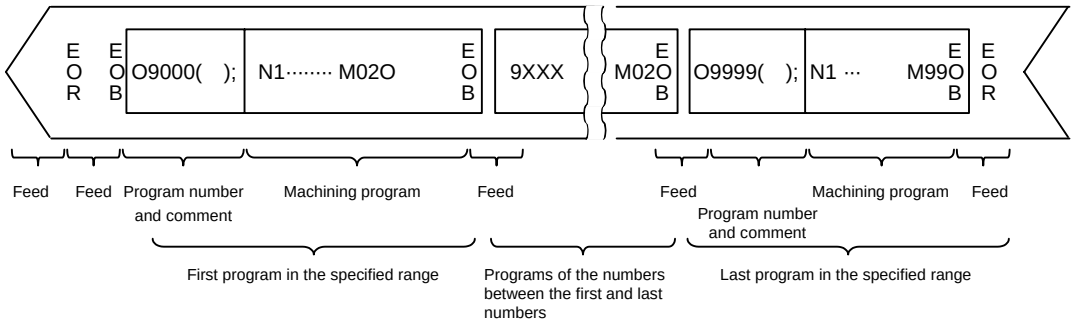
Press the INPUT key.

<OUTPUT DATA>
O 9000(M150.SUBPRO);
DATA OUT EXECUTION
(1) DATA (9000) (9999)

1) Data output starts.
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 tape format is as follows:

The machining programs are output sequentially.

<OUTPUT DATA>
%
DATA OUT COMPLETE
( ) DATA ( ) ( )

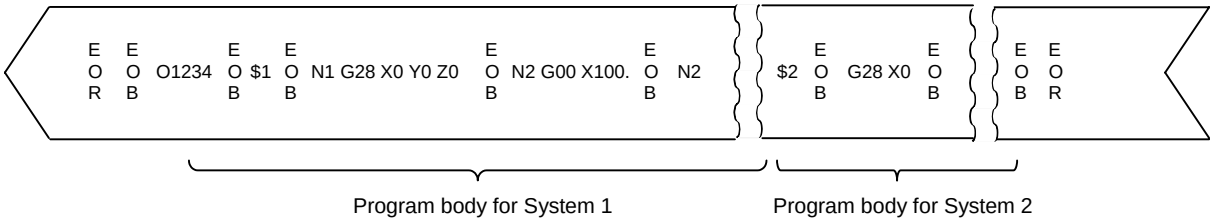


(Note) 1. If the number specified as the smallest number is not found, output starts with the machining program with the number nearest to that number.
Likewise, if the number specified as the largest number is not found, output ends with the machining program with the number nearest to that largest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

(4) For 2-part system
When using the 2-part system, the machining program output operation can be switched with the parameters.

#1050 MemPrg	#1286 ext22/bit0	Details
0, 2, 4, 6		The machining program registered in the memory common for the systems is output.
1, 3, 5, 7	OFF	The machining program registered in the memory for the selected system is output.
	ON	The machining program registered in the memory for each system is output as one file (tape) delimited with the $ mark.

(Example) When parameter "#1050 MemPrg" is set to 1, 3, 5 or 7, and "#1286 ext22 bit0" is set to 1, the machining programs registered in each system memory are output as one file (tape) delimited with the $ mark.



6.2.2 Outputting Tool Offset Data

Tool offset data and workpiece coordinate 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 ( )

(2) DATA ( )

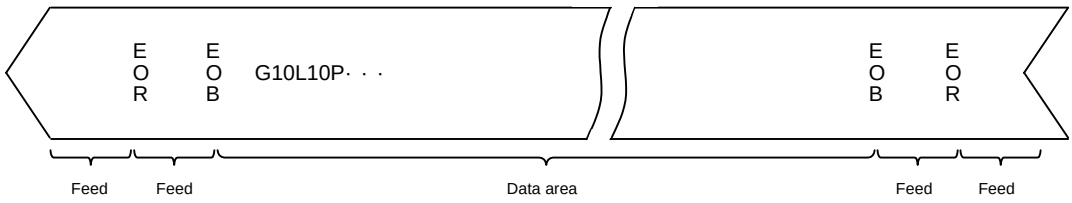
Press the INPUT key.

<OUTPUT DATA>
G10 L10 P···;
DATA OUT EXECUTION
(2) DATA ( )

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

<OUTPUT DATA>
%
DATA OUT COMPLETE
( ) DATA ( )

(2) Output tape 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.

(Note) For the multiple system, data will be output following the parameters below.

#1286 ext22/bit1
0: The data for each system will be output.
1: The data for the system selected with the system selection switch will be output.

#1051 MemTol
0: The output will follow ext22 bit1.
1: The common tool offset data specifications will be used between the systems, so the same output as the single system will be used.

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. This format allows the details of the parameters to be read when the parameter data is output to a printer, etc.
The output tape length varies with the number of axes and parameter numeric. For the three-axis specification, this length is 40 to 50m. The following data is output:

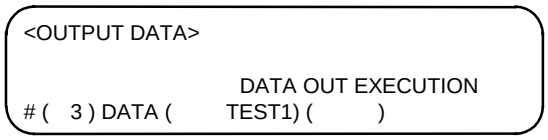
- User parameters (Machining parameters, control parameters, axis parameters)
- Data input/output parameters (I/O BASE PARAM, I/O DEVICE PARAM)
- All setup parameters
- Internal parameter data (absolute position internal data)

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

(1) Outputting parameter data

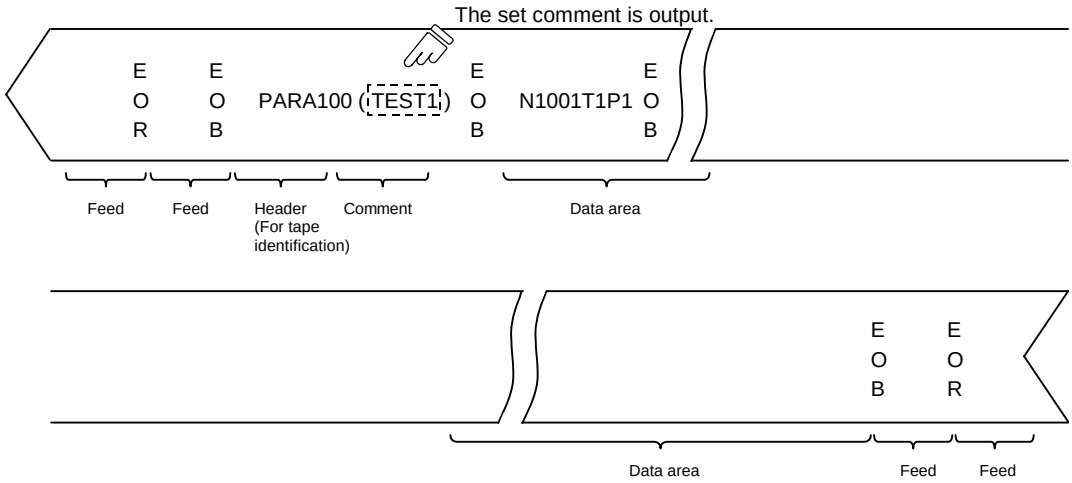
Set the data type to parameter.
([3]) DATA ([T][E][S][T][1])

Press the [INPUT] key.



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

(2) Output tape format



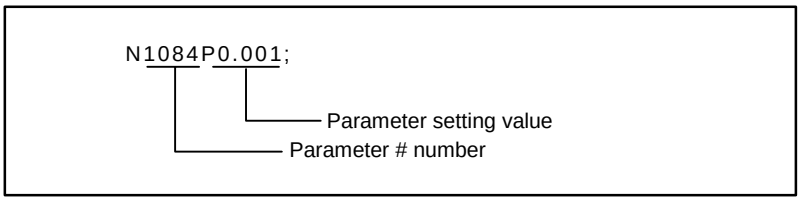
(3) Data format

The data format is as follows:

Address	Definition	Details
N	Parameter number	The parameter # number is shown with the value following N.
A	Axis number	For axis data, the axis number is shown with the value following A. The first axis will be AI.
T	Axis system number	For data per system, the system number is shown with the value following T. (1st system: T1, 2nd system: T2, PLC axis: T3)
P	Parameter data	The parameter data is shown with the value following P.

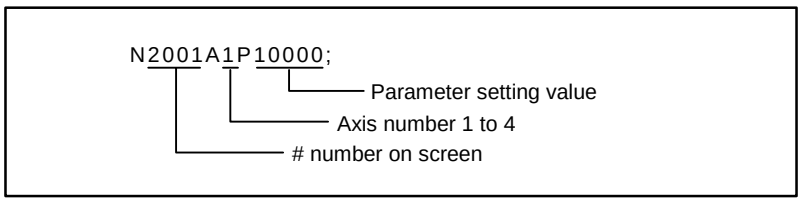
The following types of data format are used according to parameter type and display method. (The address order in one block must use the following format.)

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



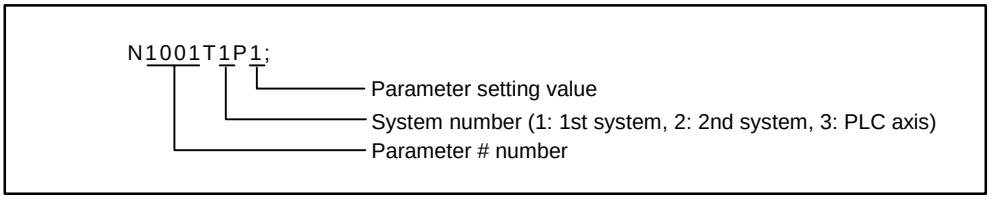
(1) When multiple axes are displayed on one screen

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

[Output example]

```
      :
N2001A1P120000 ;
N2002A1P4000 ;        } Axis 1 data
N2003A1P21 ;
      :
N2001A2P12000 ;
N2002A2P4000 ;        } Axis 2 data
N2003A2P21 ;
      :
```

3) System parameter



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

[Output example]
:
N1001T1P1 ;
N1001T2P1 ;
N1001T3P0 ;
N1002T1P2 ;
N1002T2P1 ;
N1002T3P0 ;
:

(2) The parameter data per system displayed by changing over the system ($ key) is output per system screen.

[Output example]
:
N8001T1P99 ;
N8002T1P0 ;
N8003T1P10000 ;
:
N8001T2P30 ;
N8002T2P1 ;
N8003T2P20000 ;
:

6.2.4 Outputting Common Variable Data

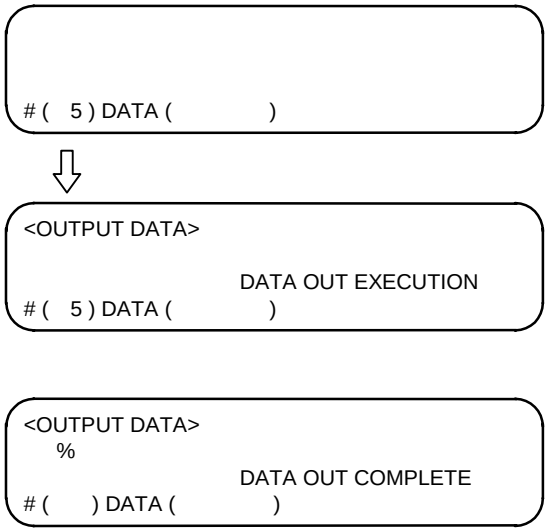
Common variable data can be output. The output operation can be performed even during automatic operation.

(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.



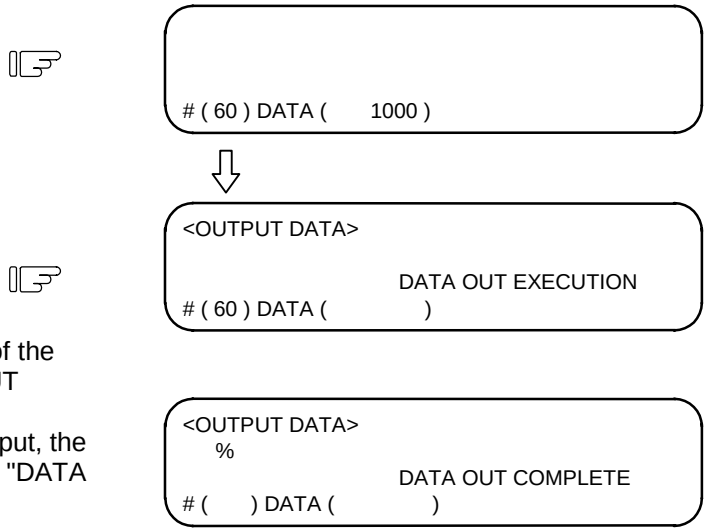
(Note) For the multiple system, data will be output following the parameters below.
#1303 V1comN (#100 to set number) system common variables
#1304 V0comN (#500 to set number) system common variables
The number of data items designated in the parameter will be output in the same manner as the single system. The other common variables will be output for each system.

6.2.5 Outputting History Data

History data can be output. The output operation can be performed even during automatic operation. The DATA IN/OUT 2 screen is used to output history data.

(1) History data output operation

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

6.2.6 Outputting Waveform Data

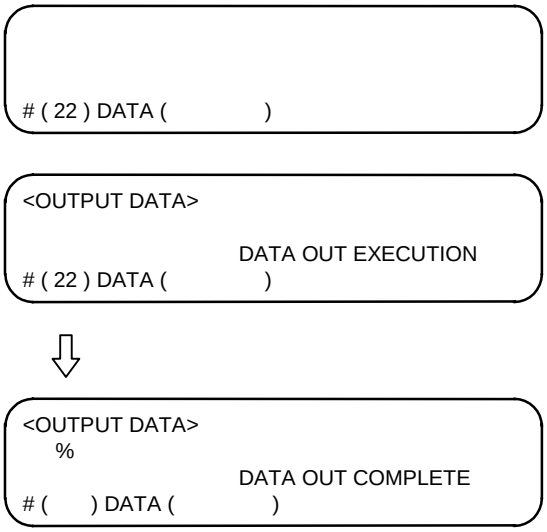
Waveform data can be output.

(1) Waveform data output operation

Set the data type to the waveform data.
(22) 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 1) If output operations are attempted while the waveform is displayed, the operation message "V-ANALYZER EXEC." will appear, and the waveform display data will not be output.

(Note 2) If output operations are attempted while the Visual analyzing function is invalid, the error "E01 SETTING ERROR" will occur.

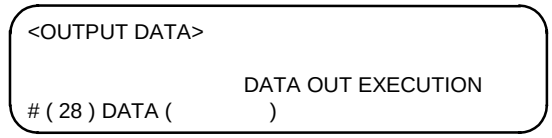
6.2.7 Outputting Auxiliary Axis Parameter Data

Auxiliary axis parameter data can be output.

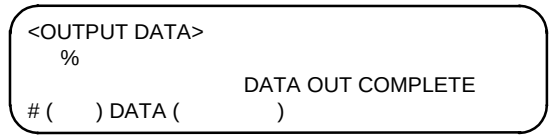
(1) Auxiliary axis parameter data output operation

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.

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	

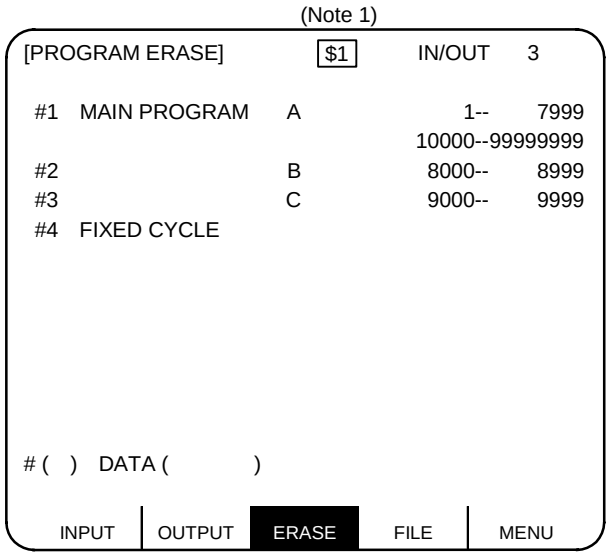
No.	Symbol name	N No.	Remarks
174	Aspeed2	50174	
175	Mspeed4	50175	
176	time4.1	50176	
177	time4.2	50177	
178	TL4	50178	
179	OD4	50179	
180	just4	50180	
181	near4	50181	
190	stpos2	50190	
191	stpos3	50191	
192	stpos4	50192	
193	stpos5	50193	
194	stpos6	50194	
195	stpos7	50195	
196	stpos8	50196	
197	stpos9	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.

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 maker.	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. Refer to the Appendix "Registration and editing of fixed cycle programs". When NC is operated without setting the parameter, SETTING ERROR occurs.			

(Note 1) When using the 2-part system, the system name of the currently selected system is displayed as $1 (system 1) and $2 (system 2). This is not displayed when using a 1-part system. (Only L system)

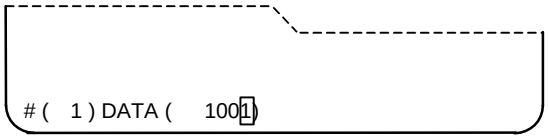
(Note 2) Whether to show or hide name of the selected system can be switched with the parameters.

#1050 MemPrg	**#1285 ext21/bit0**	**Details**
0, 2, 4, 6	-	The name of the selected system is not displayed.
1, 3, 5, 7	OFF	The name of the selected system is displayed.
	ON	The name of the selected system is not displayed.

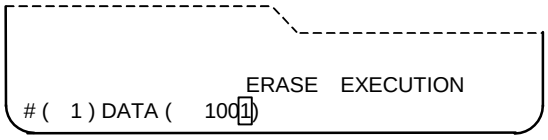
(1) To erase one machining program

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

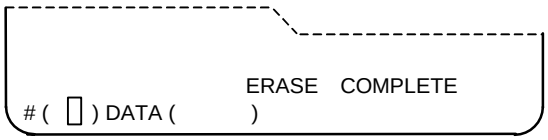
Set the data type and program number of the machining program to be erased.
(Example)
To erase machining program O1001
([1])
DATA ([1][0][0][1])



Press the [INPUT] key.



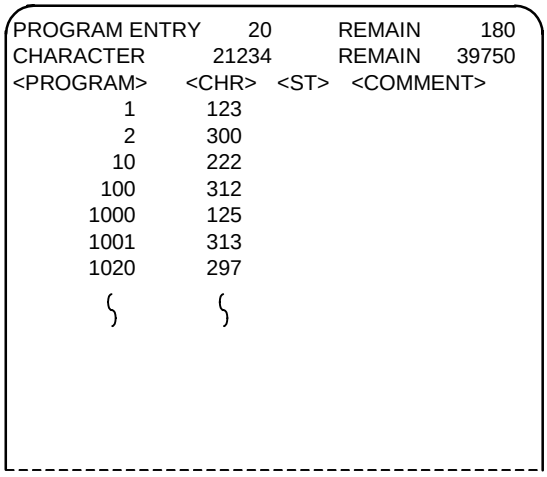
1) Program erase is started and the message "ERASE EXECUTION" is displayed in the message field. Usually program erasure is terminated in a moment.
2) When the erase is complete, the message "ERASE COMPLETE" is displayed in the message field and the data setting area becomes blank.



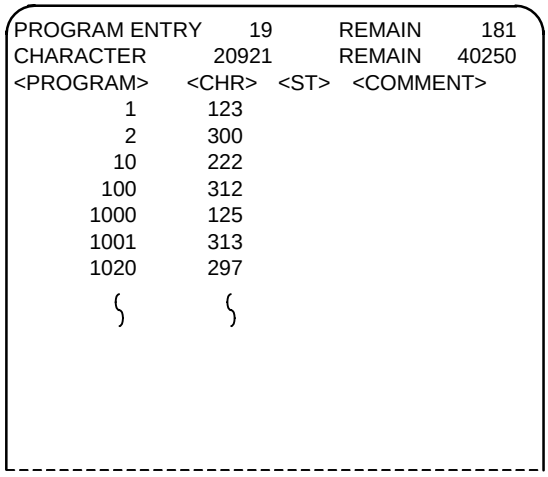
PROGRAM FILE before erase execution

Erase execution

PROGRAM FILE after erase execution



PROGRAM ENTRY	20		REMAIN	180
CHARACTER	21234		REMAIN	39750
<PROGRAM>	<CHR>	<ST>	<COMMENT>	
1	123			
2	300			
10	222			
100	312			
1000	125			
1001	313			
1020	297			
(	(			
)	)			



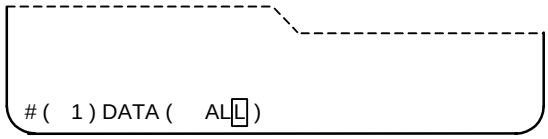
PROGRAM ENTRY	19		REMAIN	181
CHARACTER	20921		REMAIN	40250
<PROGRAM>	<CHR>	<ST>	<COMMENT>	
1	123			
2	300			
10	222			
100	312			
1000	125			
1001	313			
1020	297			
(	(			
)	)			

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.

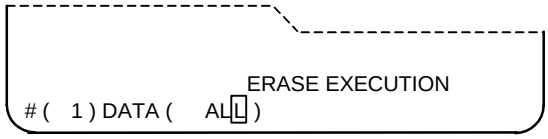
(2) To erase machining program group

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:

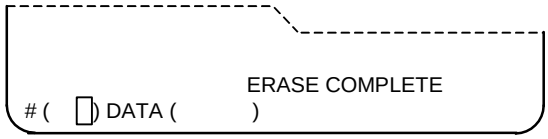
Set the data type to be erased and "ALL" in DATA ( ).
(Example)
To erase the entire machining program A group
(1) DATA (A L L)



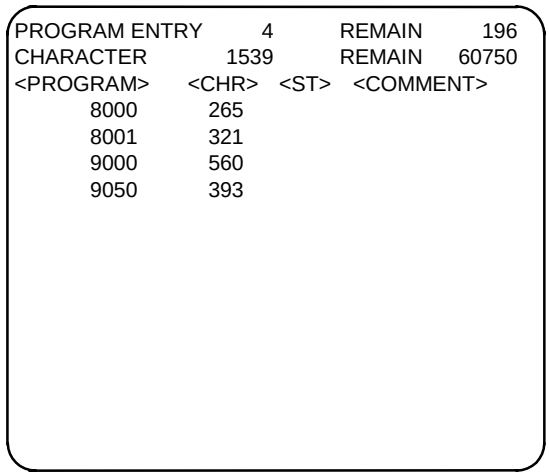
Press the INPUT key.



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.



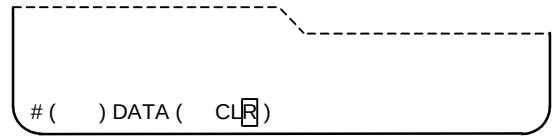
Check <REMAIN>, etc. on the PROGRAM FILE screen.



(3) To erase all machining programs registered in memory

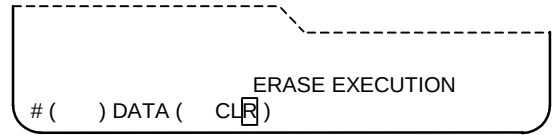
To erase all of machining programs 1~99999999 registered in memory, perform the following:

Set "CLR" in DATA ( ).
( )
DATA (C L R)

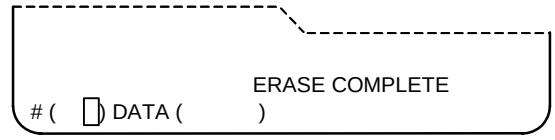


1) No value needs to be entered in # ( ).

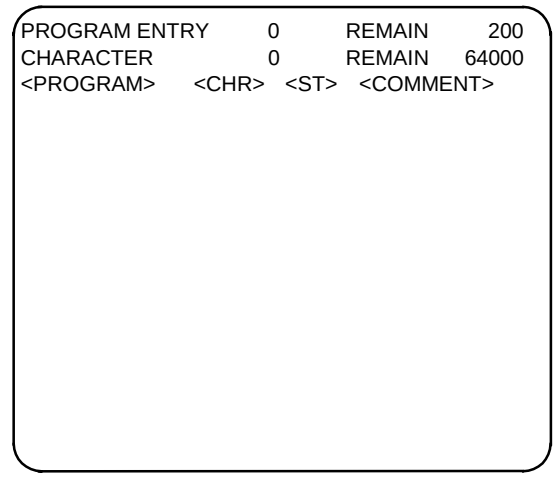
Press the INPUT key.



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.

(4) Precautions for 2-part system

When using the 2-part system, the machining program erase operation can be switched with the parameters.

#1050 MemPrg	#1285 ext21/bit0	Details
0, 2, 4, 6	-	Of the machining programs registered in the memory common for the systems, the designated machining program is erased.
1, 3, 5, 7	OFF	Of the machining programs registered in the memory for the selected systems, the designated machining program is erased.
	ON	Of all the machining programs registered in the memory for each system, the designated machining program is erased.

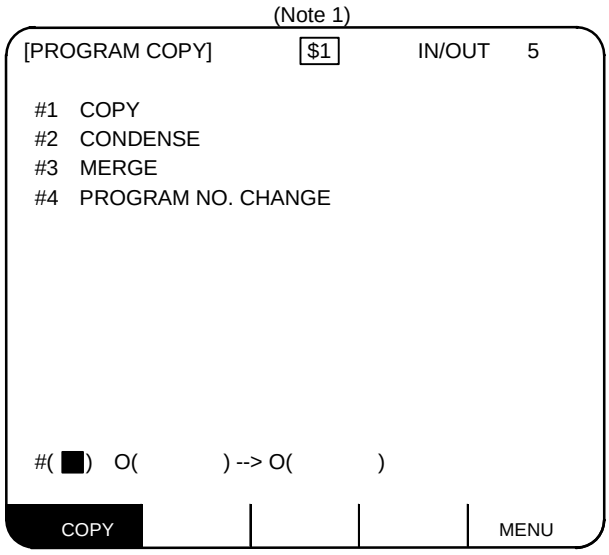
When using the 2-part system and the same number batch generation for all system programs is valid, the program is erased in the following manner according to the presence of programs in each system.

Presence of program		Operation
System 1	System 2	
Yes	Yes	The System 1 and System 2 programs are erased simultaneously.
Yes	No	Only the System 1 program is erased.
No	Yes	Only the System 2 program is erased.
No	No	An error (E14) occurs.

6.4 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.



(Note 1) When using the 2-part system, the system name of the currently selected system is displayed as $1 (system 1) and $2 (system 2). This is not displayed when using a 1-part system. (Only L system)

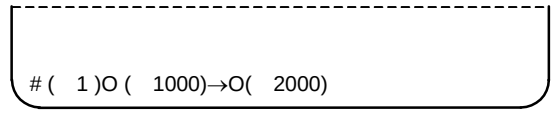
(Note 2) Whether to show or hide name of the selected system can be switched with the parameters.

#1050 MemPrg	**#1286 ext22/bit0**	**Details**
0, 2, 4, 6	-	The name of the selected system is not displayed.
1, 3, 5, 7	OFF	The name of the selected system is displayed.
	ON	The name of the selected system is not displayed.

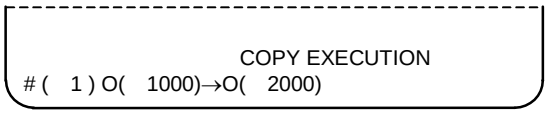
6.4.1 Machining Program Copy

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.

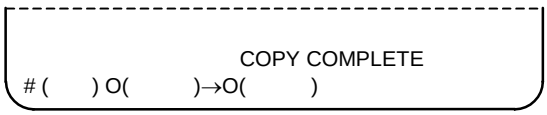
(Example)
To copy program O1000 into O2000
(1)
O (1 0 0 0) → O (2 0 0 0)



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.



(Note 1) When using the 2-part system, the machining program copy operation can be switched with the parameters.

#1050 MemPrg	#1285 ext21/bit0	Details
0, 2, 4, 6	-	Of the machining programs registered in the memory common for the systems, the designated machining program is copied to the memory common for the systems with a different machining program number.
1, 3, 5, 7	OFF	Of the machining programs registered in the memory for the selected systems, the designated machining program is copied to the memory for the selected system as a different machining program number.
	ON	Of all the machining programs registered in the memory for each system, the designated machining program is copied to the memory for each system with a different machining program number.

(Note 2) When using the 2-part system and the same number batch generation for all system programs is valid, an error (E64) will occur if the copy destination program is found in any of the systems.

(Note 3) When using the 2-part system and the same number batch generation for all system programs is valid, the program will be copied in the following manner according to the presence of the copy source program in each system.

Presence of program		**Operation**
System 1	**System 2**	
Yes	Yes	The copy source program for System 1 is copied to the copy destination program for System 1. The copy source program for System 2 is copied to the copy destination program for System 2.
Yes	No	The copy source program for System 1 is copied to the copy destination program for System 1. The System 2 copy destination program is newly created. (An empty program is created.)
No	Yes	The System 1 copy destination program is newly created. (An empty program is created.) The copy source program for System 2 is copied to the copy destination program for System 2.
No	No	An error (E14) occurs.

6.4.2 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 ( )

☞

(2) O (ALL)→O( )

Press the INPUT key.

☞

CONDENSE EXECUTIN
(2) O (ALL)→O( )

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.

⇩

CONDENSE COMPLETE
( ) O ( )→O( )

(Note 1) When using the 2-part system, the machining program condense operation can be switched with the parameters.

#1050 MemPrg	#1285 ext21/bit0	Details
0, 2, 4, 6	-	Of the machining programs registered in the memory common for the systems, the designated machining program is condensed.
1, 3, 5, 7	OFF	Of the machining programs registered in the memory for the selected systems, the designated machining program is condensed.
	ON	Of the machining programs registered in the memory for each system, the designated machining program is condensed.

(Note 2) When using the 2-part system and the same number batch generation for all system programs is valid, an error (E14) will occur if the all system condense program is not found.

(Note 3) When using the 2-part system and the same number batch generation for all system programs is valid, the programs cannot be condense in a system that does not have the condense program.

(Note 4) When using the 2-part system, and ALL is set, all machining programs registered in the memory for all systems will be condensed.

6.4.3 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)

☞

```
# (  3 ) O(  1001)→O(  1000)
```

Press the INPUT key. ☞

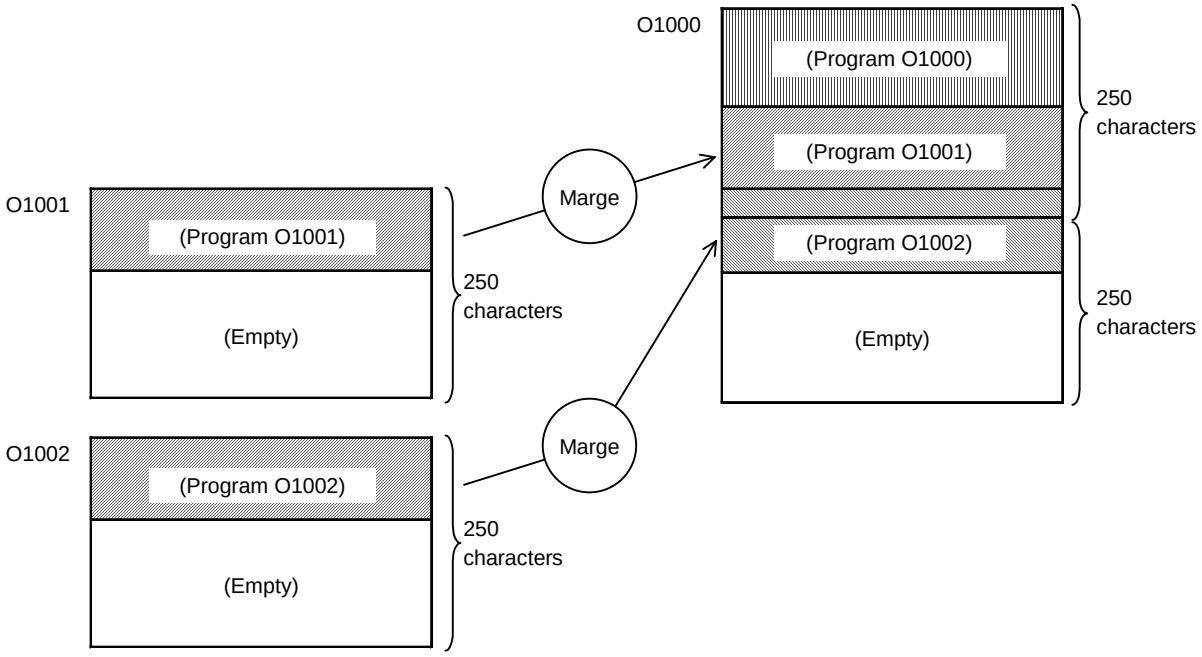
```
                              MERGE EXECUTION
# (  3 ) O(  1001)→O(  1000)
```

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.

⇩

```
                              MERGE COMPLETE
# (     ) O(        )→O(        )
```

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



(Note 1) When using the 2-part system, the machining program merge operation can be switched with the parameters.

#1050 MemPrg	#1285 ext21/bit0	Details
0, 2, 4, 6	-	Of the machining programs registered in the memory common for the systems, the machining programs with the designated program numbers are merged together.
1, 3, 5, 7	OFF	Of the machining programs registered in the memory for the selected system, the machining programs with the designated program numbers are merged together.
	ON	Of all the machining programs registered in the memory for each system, the machining programs with the designated machining program numbers are merged together in each system.

(Note 2) When using the 2-part system and the same number batch generation for all system programs is valid, an error (E14) will occur if the merge source program for all systems is not found.

(Note 3) When using the 2-part system and the same number batch generation for all system programs is valid, the program will be merged in the following manner according to the presence of the merge destination program in each system.

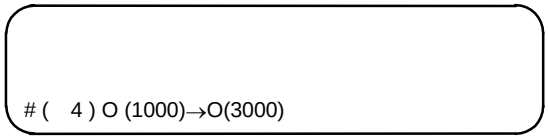
Presence of program		Operation
System 1	System 2	
Yes	Yes	The System 1 merge source program will be merged with the System 1 merge destination program. The System 2 merge source program will be merged with the System 2 merge destination program.
Yes	No	The System 1 merge source program will be merged with the System 1 merge destination program. The System 2 merge destination program will be newly created, and then the System 2 merge source program will be merged with the System 2 merge destination program.
No	Yes	The System 1 merge destination program will be newly created, and then the System 1 merge source program will be merged with the System 1 merge destination program. The System 2 merge source program will be merged to the System 2 merge destination program.
No	No	An error (E14) occurs.

(Note 4) When using the 2-part system and the same number batch generation for all system programs is valid, the programs in the systems having the merge source program are merged. The programs in systems that do not have the merge source program will not be merged.

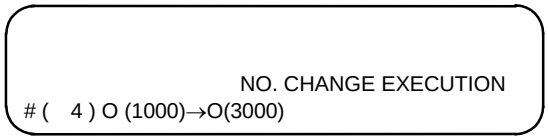
6.4.4 Changing the Machining Program Number

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.

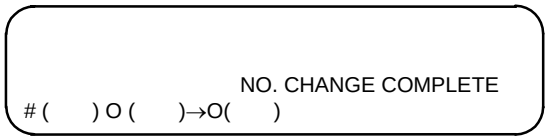
(Example)
When O1000 is changed to O3000:
([4])
O ([1][0][0][0]) → ([3][0][0][0])



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.



(Note 1) When using the 2-part system, the machining program number change operation can be switched with the parameters.

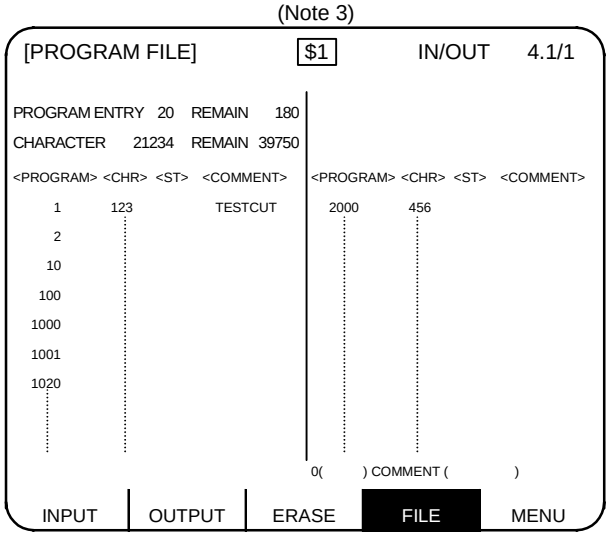
#1050 MemPrg	#1285 ext21/bit0	Details
0, 2, 4, 6	-	Of the machining programs registered in the memory common for the systems, the designated machining program will be registered in the memory common for the systems with a different program number.
1, 3, 5, 7	OFF	Of the machining programs registered in the memory for the selected systems, the designated machining program will be registered in the memory for the selected system with a different program number.
	ON	Of all the machining programs registered in the memory for each system, the designated machining program will be registered in each system memory with a different program number.

(Note 2) When using the 2-part system and the same number batch generation for all system programs is valid, an error (E64) will occur if the rename destination program is found in any of the systems.

(Note 3) When using the 2-part system and the same number batch generation for all system programs is valid, the program in the system having the rename source program will be renamed. Programs in a system that does not have the rename source program will not be renamed.

6.5 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. Note) When using the 2-part system, the total number of programs for the all systems will be displayed at PROGRAM ENTRY and REMAIN.
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. Note) When using the 2-part system, the total number of characters for all systems will be displayed at CHARACTER and 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 also be specified during input from paper tape. 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 (SP) can be written in it. But, the space is ignored after registration for efficient use of memory.

(Note 3) When using the 2-part system, the system name of the currently selected system is displayed as $1 (system 1) and $2 (system 2). This is not displayed when using a 1-part system. (Only L system)

(Note 4) Whether to show or hide name of the selected system can be switched with the parameters. The list of machining program numbers can also be switched with the parameters.

#1050 MemPrg	**#1285 ext21/bit0**	**#1286 ext22/bit0**	**System name display**	**Program number list display**
0, 2, 4, 6	-	-	The name of the selected system is not displayed.	The numbers of the machining programs registered in the memory common for the systems are listed.
1, 3, 5, 7	OFF	-	The name of the selected system is displayed.	The numbers of the machining programs registered in the memory for the selected system are listed.
	-	OFF		
	ON	ON	The name of the selected system is not displayed.	The numbers of the machining programs registered in the memory for all systems are listed.

6.6 RS-232C I/O Device Connection

6.6.1 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.

⚠ 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.

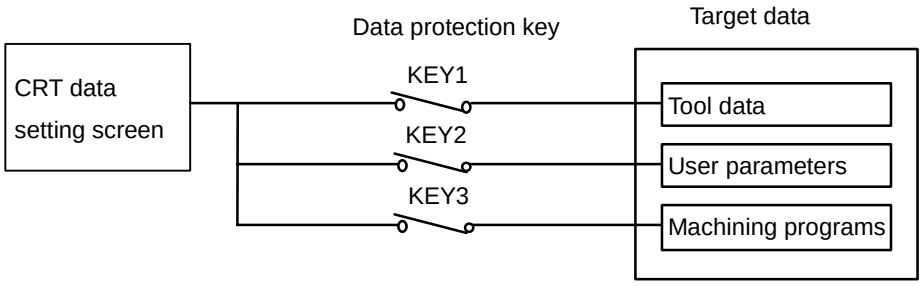
6.7 Data Protection

6.7.1 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 makers. For the details, refer to the manuals issued by individual machine makers).

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 offset	TOOL/COMP TOOL DATA
4	Setting/erasing of nose-R compensation, wear compensation and tool nose point	TOOL/COMP NOSE-R/P
5	Setting/erasing of tool life data	TOOL/COMP TOOL LIFE
6	Setting/erasing of workpiece 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 C.B CAN 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
5	Common variable setting	MONITOR/COMMON VARIABLE
6	I/O basic parameter setting	IN/OUT (PARAMETER)
7	Parameter tape input	IN/OUT/INPUT
8	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 6 in Table 2 does not enter any data, but displays message "DATA PROTECT".
When an attempt is made to perform the operation of number 7 or 8, pressing the INPUT key cannot input or output any data, causing message "DATA 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

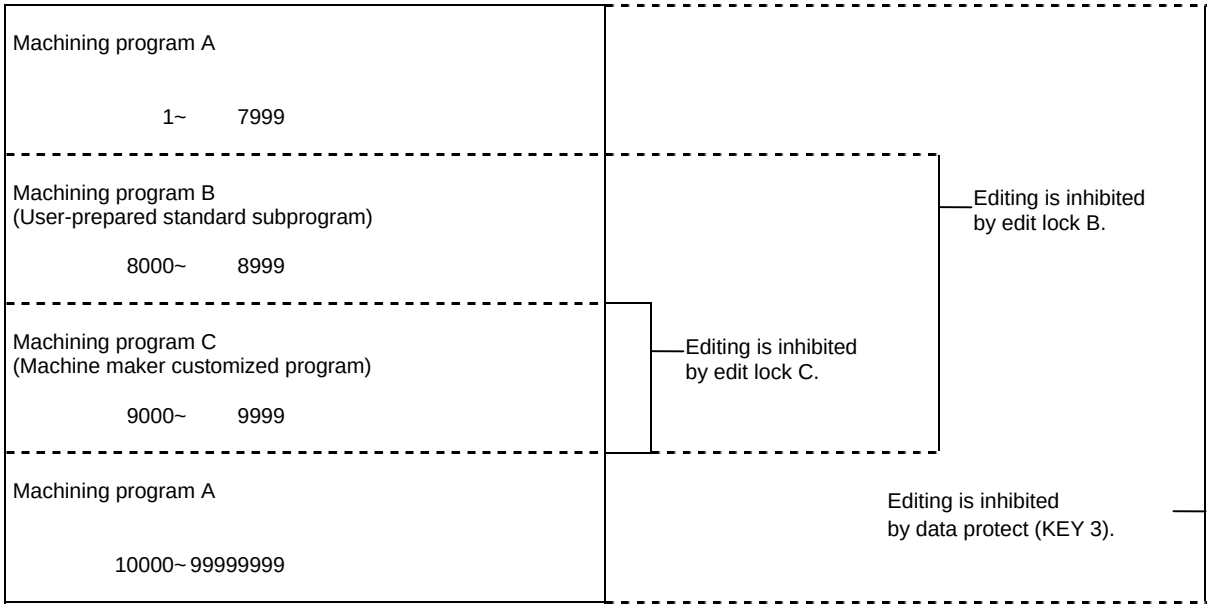
No.	Operation	Screen	Extended operation menu
1	Storing MDI data in memory	MDI	MDI ENTRY
2	Editing machining program	EDIT	—
3	Creating machining program	EDIT	PROGRAM
4	Setting comments of stored program	EDIT	FILE
5	Storing and checking machining program in memory	IN/OUT/INPUT	—
6	Erasing machining program (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 1) 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.

(Note 2) When using the 2-part system, editing, input and output of the machining programs in all systems can be prohibited with the data protection key 3 (Y23A).

6.7.2 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.

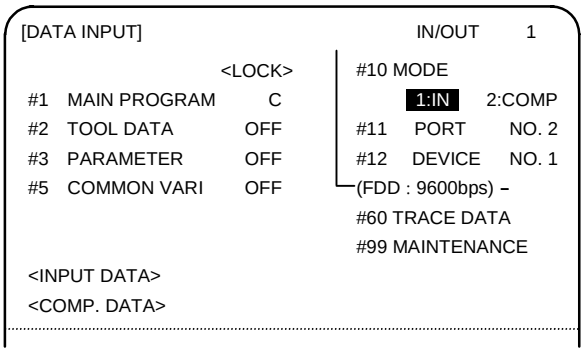
(Note 1) When using the 2-part system, the machining programs in all systems are protected with edit lock B and C.

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:

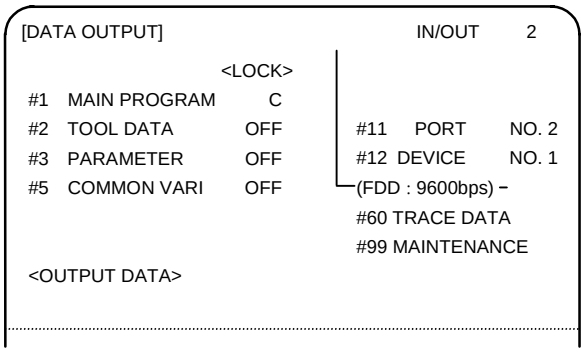
• Information displayed

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)
#3 Parameter	ON	User parameter data is protected (Protection KEY 2 OFF)
	OFF	(Protection KEY 2 ON)

DATA INPUT screen



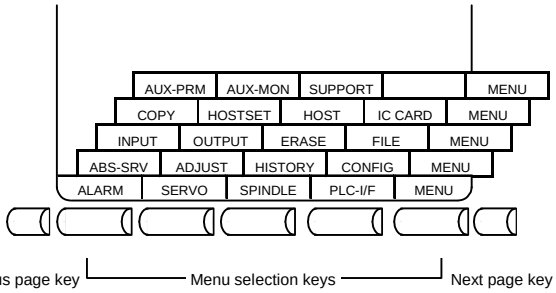
DATA OUTPUT screen



7. Diagnosis

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

Diagnosis menu (No.9 to 10)
Input/output menu (No.5), Program sever menu (No.1 to 3)
Input/output menu (No.1 to 4)
Diagnosis menu (No.5 to 8)
Diagnosis menu (No.1 to 4)



DIAGN menu
No.1 to 4

PREVIOUS PAGE
NEXT PAGE

ALARM MESSAGE
SERVO MONITOR
SERVO MONITOR-2
SERVO DIAGNOSIS
SERVO DIAGNOSIS-2
PS DIAGNOSIS
SPINDLE MONITOR
PLC-I/F

MENU

DIAGN menu
No.5 to 8

PREVIOUS PAGE
NEXT PAGE

ABS SERVO MONITOR
ABS POSITION SET
ADJUST S-ANALOG
OPERATION HISTORY
OPERATION HISTORY
S/W MODULE TREE
H/W MONITOR

MENU

Input/Output menu — Refer to the section on data input/output.

MENU

Program server menu — Refer to the section on program server.

MENU

DIAGN menu
No.9 to 10

AUXILIARY AXIS PARAMETER
AUXILIARY AXIS MONITOR
MELDASNET SUPPORT PARAMETER
NC-DATA SAMPLING

7.1 ALARM MESSAGE

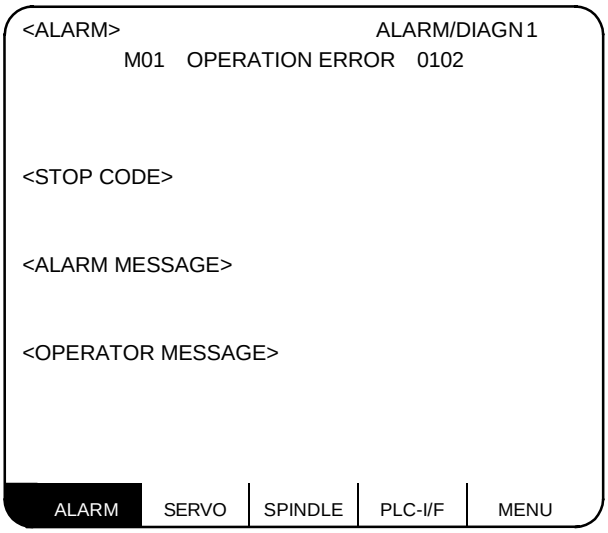
When the menu key ALARM is pressed, the ALARM/DIAGN screen is displayed.

(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.

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.

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.

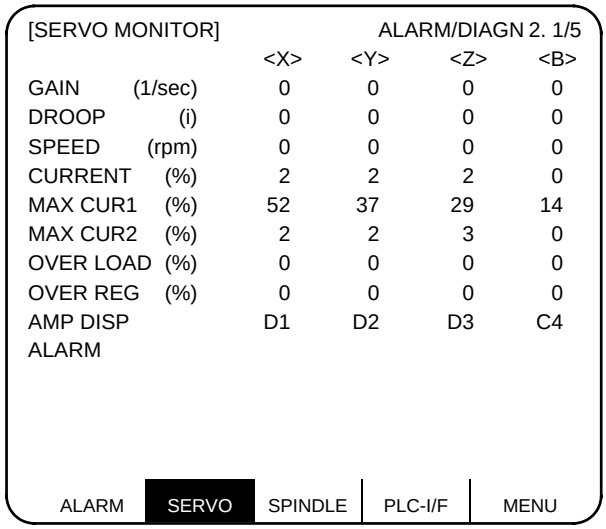
The display can be changed to the previous and next alarms with the [↑][↓] keys.	☞	The [↑] key will show the previous data occurrence. The [↓] key will show the next data occurrence.
Press the [SHIFT] and [C.B CAN] keys.	☞	The display will return to the normal current alarm display screen. This is the same for changing over to other screens once.

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

7.2 SERVO MONITOR

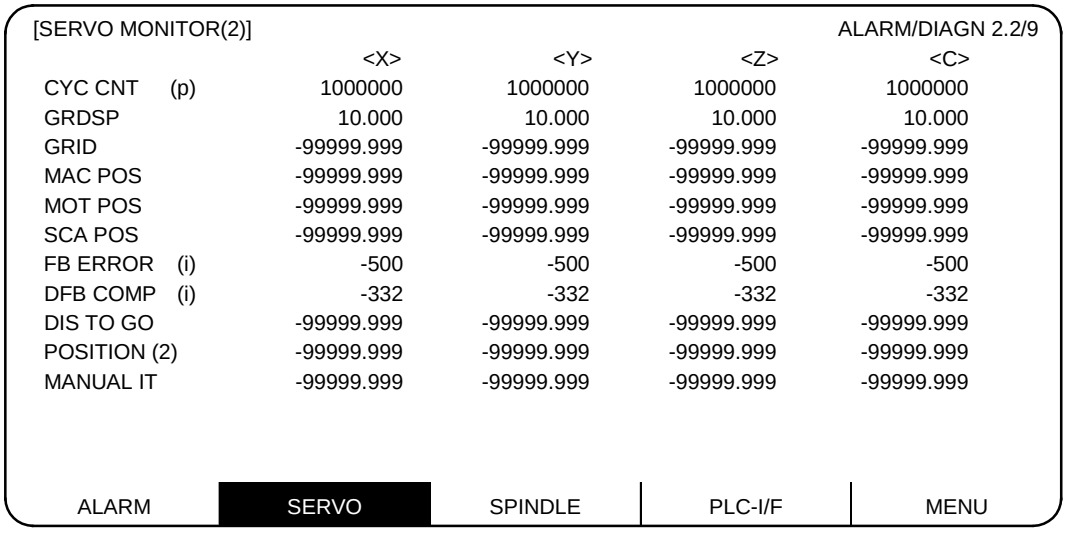
7.2.1 Servo Monitor

When the menu key SERVO is pressed, the SERVO MONITOR screen is displayed.



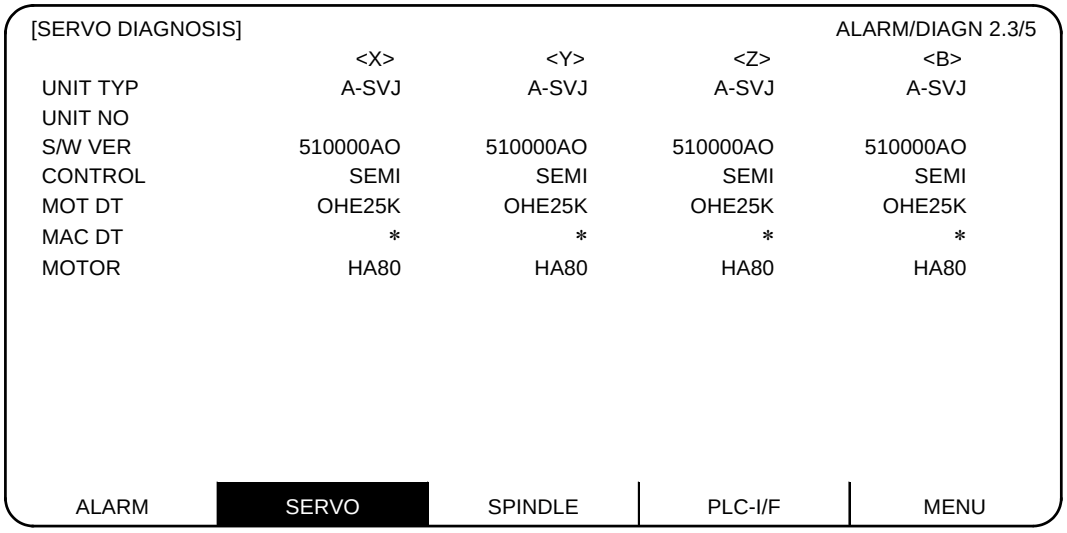
Data	Display unit	Explanation
GAIN	1/s	The position loop gain is displayed. The position loop gain is: $\frac{\text{feedrate (mm/s)}}{\text{tracking delay error (mm)}}$
DROOP	i	An error of the actual machine position to the command position is called droop. This error is proportional to the command speed 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	—	The same details as the driver's 7-segment LED are displayed.
ALM	—	The alarms and warnings other than the driver display are displayed.

7.2.2 Servo Monitor (2)



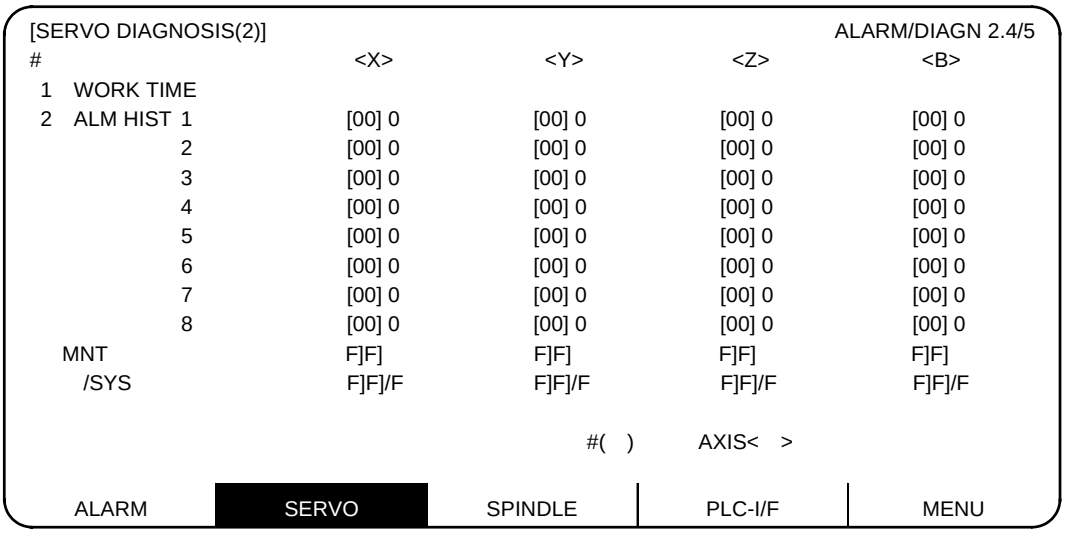
Data	Display unit	Explanation
CYC CNT	Pulse	The position within one rotation of the encoder detector is displayed. The position uses the grid point value as 0, and will display the position within one rotation within the range of 0 ~ RNG (movement unit) ∗1000.
GRDSP	Command unit	The grip interval for zero 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 machine end 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.

7.2.3 Servo Diagnosis



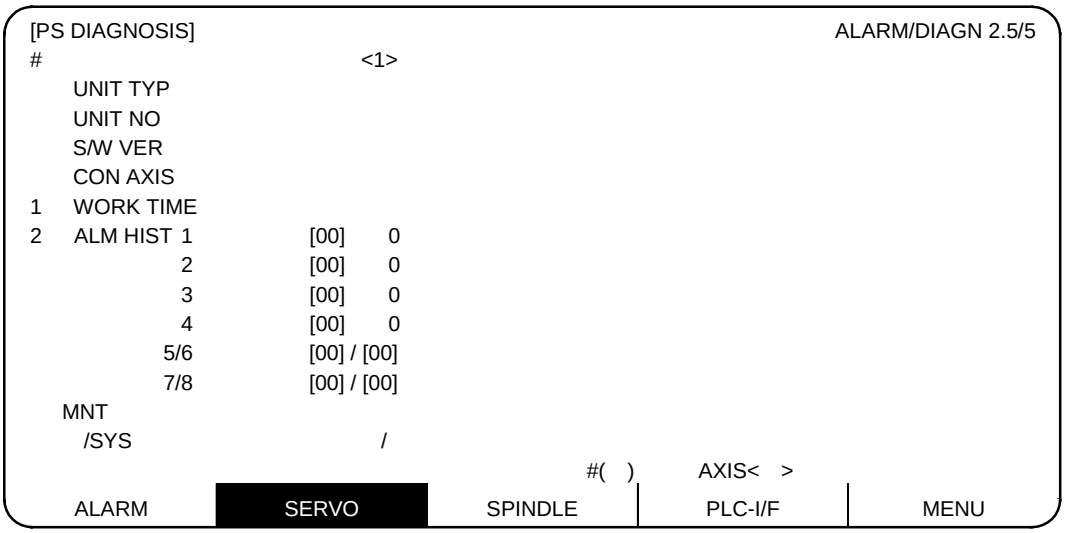
Display item	Explanation
UNIT TYP	This is the servo driver type.
UNIT NO	This is the servo driver manufacturing number.
S/W VER	This is the servo driver's software version.
CONTROL	SEMI: semi-closed
MOT DT	This is the motor detector type.
MAC DT	This is the machine end detector type. If the SEMI control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

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. [aa] b aa : Alarm No. b : Operation time at alarm occurrence
MNT/SYS	Maintenance history NVRAM status

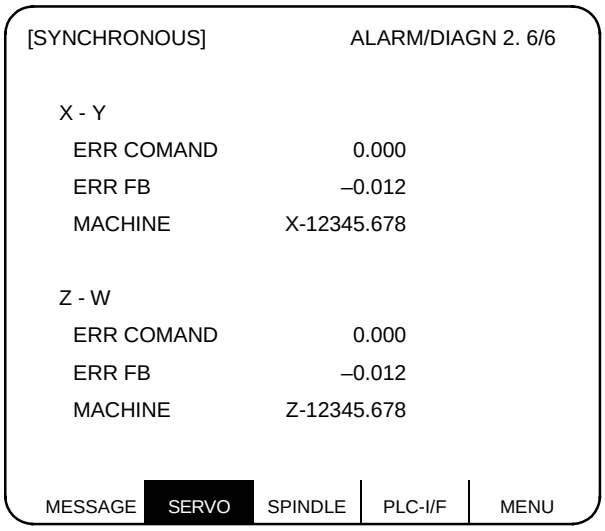
7.2.5 PW Diagnosis



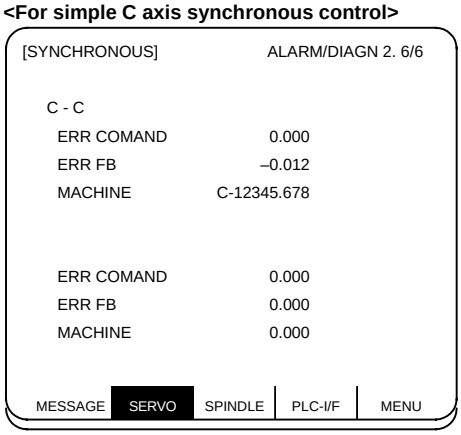
Data	Display unit	Explanation
UNIT TYP	—	This is the power supply unit type.
UNIT NO	—	This is the power supply unit manufacturing number.
S/W VER	—	This is the software version.
CON AXIS	—	This is the driver connecting each power supply unit.
WORK TIME	Time	This is the cumulative ready ON time.
ALM 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

7.2.6 Display Items for the Synchronous Error

The various data related to the synchronous error is monitored.



<For simple C axis synchronous control>



Display item	Explanation
ERR COMAND	This is the deviation of the slave axis in respect to the master axis. The error of the command value to the servo control section is displayed. For simple C axis synchronous control, this indicates the difference of the command value to the C axis control section caused by deviation of the slave axis in respect to the master axis. If this error occurs, the parameters that should be the same for the synchronous axes are different. Check the setting value. The command difference is calculated with the following expression. Command error = Command $_S$ – command $_M$ – Δ Command $_S$: Slave axis command value Command $_M$: Master axis command value Δ : Command $_S$ when synchronous operation is selected – command $_M$
ERR FB	This is the deviation of the slave axis feedback value in respect to the feedback value from the master axis servomotor. The error of the actual machine position is displayed. For simple C axis synchronous control, this indicates the error in the actual machine position caused by deviation of the slave axis feedback value in respect to the feedback value from the master axis' C axis motor. The synchronous error check is carried out on this error. FB error = FB $_S$ – FB $_M$ – Δ FB $_S$: Slave axis feedback value FB $_M$: Master axis feedback value Δ : FB $_S$ when synchronous operation is selected – FB $_M$
MACHINE	This displays the machine position of the master axis.

(Note 1) If the operation method is changed due to changes in the R435/436 register, the error offset Δ will be created at the rising edge of the changes.
Note that if the zero point is not set, the error offset Δ will be created after the zero point is set.

(Note 2) The axis names in this screen are displayed only during the synchronous or independent operation.
The value of "ERR COMAND", "ERR FB" and "MACHINE" are displayed only during the synchronous operation. These are all "0.000" during independent or asynchronous operation.
For simple C axis synchronous control, the axis name on the SYNCHRONOUS screen is displayed only during synchronization. The command error, FB error and machine values indicate the values only for synchronization. These are all "0.000" during asynchronous operation.

(Note 3) If the FB error exceeds the tolerable value, the operation alarm (51: Synchronization error too large) will occur, and the motor feed will stop at the point the tolerable value is exceeded. If the synchronization error too large alarm occurs, refer to the SYNCHRONOUS screen, and move the axis in the correction mode so that it is positioned within the tolerable range. Note that movement commands that cause the synchronization error to increase will not move the axis during the operation alarm (51: Synchronization error too large) so the alarm cannot be reset. The FB error is also checked during the correction mode, so if the tolerable error range is exceeded during the correction mode, the operation alarm (51: Synchronization error too large) will occur.
If the synchronization error too large alarm occurs during simple C axis synchronous control, cancel the synchronization designation (asynchronous), and input reset.

(Note 4) If the FB error is "0" (acceleration/deceleration type or acceleration/deceleration time constant differ, etc.) during the operation alarm (51: Synchronization error too large), the operation alarm (51: Synchronization error too large) will be held. This alarm can be canceled by resetting.

(Note 5) The error will not be checked before the first reference point return and when the synchronization error tolerance value is 0 (synchronization error check invalid).

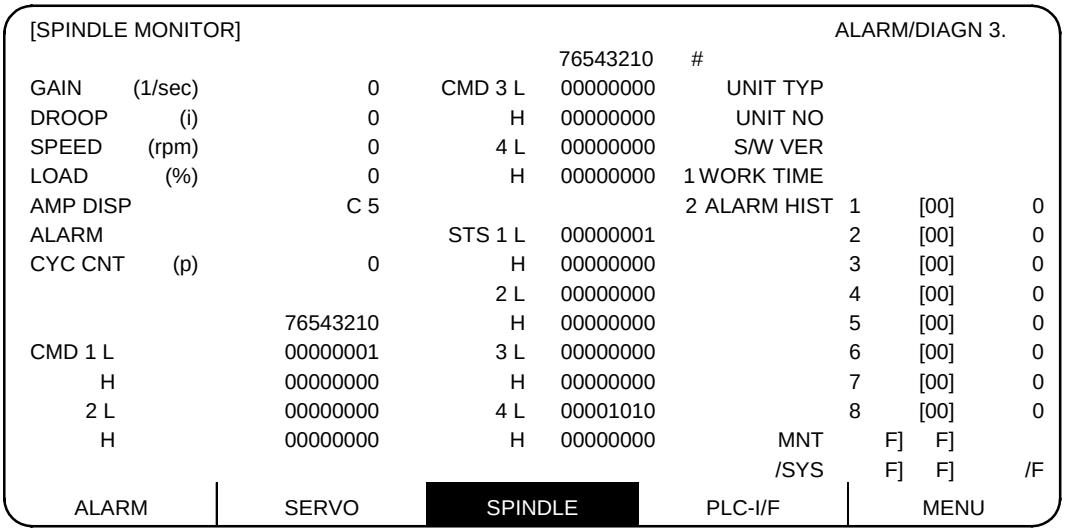
(Note 6) Positions where NC interprets synchronization error as 0 (position used as reference for synchronization error check)

(a) When zero point is established
  (i) For relative position detection or dog type absolute position detection
    ... 1st reference point
  (ii) For dogless absolute position detection
    ... Basic machine coordinate system zero point (G53 zero point)
(b) When setting operation method after establishing zero point
  (i) When operation method is set with first ladders after power ON
    ... 1st reference point after zero point is established
    (during relative position detection)
    The current error is held
    (during absolute position detection)
  (ii) When operation method is set with other method
    … Machine position when synchronous control is turned ON

Positions where NC interprets synchronization error as 0 during simple C axis synchronous control (position used as reference for synchronization error check)

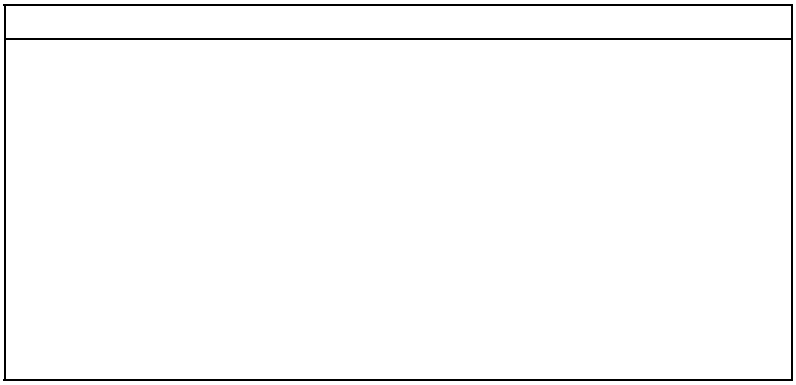
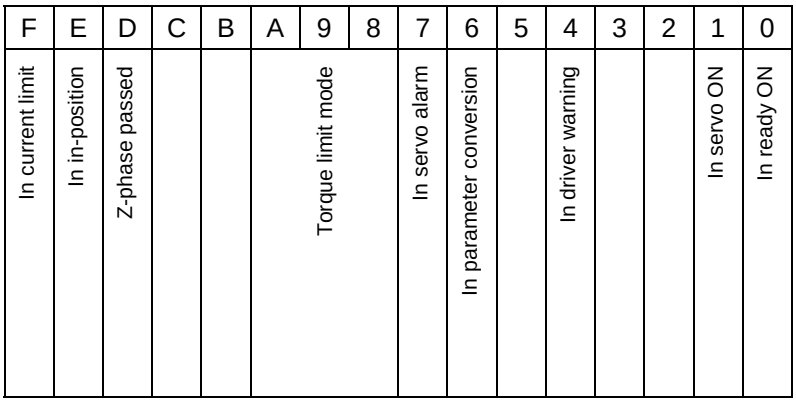
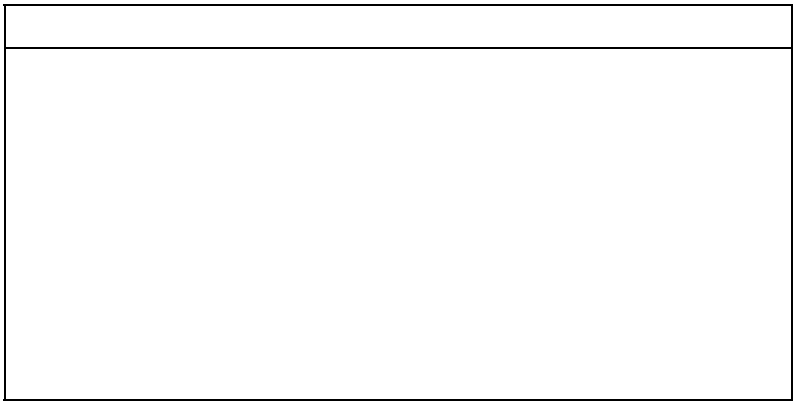
(a) When first zero point is established after power ON
  1st reference point
(b) When setting operation method after zero point is established
  Machine position at synchronous control ON

7.3 SPINDLE MONITOR



Data	Display unit	Explanation
GAIN	1/s	Spindle position loop gain
DROOP	i	Position deflection amount
SPEED	r/min	Motor rotation speed
LOAD	%	Motor load
AMP DISP		7-segment display of driver
ALARM		Alarms other than the amplifier display (max. 3 alarms)
CYC CNT	pulse	Angle data from detection basic position (Z phase)
CMD		Control input signal from control unit
STS		Control output signal from driver
UNIT TYP		This is the spindle controller type.
UNIT NO		This is the spindle controller manufacturing number.
S/W VER		This is the spindle controller 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

<table>
<tr><th>Display item</th><th>Explanation</th></tr>
<tr><td>Control input 1</td><td>Control input signal from control unit
<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 G1 modal</td><td></td><td></td><td></td><td></td><td colspan="3">Torque limit mode</td><td>Servo alarm reset command</td><td>Parameter conversion command</td><td></td><td></td><td></td><td></td><td>Servo ON command</td><td>Ready ON command</td></tr>
</table></td></tr>
<tr><td>Control input 2</td><td>Control input signal from control unit
<table>
<tr><td></td></tr>
<tr><td></td></tr>
</table></td></tr>
<tr><td>Control input 3</td><td>Control input signal from control unit
<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 colspan="2">Sub-motor selection command
0:Main motor 2: Sub motor2
1:Sub motor1 3: Sub motor3</td><td>L winding selection command</td><td>Orient start</td><td>Indexing reverse run</td><td>Indexing forward run</td><td>Reverse run start command</td><td>Forward run start command</td><td colspan="3">Gear selection command
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="5">Spindle control mode selection command</td></tr>
</table></td></tr>
</table>

<table>
<tr><th>Display item</th><th>Explanation</th></tr>
<tr><td>Control input 4</td><td>Control input signal from control unit
  </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></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
  </td></tr>
</table>

Display item	Explanation
Control output 3	Driver control output signal
Control output 4	Driver control output signal

Control output 3

F	E	D	C	B	A	9	8	7	6	5	4	3	2	1	0
In sub-motor selection 0:Main motor 2: Sub motor2 1:Sub motor1 3: Sub motor3		In L winding selection	In orient start	In index reverse run	In index forward run	In reverse run start	In forward run start	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			In spindle control mode selection				

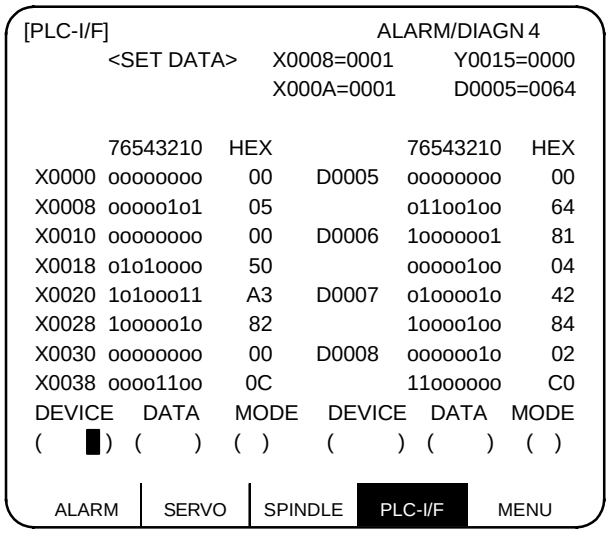
Control output 4

F	E	D	C	B	A	9	8	7	6	5	4	3	2	1	0
No.2 in-position											Orient completed	Zero speed detection	Speed reached	Speed detection	Current detection

7.4 PLC Interface Diagnosis

When the menu key [PLC-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) 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.

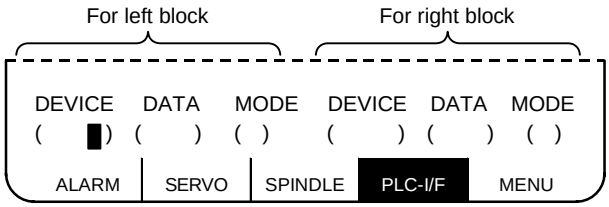


7.4.1 PLC-I/F Setting and Display

(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.

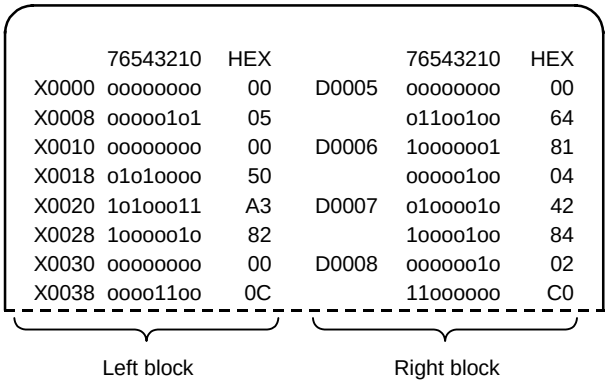
MODE ( )

Specify the type of forcible definition.

1: Single-shot type
2: Modal type

(2) Device data display area

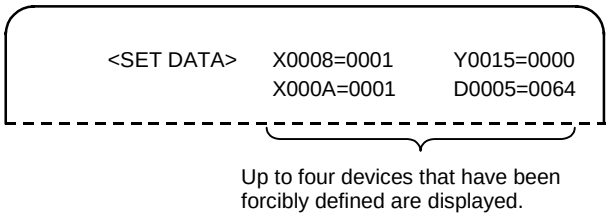
Data corresponding to the device numbers specified in the setting area is displayed.
Data is displayed in both binary notation and hexadecimal notation.
The device numbers can be displayed in the left and right blocks separately.



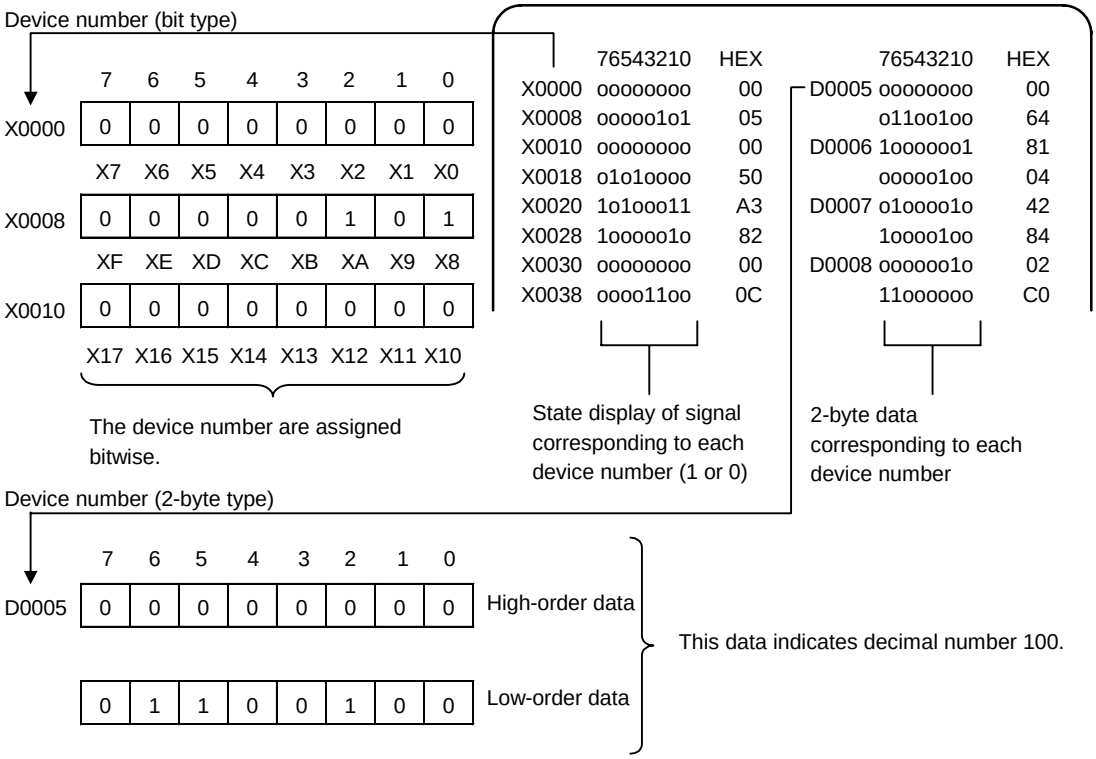
(3) Forcible definition data display (modal type)

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) How to read device numbers and display data



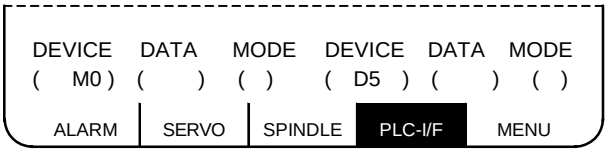
7.4.2 PLC Device Data Display

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.

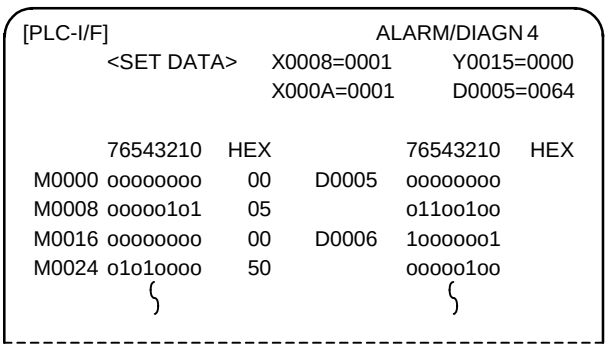
(1) Display device number setting
Set the device number in DEVICE ( ). If a different device number or device address is set in the right DEVICE ( ), the specified device numbers are displayed in the left and right halves of the screen from the display area top to bottom.

(Example 1)

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

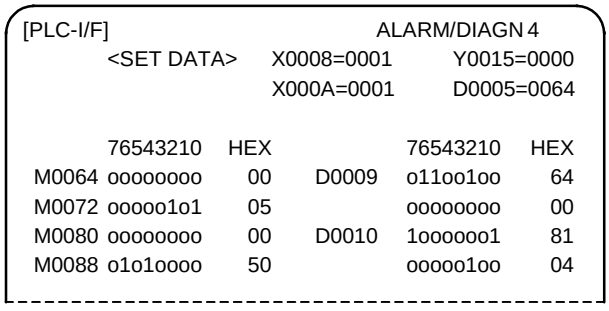


Press the INPUT key.



(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 PREVIOUS PAGE or NEXT PAGE key.

When the display screen shown in (Example 1) appears, press the NEXT PAGE 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.

7.4.3 PLC Interface Signal Forcible Definition (Single-shot Type)

This function is used to forcibly define signals to check the PLC functions.
This function is available for the following device numbers: X, Y, U, W, S, M, G, F, L, E, T, Q, C, B, D, and R.

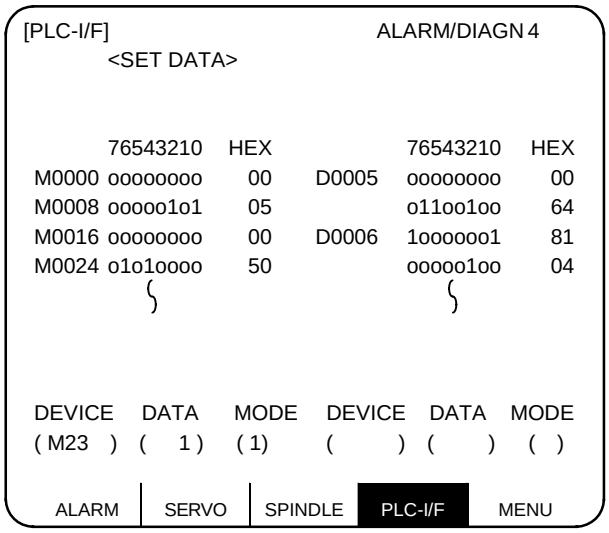
(1) Mode setting

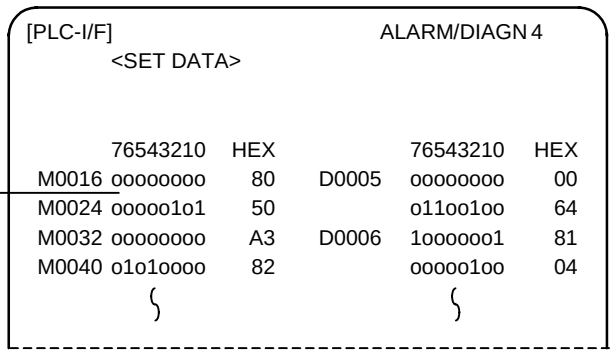
Set the device number to be forcibly set in DEVICE ( ), definition data in DATA ( ), and 1 in MODE ( ), then press the [INPUT] key. The setup data is processed and forcibly defined at the top of one cycle of user PLC.

(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 (X or U) 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.

7.4.4 PLC Interface Signal Forcible Definition (Modal Type)

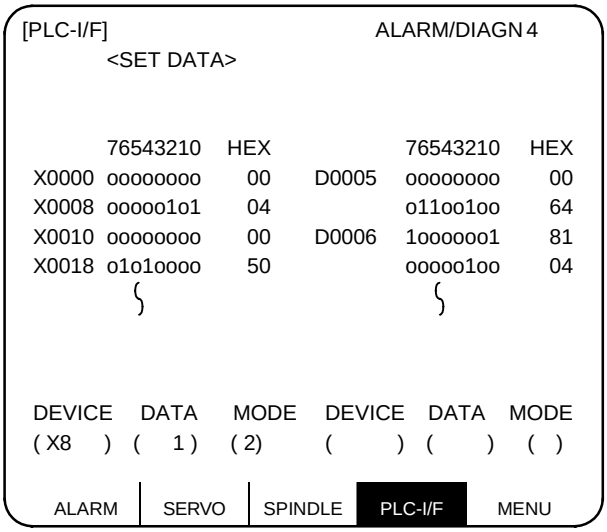
This is PLC interface signal forcible definition of modal type. Once it is set, it is held until canceled.
The device signals that can be set by using this function are X, Y, U, W, S, M, G, F, L, E, T, Q, C, B, D and R.

(1) Mode setting

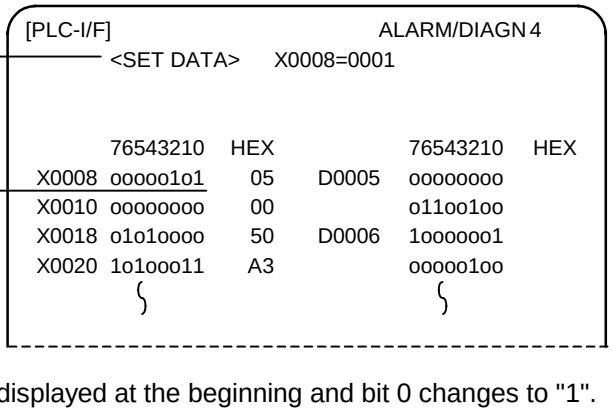
Set the device number to be forcibly defined in DEVICE ( ), definition data in DATA ( ), and 2 in MODE ( ), then press the INPUT key. The data is processed and forcibly defined.

(Example 3)

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



Press the INPUT key.



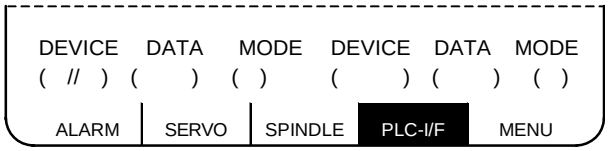
A maximum of four sets of forcibly defined device numbers and numeric data are displayed.

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

A maximum of four sets can be forcibly defined in mode 2. If four sets have been defined and additional setting is made, forward feed is made and the subsequent 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.)

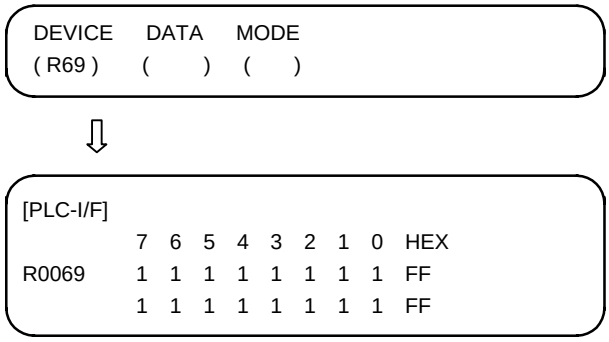
Forcibly defined data is also canceled by turning ON/OFF power.

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 of the setting and display unit. At this time, the cause can be known.

(1) Check using the DIAGN screen

Set R69 in the DEVICE ( ) and then press INPUT key.



R0069

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

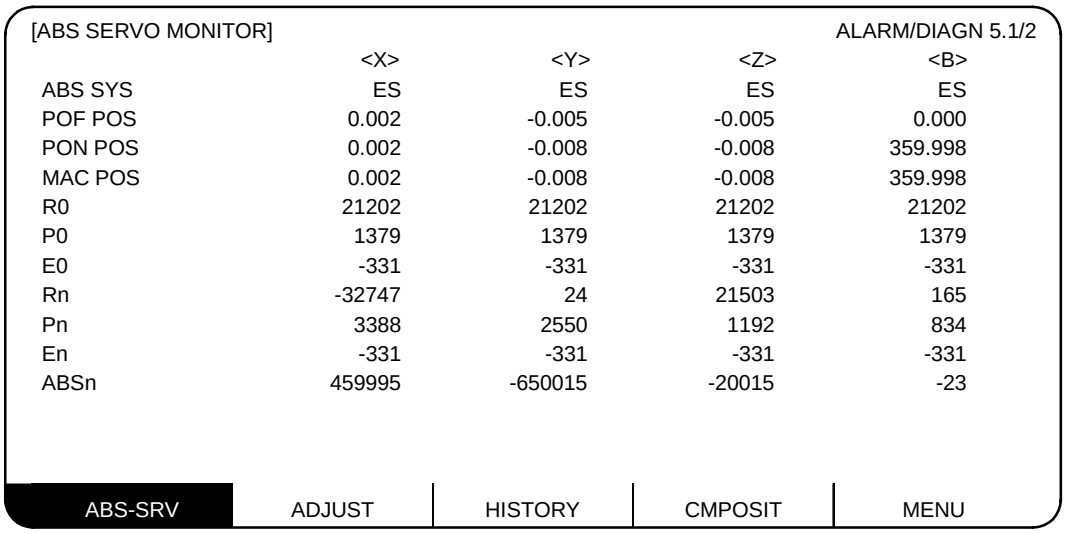
(0: EMG stop 1: Normal)

- Bit 0: Built-in PLC Stop state
- Bit 1: External PLC FROM, TO command not executed
- Bit 2: External PLC Not ready
- Bit 3: External PLC Renewal error
- Bit 4: Control unit EMG connector Emergency stop state
- Bit 6: Built-in PLC Software emergency stop output device Y29F is "1".
- Bit B: User PLC Illegal codes exist.
- Bit C: PLC high-speed processing error
- Bit D: Door interlock, dog/OT arbitrary allocation device illegal
- Bit E: Spindle amplifier emergency stop output
- Bit F: Servo amplifier emergency stop output

7.5 Absolute Position Monitor

7.5.1 ABS SERVO MONITOR

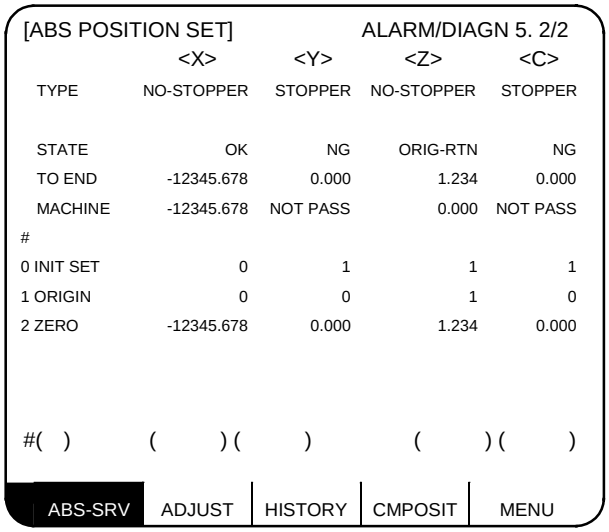
The servo state in the absolute position detection system is displayed. This can be used to confirm each detector data for the current machine value.

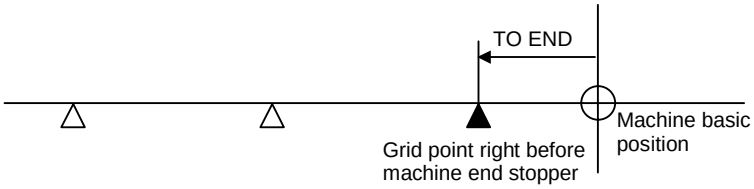
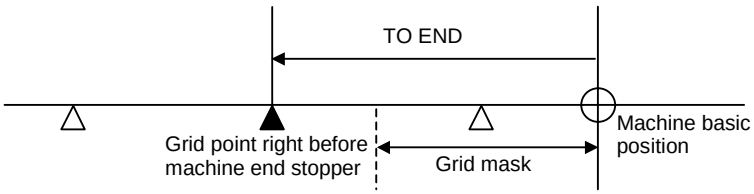


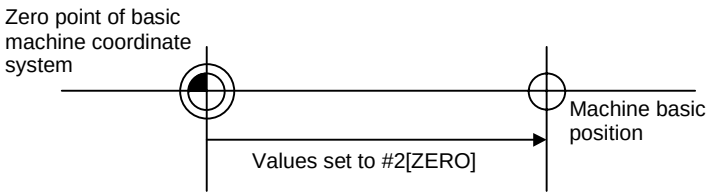
Data	Display unit	Explanation
ABS SYS		The status of the absolute position detection system on the servo side is displayed. ES : Semi-closed encoder ESS : Semi-closed high-speed serial encoder INC : Incremental
POF POS	Command unit	The absolute position when the power is turned OFF is displayed.
PON 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.

7.5.2 Absolute Position Initialization

Pressing the menu key ⟩ABS-SRV⟨ displays the ABS SERVO MONITOR screen. When the ⟩NEXT PAGE⟨ 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.



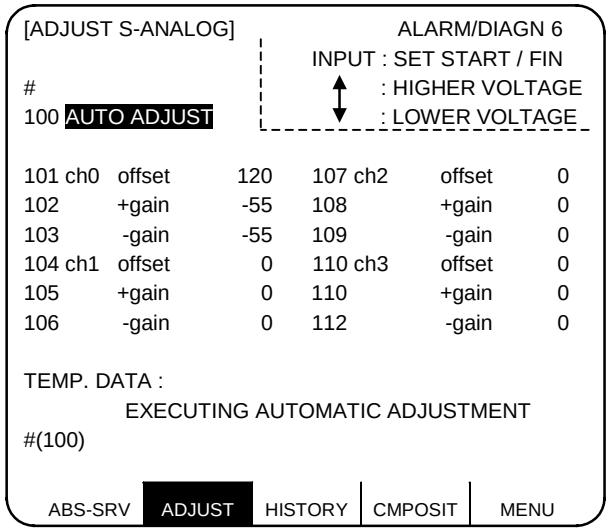
Display item	Explanation
TYPE	The position detection system selected by the absolute position parameter is displayed. INC. : Incremental system DOG : Dog type absolute position detection STOPPER : Dogless absolute position detection machine end method NO STOPPER : Dogless absolute position detection marked 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 marked point alignment method is used. Specify "1" when the machine has been positioned to the basic 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 III Section 4 "Absolute Position Detection System".

7.6 Adjustment

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



7.6.1 Adjustment Preparation

(1) The PLC must be stopped before adjusting the S-analog output.
The following two methods can be used to stop the PLC.
1) Stopping with rotary switch
Set the rotary switch NCSYS on the bottom of the unit to 1.
2) Stopping with PLC onboard
Press the F0, 〉MENU〉, 〉FILE〉 and 〉RUN/STOP〉 keys, and input "1".

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

7.6.2 Automatic Analog Output Adjustment

Select #100 to execute automatic adjustment.
→ "AUTO ADJUST" will be highlighted, and the message "EXECUTING AUTOMATIC ADJUSTMENT" 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 "AUTOMATIC ADJUSTMENT COMPLETE" will display, and the "AUTO ADJUST" item will return to the normal display.

7.6.3 Adjustment Procedure

1) Offset adjustment (Ex. ch0)
   (1) Select #101.
   → "ch0 offset" will be highlighted, indicating that it is being adjusted.
   The current setting value will display in the "TEMP.DATA" column.
   (2) Monitor the analog output voltage.
   (3) 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.
   (4) 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)
   (1) Select #102.
   → "+gain" will be highlighted, indicating that it is being adjusted.
   (2) The current setting value will display in the "TEMP.DATA" column.
   (3) 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.
   (4) 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)
   (1) Select #103.
   → "−gain" will be highlighted, indicating that it is being adjusted.
   The current setting value will display in the "TEMP.DATA" column.
   (2) Monitor the analog output voltage.
   (3) 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.
   (4) 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.

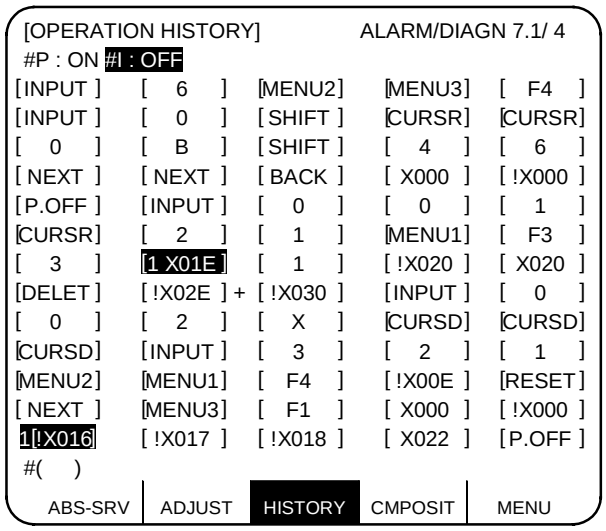
7.6.4 Parameter Input/Output

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

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.
Information that can be traced includes alarm data, and stop code and input signal change data. This data is held even after the power is turned OFF.
Latest data is displayed on the upper left of the screen, previous data is displayed toward the right, and the least recent data is displayed on the lower part of the screen. This screen has four pages. The least recent data is displayed on the lower right of the fourth page. The pages can be switched by the ⟨PREVIOUS PAGE⟩ and ⟩NEXT PAGE⟨ keys.
Up to 256 points of data can be displayed.



[X000] ~ [X2FF] : Change in input signal (ON → OFF)
[!X000] ~ [!X2FF] : Change in input signal (OFF → ON)
\+ [X000] ~ + [!X2FF] : Simultaneous input
[F1] ~ [F6] : Function keys
[1] ~ [Z] : Alphanumeric keys
[CURSL] : Cursor left movement
[CURSR] : Cursor right movement
[CURSU] : Cursor up movement
[CURSD] : Cursor down movement
[P. OFF] : Power OFF
[MENU1] ~ [MENU5] : MENU key
n [!X000] Display in reverse video : n: Number of alarms: An asterisk (∗) is displayed if the number exceeds 9.

(Note 1) If two or more input signals change at the same timing, those signals except the first one are indicated with a plus sign (+).
(Note 2) The NC alarm 5 (X2A1/U121) is not saved in the operation history.

(1) Suppressing the operation history function

Data is kept updated according to changes in input signals and key inputs. If it is desired to suppress the operation history function and retain the current data, enter "I" in # ( ) in the setting field and press the ⟩INPUT⟨ key.

This prevents operation history data from being updated. "TRACE OFF" is displayed in reverse video on the upper right of the alarm message screen. To resume the operation history function, enter "P" in # ( ) in the setting field and press the ⟩INPUT⟨ key.

(2) Clearing operation history data

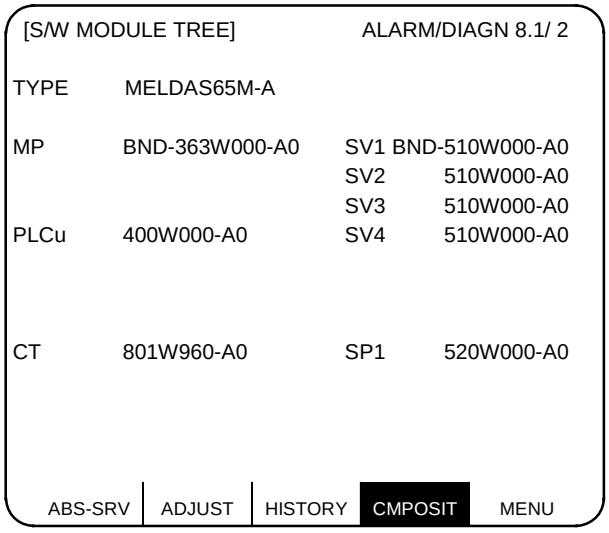
Pressing the ⟨C.B CAN⟩ key while holding down the ⟩SHIFT⟨ key and then pressing the ⟩INPUT⟨ key clears operation history data completely. "X00" is displayed all over the screen.

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.

7.8.1 S/W MODULE TREE

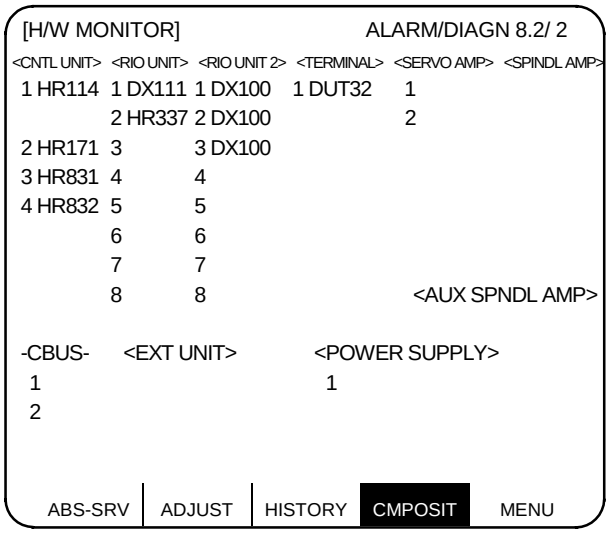
Pressing the menu key ⟩ CMPOSIT ⟨ displays the S/W MODULE TREE screen.



(Note) The "SERIAL" display may not be provided depending on the model.

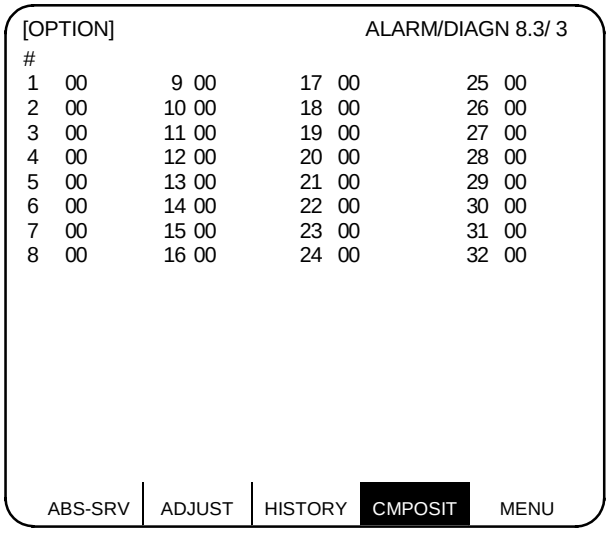
7.8.2 H/W MONITOR

Pressing the menu key ⟩ CMPOSIT ⟨ and then ⟩ NEXT PAGE ⟨ displays the H/W MONITOR screen.



7.8.3 Option

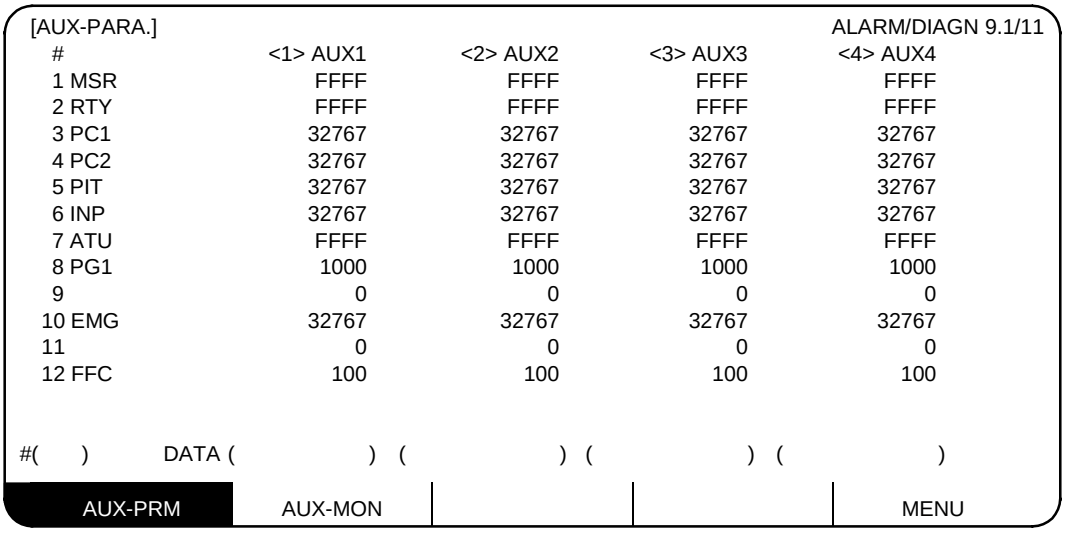
The following screen will open when the NEXT PAGE key is pressed.



7.9 Auxiliary Axis Parameter

7.9.1 Auxiliary Axis Parameter Screen

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



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

7.9.2 Backup

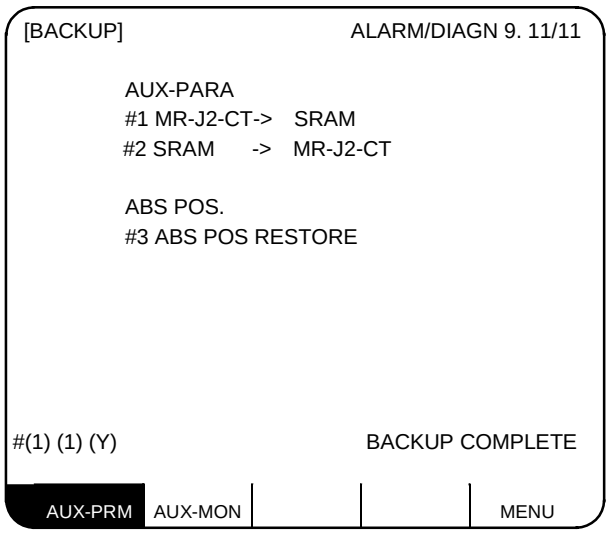
SRAM backup of the auxiliary axis parameters and absolute position restoration during amplifier replacement are executed on the BACKUP screen.
Only execute the operations during amplifier replacement.

(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.

(a) 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.



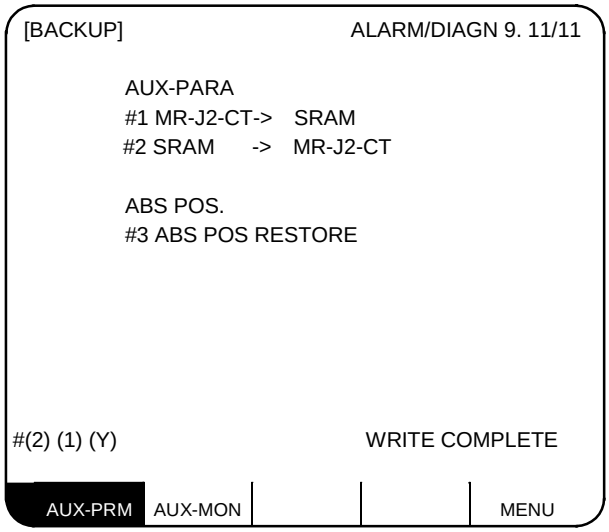
Input "(1) (a)" from the BACKUP screen. (a: axis No. (1 to 4))
After inputting, the message "CONTINUE Y/N" will appear for confirmation of the data backup. The data is backed up in the SRAM only if "(Y)" is input.
The message " BACKUP EXECUTION " will appear when the backup operation is carried out.
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.

(b) Writing auxiliary axis parameters to MR-J2-CT

After backing up the auxiliary axis parameters in the SRAM, turn the NC and MR-J2-CT power OFF, and replace the MR-J2-CT amplifier.
The replacement amplifier must be initialized beforehand.
After replacing the MR-J2-CT amplifier, turn the NC and MR-J2-CT power. Then, 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 4))
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 "AUX. WRITTING EXEC." will appear when the writing operation is carried out.
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.

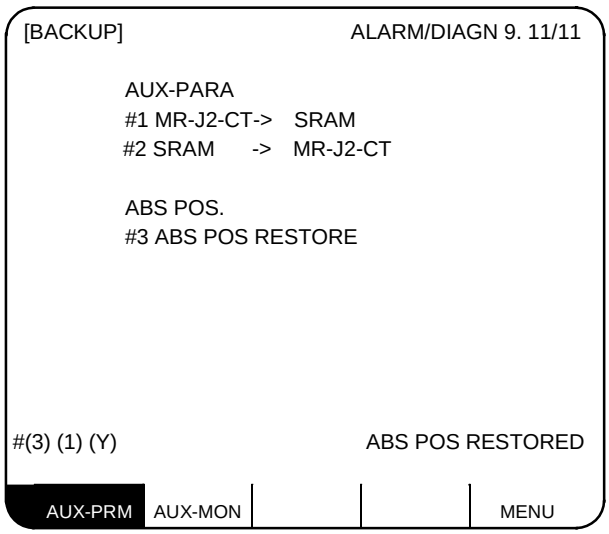
#7 ATU in SRAM	#7 ATU set in MR-J2-CT	Auto-tuning parameter written
Auto-tuned (0 or 1)	Auto-tuned (0 or 1)	Not written
Not auto-tuned (2)	Auto-tuned (0 or 1)	Written **(Note 1)**
Auto-tuned (0 or 1)	Not auto-tuned (2)	Written **(Note 1)**
Not auto-tuned (2)	Not auto-tuned (2)	Written **(Note 1)**

(Note 1) Which parameters can be input depends on the setting of “#7 ATU” parameter.
(Refer to "6.2.7 Outputting Auxiliary Axis Parameter Data" for details on auto-tuning target parameters.)

(Note 2) The NC 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.

(c) 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 4))
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.

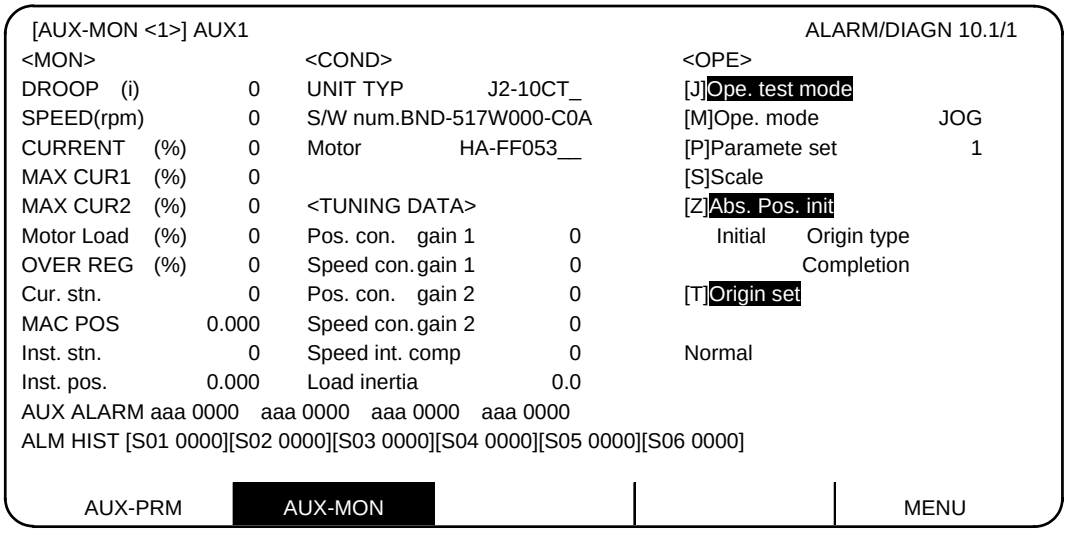
	No. of axes set No. of axes connected, or the No. of axes set = 0.
MR-J2-CT parameter input	MR-J2-CT parameter input not carried out.
MR-J2-CT parameter output	MR-J2-CT parameter output not carried out.

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.

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 rotation 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 "7.10.2 Auxiliary Axis Adjustment Function" for details on the <OPE> area on the screen.

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]

[Sxx ****]
Sxx : Alarm No.
**** : Alarm information

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 zero 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>3</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 : Marked 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>Marked 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 BASIC POINT SETTING ON</td><td>—</td><td>Ret. Ref. P</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>Completion</td><td>Completion</td><td>Completion</td></tr>
</table>
Explanation of display details
Completion : This is displayed when the absolute position of the stopper, marked point alignment, or dog-type method is established.
For the stopper and marked 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, marked 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 marked point alignment method, this indicates the status when the tool is moving in the basic 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 marked 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 basic point is designated.

Zero-P.Ret : This shows that the tool has returned to the zero 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.

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 (R1784 bit0) must be ON.

3) The SERVO OFF signal and INTERLOCK CANCELED signal (R1703/R1709/R1715/R1721 bits 0, 4, 5) must be OFF.

4) The OPERATION START signal (R1702/R1708/R1714/R1720 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] (SHIFT + Q on the operation board)

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

Operation key: [J] (SHIFT + Q on the operation board), or carry 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 "Marked point alignment" method. This function operates when initializing the absolute position. The function is turned ON/OFF every time the key is pressed.
Marked point set	Operation key: [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 basic 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.

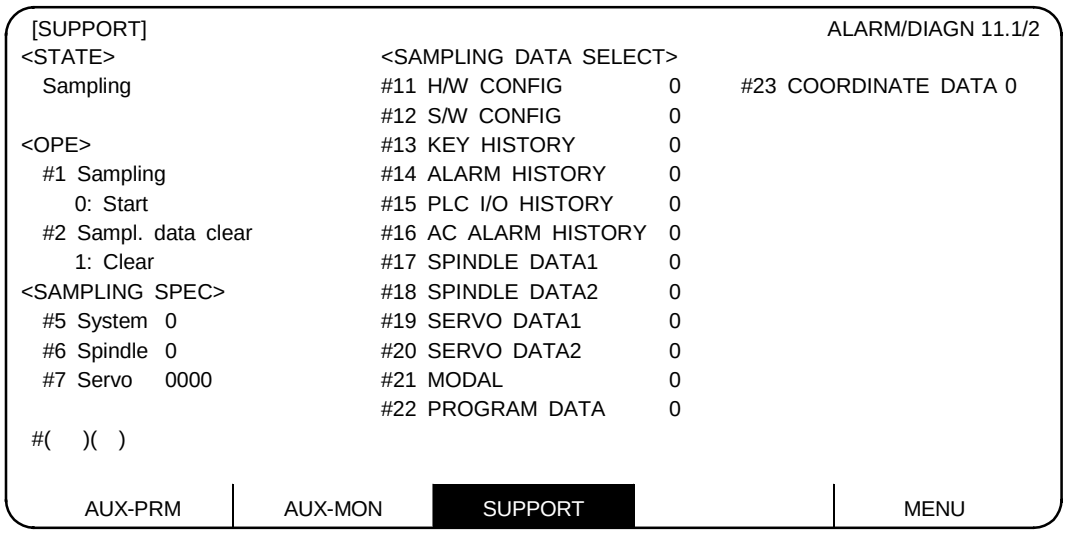
The operation is stopped and the operation adjustment mode is canceled if after starting, the screen is changed from the AUXILIARY AXIS MONITOR screen to another screen, or the monitor axis is changed.
The following table shows the relation between the various key operations and the operation start/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

7.11 MELDASNET Support Parameters

Pressing the menu key [SUPPORT] displays the MELDASNET SUPPORT PARAMETER screen.



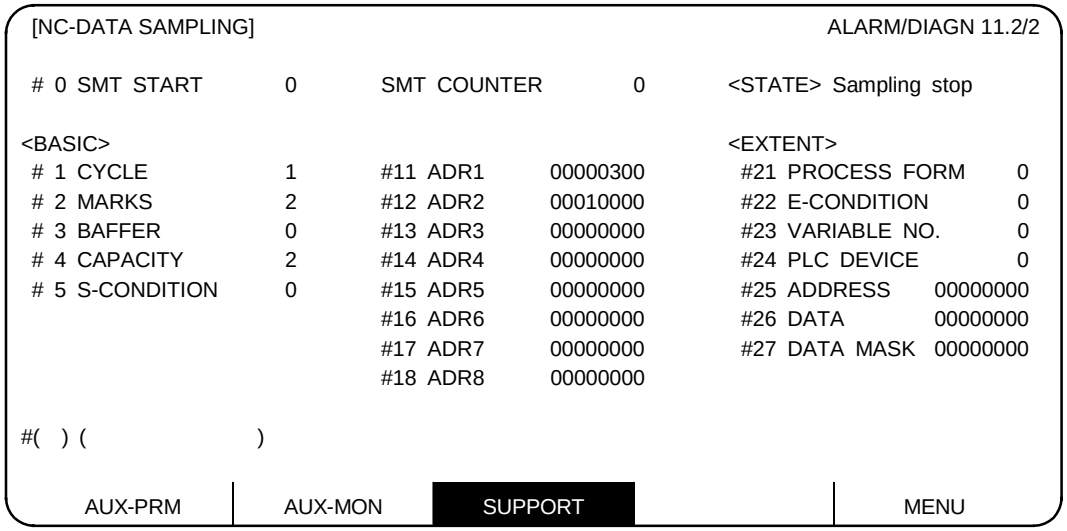
After setting #5 to #7 and #11 to #23, turn the NC power OFF. The settings will be validated after the power is turned ON again.

Main item	#	Item	Details	Setting range
STATE	–	Sampling	The sampling execution state is displayed. "sampling" : Sampling is being executed. "sampling stop" : Sampling is stopped. "sampling invalid" : The MELDASNET function is invalid.	–
OPE	1	Sampling	Restart of sampling is designated. 0: Sampling is restarted.	0
	2	Sampl. data clear	Sampling is stopped, and the sampled data is cleared. 1: The sampled data is cleared. #(1)(1) INPUT ↓ Clear? (Y/N) ↓ ↓ To clear sampled data: Y INPUT To not clear sampled data: N INPUT "Data clear complete" appears **(Note)** Sampling will stop when the sampling data clear operation is executed. To restart sampling, the restart sampling operation will be executed or the power must be turned ON again.	1
SAMPLING SPEC	5	System	Designate the system that is not to be sampled. Set the system with a bit unit. bit 7 6 5 4 3 2 1 0 1: System 1, 0: System 2 The entire system will be sampled when 0 is set.	Set with a HEX.
	6	Spindle	Designate the spindle that is not to be sampled. Set the spindle with a bit unit. bit 7 6 5 4 3 2 1 0 1: 2nd spindle, 0: 1st spindle The entire system will be sampled when 0 is set.	Set with a HEX.
	7	Servo	Designate the axis that is not to be sampled. Set all servo axes with a bit unit regardless of the system. bit 7 6 5 4 3 2 1 0 1: 6th axis, 5th axis, 4th axis, 3rd axis, 2nd axis, 0: 1st axis The entire system will be sampled when 0 is set.	Set with a HEX.

Main item	#	Item	Details	Setting range
SAMPLING DATA SELECT	11	H/W CONFIG	Select whether to sample the hardware configuration data. 0: Sample. 1: Do not sample.	0/1
	12	S/W CONFIG	Select whether to sample the software configuration data. 0: Sample. 1: Do not sample.	0/1
	13	KEY HISTORY	Select whether to sample the key history. 0: Sample. 1: Do not sample.	0/1
	14	ALARM HISTORY	Select whether to sample the alarm history. 0: Sample. 1: Do not sample.	0/1
	15	PLC I/O HISTORY	Select whether to sample the PLC input/output signal history. 0: Sample. 1: Do not sample.	0/1
	16	AC ALARM HISTORY	Select whether to sample the AC input power error history. 0: Sample. 1: Do not sample.	0/1
	17	SPINDLE DATA1	Select whether to sample the spindle current waveform (speed FB, current FB, current command, position FB, position command, droop). 0: Sample. 1: Do not sample.	0/1
	18	SPINDLE DATA2	Select whether to sample the spindle current waveform (driver status 1, 3, 4). 0: Sample. 1: Do not sample.	0/1
	19	SERVO DATA1	Select whether to sample the servo current waveform (speed FB, current FB, current command, position FB, position command, droop). 0: Sample. 1: Do not sample.	0/1
	20	SERVO DATA2	Select whether to sample the servo current waveform (detector position Rn, Pn). 0: Sample. 1: Do not sample.	0/1
	21	MODAL	Select whether to sample the program and modal data. 0: Sample. 1: Do not sample.	0/1
	22	PROGRAM DATA	Select whether to sample the program sampling data. 0: Sample. 1: Do not sample.	0/1
	23	COORDI-NATE DATA	Select whether to sample the coordinate data. 0: Sample. 1: Do not sample.	0/1

7.12 NC Data Sampling

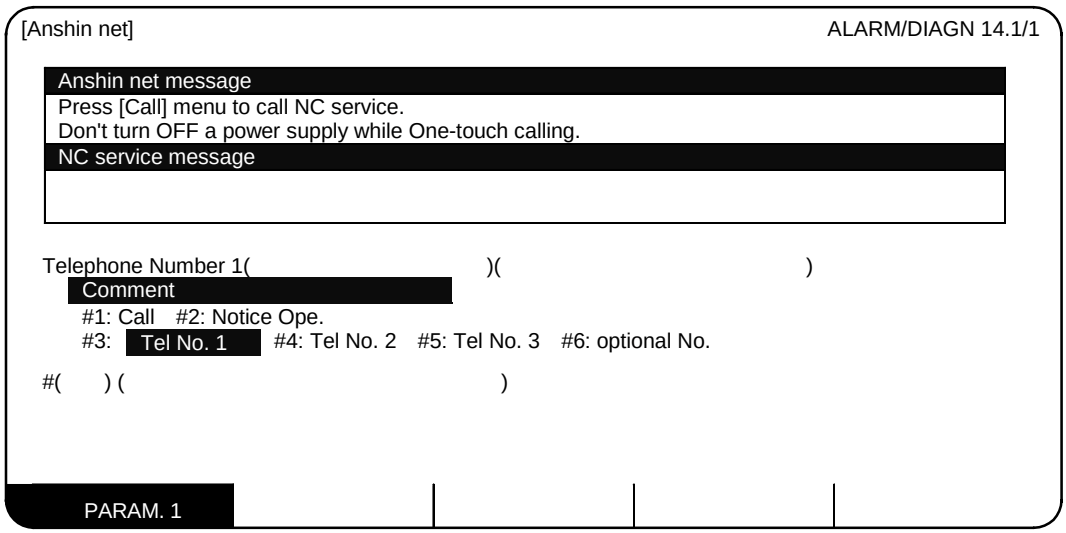
When the page key NEXT PAGE is pressed on the MELDASNET SUPPORT PARAMETER screen, the NC-DATA SAMPLING screen will open.



Refer to section "8.2 Data Sampling" in "III. Setup" for details on the items displayed on the screen and on data sampling.

7.13 Anshin-net

Pressing the menu key [Anshin net] displays the ANSHIN-NET screen.



Detailed information services and speedy maintenance are provided by connecting the Ryoden Koki Engineering (RKE) call center with the end user's CNC over a network.
A contract must be bound with Ryoden Koki Engineering, Co., Ltd. to use this service.

[Service menu]

(1) Anshin menu

Automatic alarm notification	The call center can see the state of an alarm occurrence during automatic operation. Quick measures can be taken even during unmanned operation.
One-touch call	The machine status information can be sent just by pressing the one-touch button. Expert engineers provide technical support based on the received information.
Periodic monitor	The warning messages are periodically monitored and analyzed to prevent faults.
Backup	The parameter data is periodically backed up to provide swift service even if the memory is lost, etc.

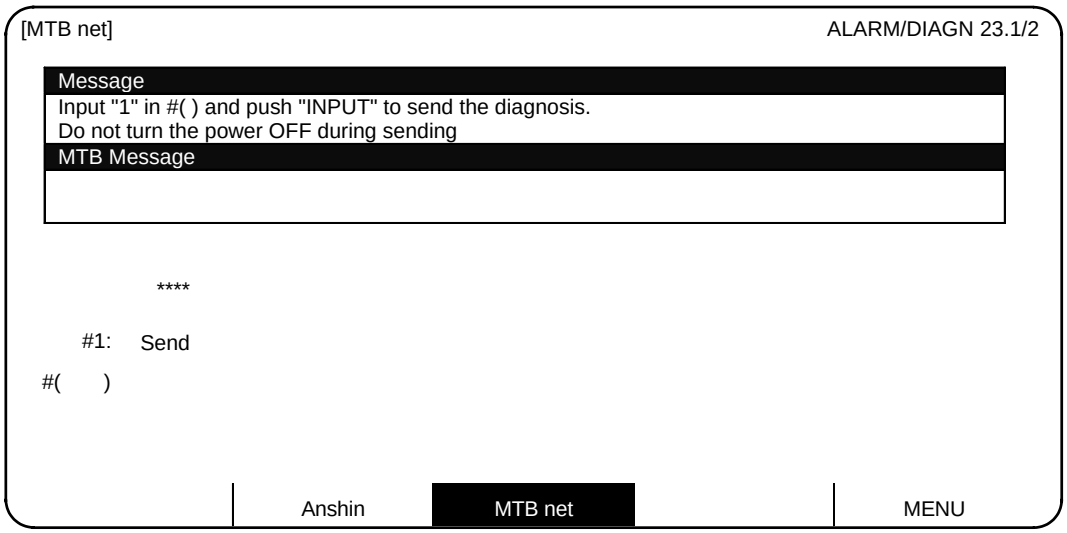
(2) Handy menus

Operator notification	If machining ends or an error such as an alarm stop occurs, the state is automatically notified to the operator's cell phone with audio messages.
Periodic monitor warning	A report created by tabulating the monitor data is sent to help with controlling or planning production plans.
Remote monitor	The machine's operation status can be understood by accessing the call center from a phone.
Sharing of machining data	The call center's server is opened to users for use as a medium to store programs.

Contact Ryoden Koki Engineering, Co., Ltd. for details.

7.14 MTB net

Pressing the menu key [MTB net] displays the MTB net screen.



MTB net is a function that connects the personal computer software "MELDAS Remote Diagnosis Tool Kit" (hereinafter, remote diagnosis tool kit) which Mitsubishi sells separately to machine makers and the NC with a communication line, and sends and receives the diagnosis information files required for service.

MTB net has the following two functions.

(1) Machine tool diagnosis data send function

When Send is selected on the MTB net screen, a connection is established with the remote diagnosis tool kit, and the diagnosis information is sent.

1)

Set "1" at the setting area on the MTB net screen, and then press the [INPUT] key.

☞ The connection to the remote diagnosis tool kit will start. After connecting, the files are sent in the background according to the requests from the remote diagnosis tool kit.

(Note) If a send operation is attempted during communication, an operation error (E05) will occur.

(2) Reception response function

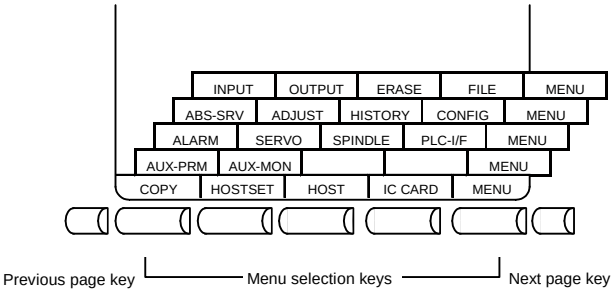
When a phone connection is established from the remote diagnosis tool, the diagnosis information is sent.
This function is carried out in the background, and does not require onscreen operations.

Contact the machine maker for more details on MTB.

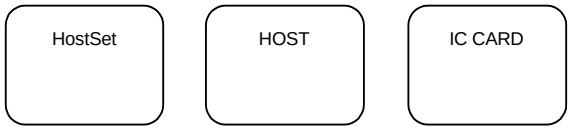
8. High-speed Program Server

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

Input/output menu (No.1 to 4)
Diagnosis menu (No.5 to 8)
Diagnosis menu (No.1 to 4)
Diagnosis menu (No.9 to 10)
Input/output menu (No.5) , Program sever menu (No.1 to 3)

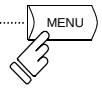


Program server menu.



DIAGN menu
No.1 to 10

Refer to the section on diagnosis.

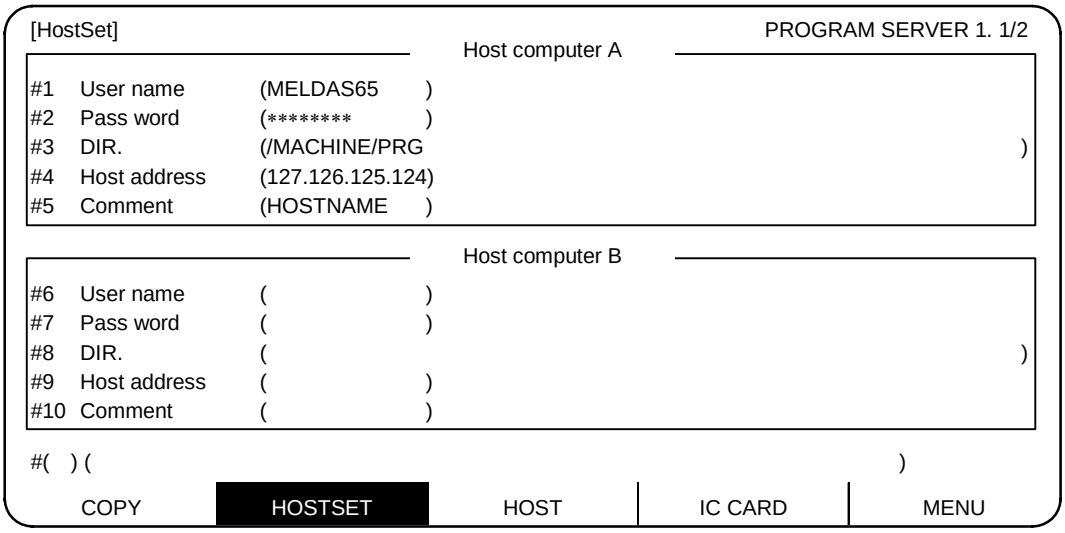


Input/Output menu

Refer to the section on data input/output.

8.1 Host Setting

When the menu key ⟩HOSTSET⟨ is pressed, the HostSet screen is displayed.



Up to four host computers (host computers A to D) can be set on the HostSet screen. Set host computers A and B on the first page, and host computers C and D on the second page.

(Note) Depending on the model, only one screen equivalent to host computer A will appear.

Item	Details	
		No. of input characters
#1 User name	Set the user name for logging into the host computer.	15
#2 Password	Set the user password for logging into the host computer.	15
#3 DIR.	Designate the host directory to access.	63
#4 Host address	Set the IP address of the host computer.	15
#5 Comment	Set the comment of the host computer.	10
#6 to #10	Same as #1 to #5.	

The following explanations indicate the case for setting the upper level (host computer A on page 1).

(Note) Designate the path under the open directory set in the host computer's FTP server for the directory designated in #3 and #8.
(The open directory is the directory that can be referred to by the client (NC).
The open directory is handled as the root directory (/) by the client.)

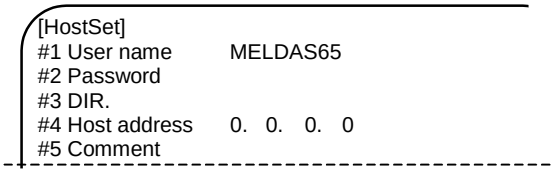
(Example) When open directory is "c:\PRG"
... The NC handles the above directory as the root directory (/).

8.1.1 Setting the User Name

(1) Input "1" in the setting area # ( ), and press the cursor key [→].

(2) Input the user name for logging in into the setting area ( ).
The user name can have up to 15 alphanumeric characters.
* Only uppercase alphabetic characters can be set.

(3) Press the [INPUT] key.



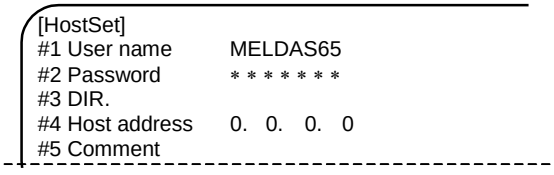
The setting area will be cleared, and the "#1 User name" data will be updated. The password will also be cleared.

8.1.2 Setting the Password

(1) Input "2" in the setting area # ( ), and press the cursor key [→].

(2) Input the user password for logging in into the setting area ( ). The password can have up to 15 alphanumeric characters.
* Only uppercase alphabetic characters can be set.

(3) Press the [INPUT] key.

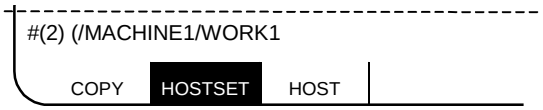


The setting area will be cleared, and the "#2 Password" data will be updated. The display will change to "********".

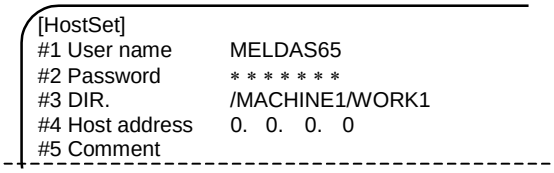
8.1.3 Designating the Directory

(1) Input "3" in the setting area # ( ), and press the cursor key [→].

(2) Designate the host computer directory as an absolute path in the setting area ( ). The directory can have up to 31 alphanumeric characters.
* Only uppercase alphabetic characters can be set.



(3) Press the [INPUT] key.



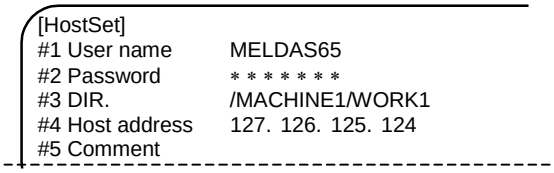
The setting area will be cleared, and the "#3 DIR." data will be updated.

8.1.4 Setting the Host Address

(1) Input "4" in the setting area # ( ), and press the cursor key [→].

(2) Input the host computer's IP address in the setting area ( ).

(3) Press the [INPUT] key.



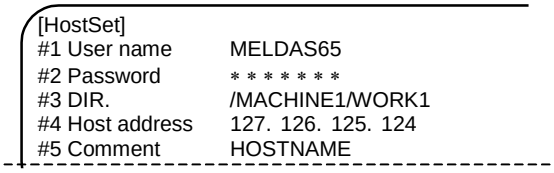
The setting area will be cleared, and the "#4 Host address" data will be updated.

8.1.5 Inputting a comment

(1) Input "5" in the setting area # ( ), and press the cursor key [→].

(2) Input the comment in the setting area ( ).
Up to ten alphanumeric characters can be input for the comment.
* Alphabetic characters can be input only in uppercase.

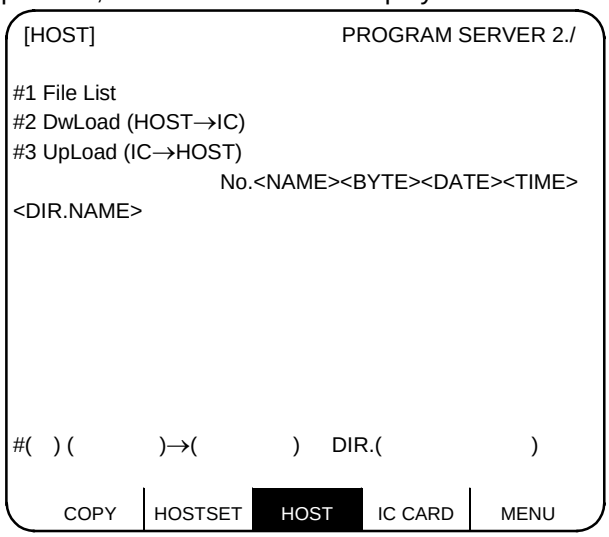
(3) Press the [INPUT] key.



The setting area will be cleared, and the "#5 Comment" data will be updated.

8.2 Host (Compatible with M60 Series)

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



Function	Details
#1 File List	Designate the host computer directory to be accessed. The list of the files in the designated directory can be displayed.
#2 DwLoad (HOST → IC)	Machining programs can be received from the host computer and saved in the IC card.
#3 UpLoad (IC → HOST)	Machining programs saved in the IC card can be transmitted to the host computer.

Item	Details
<DIR. NAME>	The directory name designated in "HOSTSET" screen is displayed (absolute path).
[File List]	The files in the designated directory are displayed as a list. Note that the subdirectory is not displayed.
<NAME>	The file name is displayed. If the file name is too long, the first 12 characters will be displayed.
<BYTE>	The file size is displayed as a byte unit. If the file size exceeds 99999999 bytes, "********" will display.
<DATE>	The date (following the host computer clock) that the file was saved is displayed. The display differs according to the saved period. • File saved more than a year ago → The year, month and date are displayed. (Example: 1999/02/02) • File saved within the year → The month and date are displayed. (Example: /02/02)
<TIME>	The time (following the host computer clock) that the file was saved is displayed. The display differs according to the saved period. • File saved more than a year ago → The time is not displayed. • File saved within the year → The time is displayed. (Example: 14:57)

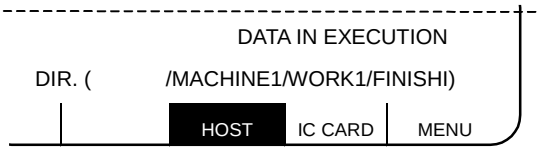
If all of the saved files cannot be displayed on one page, the list will cover several pages.
Display the previous and following pages by pressing the page keys ([PREVIOUS PAGE], [NEXT PAGE]).
The file list is displayed in the order of the information sent from the host computer.

8.2.1 Displaying the File List

(1) Input "1" in the setting area # ( ), and press the cursor key [→] three times.

(2) Designate the host computer directory as an absolute path in the setting area DIR. ( ). The directory can have up to 31 alphanumeric characters.
* Only uppercase alphabetic characters can be set.
* The designated directory is held until the power is turned OFF.
* The directory designation can be omitted.

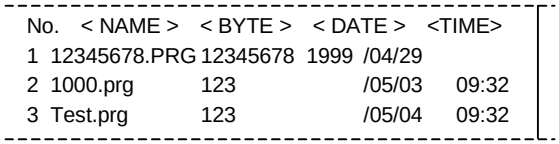
(3) Press the [INPUT] key.



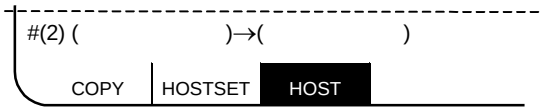
The message "DATA IN EXECUTION" will appear. When the input is completed, the message
"DATA IN COMPLETE" will appear, and the setting area will be cleared. The displayed file list will be updated.

8.2.2 Downloading (IC to host)

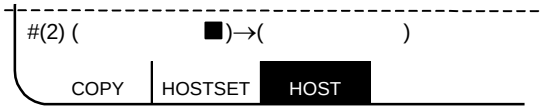
An example of registering the machining program file "Test.prg (file No. 3)" saved in the host computer as "2000.prg" in the IC card is explained in this section.



(1) Input "2" in the setting area # ( ), and press the cursor key [→].

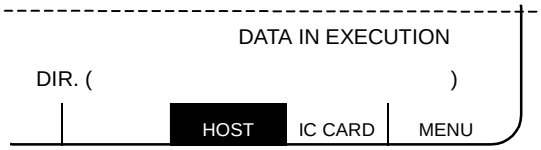


(2) Input the file No. of file name to be uploaded from the file list in the setting area ( ), and press the cursor key [→].
* Up to 12 uppercase alphabetic characters can be designated as the file name.
* The values 1 to 10 are used only to designate the "file No.".



(3) Input the program No. to enter into the IC as a number in the following setting area. Up to eight digits can be designated for the program No.

(4) Press the [INPUT] key.



The message "DATA IN EXECUTION" will appear. When the transmission is completed, the message "DATA IN COMPLETE" will appear, and the setting area will be cleared.

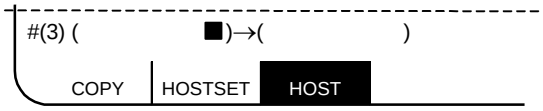
(Note 1) The machining program being input from the host computer cannot be automatically run. (The host computer machining program cannot be searched.)
(Note 2) The input cannot be canceled.
(Note 3) Operations on the screen cannot be carried out during the input.

8.2.3 Uploading (Host to IC)

An example of saving the machining program "3000.prg" registered in the IC card as "TESTCUT2" in the host computer is explained in this section.

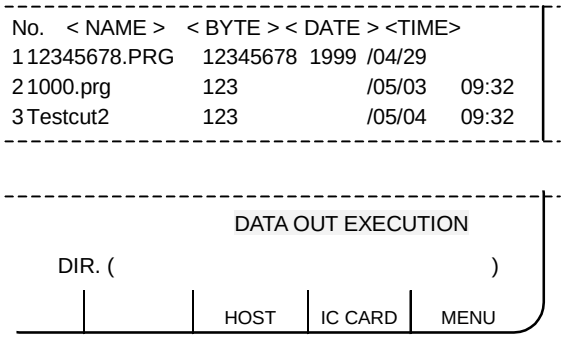
(1) Input "3" in the setting area # ( ), and press the cursor key [→].

(2) Input the number of the program to be transmitted from the IC card to the host computer in the setting area ( ), and press the cursor key [→].
* Input only the program No. (The extension ".PRG" does not need to be input.)



(3) Input the name of the file to be saved in the host computer in the following setting area . Up to 12 uppercase alphabetic characters can be designated as the file name.
If the input is omitted, the program file name will be the same as that designated in step (2).

(4) Press the [INPUT] key.



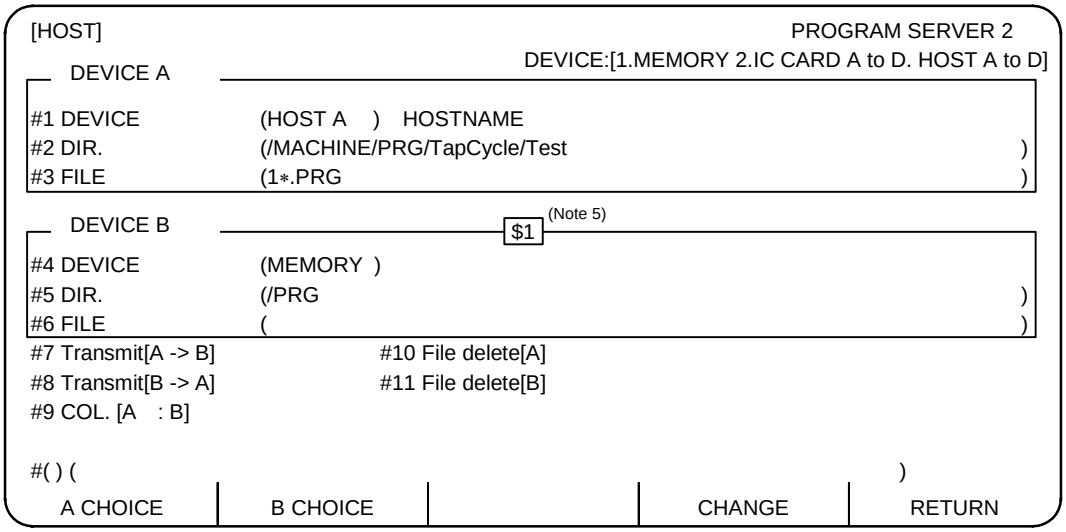
The message "DATA OUT EXECUTION" will appear. When the transmission is completed, the message "DATA OUT COMPLETE" will appear, and the setting area will be cleared. The displayed file list will be updated.

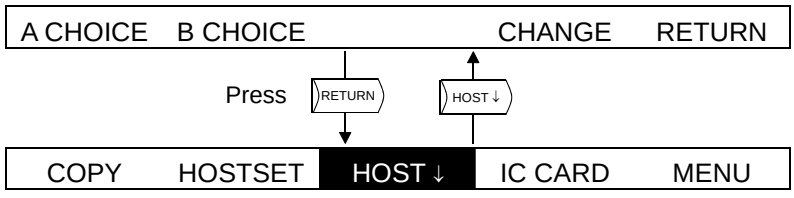
(Note 1) The transmission cannot be canceled.
(Note 2) Operations on the screen cannot be carried out during the transmission.

8.3 Host (Compatible with M60S Series)

8.3.1 Host Communication Screen

When the menu key [HOST] is pressed, the Host Communication screen is displayed.



Menu	Function
[A CHOICE]	The "File Selection" screen opens, and the list of device files set in device A is displayed.
[B CHOICE]	The "File Selection" screen opens, and the list of device files set in device B is displayed.
[CHANGE]	Changes the alphabet key input between uppercase and lowercase. Use this when setting the directory and file. The menu is highlighted during selecting lowercase.
[RETURN]	Changes the display menu. 

Item		Details	No. of input characters
DEVICE A	#1 DEVICE	Set the device. The "HOST A" is set immediately after the power is turned ON. When host A to D is set, the comment set on the "HOST COMMUNICATION screen" will appear.	1 **(Note 1)**
	#2 DIR.	Set the directory.	63 **(Note 2)**
	#3 FILE	Set the file. Several files can be set by using "∗".	63
DEVICE B	#4 DEVICE	Set the device name. The "MEMORY" is set immediately after the power is turned ON. When host A to D is set, the comment set on the "HOST COMMUNICATION screen" will appear.	1 **(Note 1)**
	#5 DIR.	Set the directory.	63 **(Note 2)**
	#6 FILE	Set the file. Several files can be set by using "∗".	63
#7 Transmit[A->B]		The file (#3) set in device A is sent to device B. **(Note 3)**	
#8 Transmit[B<-A]		The file (#6) set in device B is sent to device A. **(Note 3)**	
#9 COL. [A:B]		The file (#3) set in device A and the file (#6) set in device B are compared. **(Note 3)**	
#10 File delete[A]		The file (#3) set in device A is erased. **(Note 4)**	
#11 File delete[B]		The file (#6) set in device B is erased. **(Note 4)**	

(Note 1) Set the device as shown below.

Setting value	Set device name
1	Memory (NC internal memory)
2	IC card
A	Host A
B	Host B
C	Host C
D	Host D

(Note 2) Up to 63 characters can be input at once. However, if a relative path is input, 63 characters could be exceeded. In this case, the last 62 characters will be displayed, and the first character will be displayed as "¯" (tilde).
(Up to 127 characters can be set for the path.)

(Example) /MACHINE1/123456789/123456789/123456789/123456789/123456789/1234
↓
¯ACHINE1/123456789/123456789/123456789/123456789/123456789/1234

(Note 3)

(1) If the same device is set for device A and device B, an error will occur.

(2) Set the IC card device's file name as an identifier (max. 8 characters) + extension (max. 3 characters). If a file name exceeding this number of characters (a long file name), an operation error (E84) will occur, and the file will not be transmitted.

(Example) File name usable for IC card device
OK (transfer possible) 12345678.PRG
NG (transfer not possible) 123456789.PRG

(3) The directory created in the IC card (created on the personal computer) can have up to 18 hierarchies. If a file is transferred or a directory is moved (file list displayed) to a directory having more than 18 hierarchies, an operation error (E319) will occur, and the file cannot be transferred or listed.

(4) There is an upper limit to the number of files (approx. 200 files: differs for each IC card) that can be created in the IC card's root directory (directory immediately below the IC card drive). If files exceeding this upper limit are transferred, an operation error (E51) will occur, and the file will not be transferred.

(Note) Normal file transfer may not be possible with an IC card used in (2) to (4) above. (This also applies when operation are carried out on the PC.)
If an operation error (E51) occurs, format the IC card.

(5) When the memory device programs and IC card or host device files are compared, the following line will be ignored.

Line not subject to comparison
- Line containing a "%" code (when "%" is described at head of file)
- "0" No. + comment line

Use the CR+LF code for the carriage return code of the IC card or host device file. Files that do not use the CR+LF code for carriage return cannot be compared. An operation error (E35) will occur if compare is attempted.

(Note 4) Only the files in the memory and IC card can be erased.

(Note 5) When using the 2-part system, the device name is memory and the directory is /PRG/USER, the name of the selected system can be displayed. Whether to show or hide name of the selected system can be switched with the parameters.

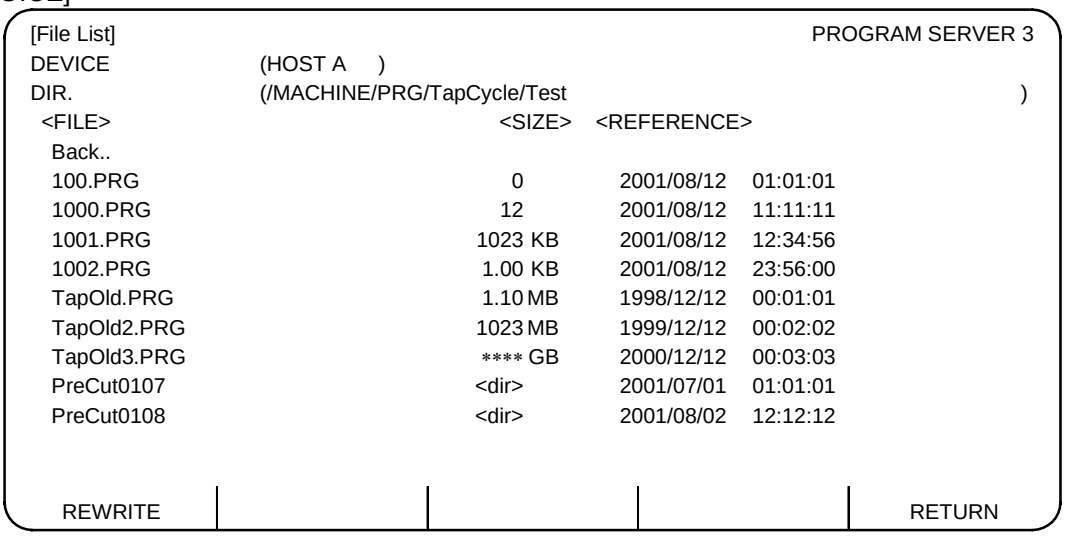
#1050 MemPrg	#1285 ext21/bit0	#1286 ext22/bit0	Details
0, 2, 4, 6	-	-	The name of the selected system is not displayed.
1, 3, 5, 7	OFF		The name of the selected system is displayed.
		OFF	
	ON	ON	The name of the selected system is not displayed.

(Note 6) When parameter (#1220 aux04 (bit7)) is set, automatic start can be executed while communicating with the host (during input/output). However, the machining performance could be affected.

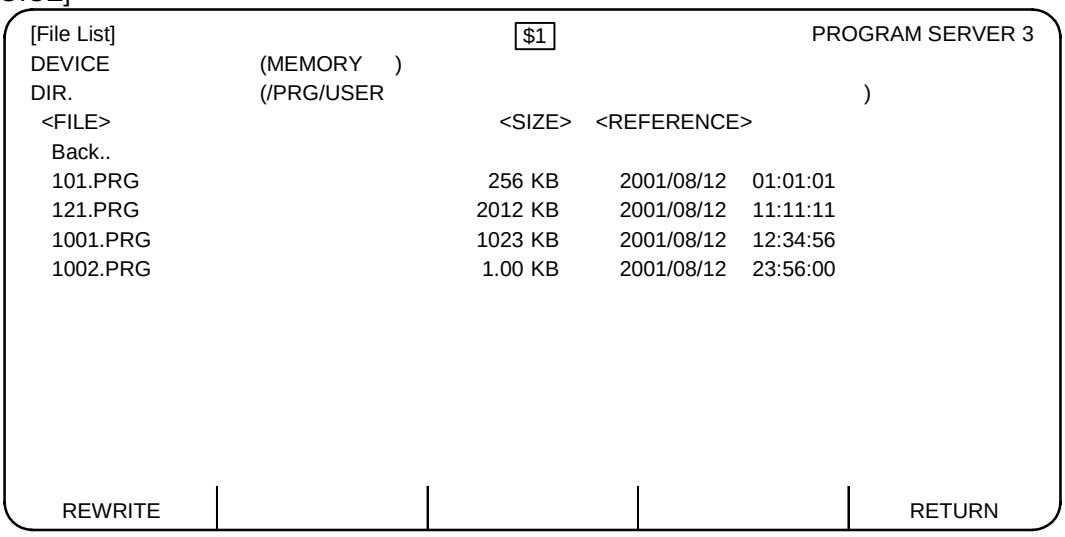
8.3.2 File Selection Screen

When the menu key [A CHOICE] or [B CHOICE] is pressed on the Host Communication screen, the File Selection screen will appear.

[A CHOICE]



[B CHOICE]



If there are too many files saved to be listed on one page, the files will be displayed on several pages. Feed the pages by pressing the page keys ([PREVIOUS PAGE], [NEXT PAGE]).
If the host files are displayed, the information is displayed in the order that it was sent from the host computer. The time displayed in the "<REFERENCE>" field also follows the host computer's clock.
When the IC card files are displayed, the files are displayed in the order that they were saved in the IC card's file system. If the files in the IC card are erased and files are output to the IC card repeatedly, the files will be written into the area opened by erasing the files. Thus, the files may not be displayed in the actually saved order.

Menu	Function
[REWRITE]	The contents displayed in the file list are redisplayed.
[RETURN]	The "Host Communication" screen is opened.

(Note) If another screen is displayed while the files are listed and then the File Selection screen is opened again, the files will not be listed. Press the [REWRITE] menu to display the files again.

Item		Details
DEVICE		The designated device name is displayed. The device is set on the "Host Communication" screen.
DIR.		The directory name is displayed with the full path. **(Note 1)** The directory name display will be updated when the following step is carried out. • Select Directory from the file list. • Select "Back.." from the file list.
File List	<FILE>	The files and directories in the designated directory are displayed. **(Note 2)** The selected file is highlighted. Another file can be selected using the ↑ and ↓ keys. If the device is "MEMORY", the file name and directory name will be fixed names. (Refer to "8.3.3 NC data file name".)
	<SIZE>	The file size is displayed. (Unit: Byte) The unit display will change according to the file size. • When 1024Byte is exceeded: KB • When 1024KB is exceeded: MB • When 1024MB is exceeded: GB If 4.0GB is exceeded, the size will be displayed as "****GB". A directory will be displayed as "<dir>".
	<REFERENCE>	The date and time (hour, minute, second) that the file was saved are displayed. **(Note 3)** If the device is "MEMORY", the following will be displayed. • When file is a program: A comment will display • When file is not a program: There will be no display.

(Note 1) If 63 characters are exceeded, the last 62 characters will be displayed, and the first character will be displayed as "¯" (tilde). (Up to 127 characters can be set for the path.)

(Example) /MACHINE1/123456789/123456789/123456789/123456789/123456789/1234
↓
¯ACHINE1/123456789/123456789/123456789/123456789/123456789/1234

(Note 2) If 36 characters are exceeded, the last 35 characters will be displayed, and the first character will be displayed as "¯" (tilde). (When a file name having more than 64 characters is selected, an error will occur during the transfer of the file. If a file with 63 characters or less and a "¯" at the head is selected, it will be correctly displayed on the Host Communication screen.)

(Example) 12345678901234567890123456789012345.PRG
↓
¯345678901234567890123456789012345.PRG

(Note 3) If the host computer's file type is UNIX, and the file was saved in the previous year, the time will not be displayed.

(Note 4) When using the 2-part system with the B selection screen, the device name is memory and the directory is /PRG/USER, the name of the selected system can be displayed. Whether to show or hide name of the selected system can be switched with the parameters.

#1050 MemPrg	#1285 ext21/bit0	#1286 ext22/bit0	Details
0, 2, 4, 6	-	-	The name of the selected system is not displayed.
1, 3, 5, 7	OFF		The name of the selected system is displayed.
		OFF	
	ON	ON	The name of the selected system is not displayed.

8.3.3 NC Data File Name

When memory is selected as the device on the File Selection screen, a fixed directory name and file name will be displayed. (When host A to D or the IC card is selected, the random directory name and file name will appear.)
The NC data can be input and output by selecting each file.

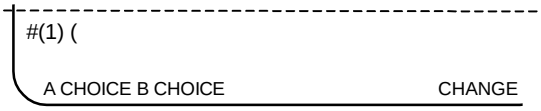
The directory names (file names) displayed in the File List when memory is selected are shown below.

Directory name	File name	Data type
/PRG/USER/	(A list of the programs saved in the NC memory is displayed)	Machining program
/PRM/	ALL.PRM	Parameters
/DAT/	COMMON.VAR	Common variables
	TOOL.OFS	Tool compensation data
	TLIFE.TLF	Tool life control data
	SRAM.BIN	SRAM data
	EXTSRAM.BIN	Extension SRAM data
/LAD/	USERPLC.LAD	User PLC ladders

8.3.4 Using the Host Communication Screen

(1) Setting the device

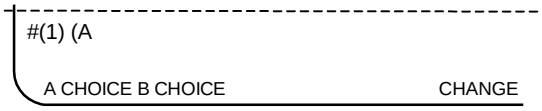
1) Input "1" in the setting area # ( ), and press the cursor key [→]. (Input "4" when setting device B.)



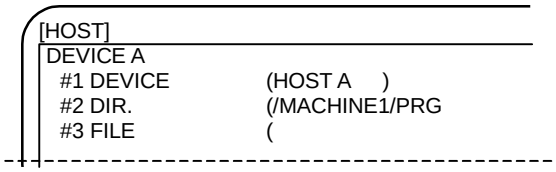
2) Designate the device in the setting area ( ).

(Example)

- To select memory → 1
- To select IC card → 2
- To select host A → A
- To select host B → B
  :
  :
- To select host D → D



3) Press the [INPUT] key.

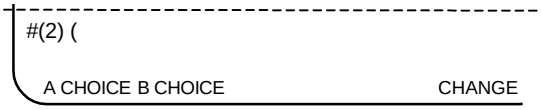


The setting area will be cleared, and the "#1 DEVICE" data will be updated.
The previously selected files will be set in "#2 DIR." and "#3 FILE".

1. The previously selected files will each be saved in host A to D.
2. If host A to D is selected immediately after the power is turned ON, the directory set on the Host Communication screen will be set.

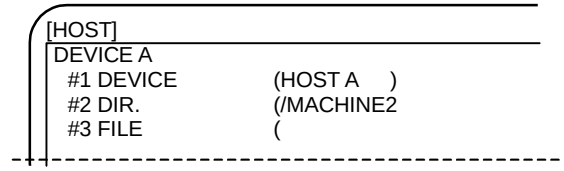
(2) Setting the directory 1 (direct input)

1) Input "2" in the setting area # ( ), and press the cursor key [→]. (Input "5" when setting device B.)



2) Designate the directory name in the setting area ( ).
   - To select the slave directory of the directory set in #2, input only the directory name. (Relative path)
   - To change the directory set in #2, input the directory name after "/". (Absolute path)

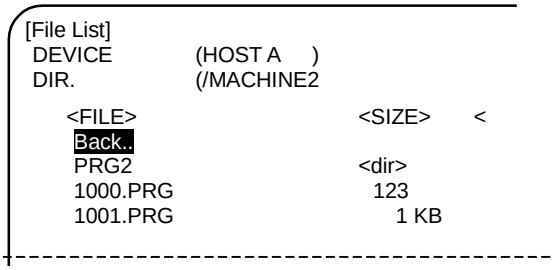
3) Press the [INPUT] key.



The setting area will be cleared, and the "#2 DEVICE" data will be updated.

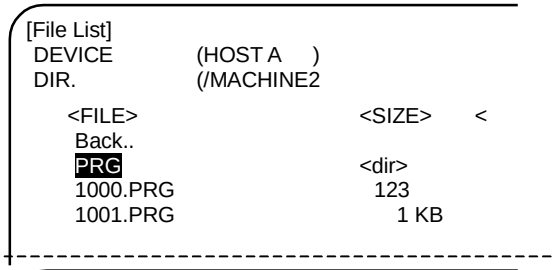
(3) Setting the directory 2 (selective input)

1) Press the menu key [A CHOICE]. (Press [B CHOICE] when setting device B.)

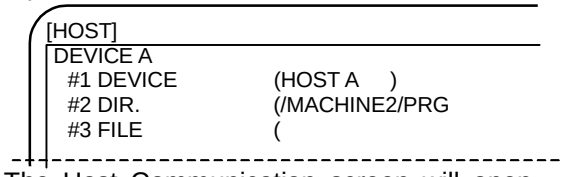


The File Selection screen will open.

2) Press the arrow keys [↑] and [↓], and move to the target directory. (Refer to section "8.3.5 Using the File Selection Screen" for details on operations in the File Selection screen.)



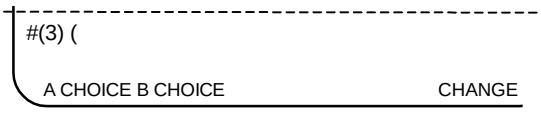
3) Press the [INPUT] key, and then press the [RETURN] key.



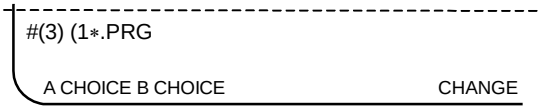
The Host Communication screen will open, and "#2 DIR." will be updated.

(4) Setting the file 1 (direct input)

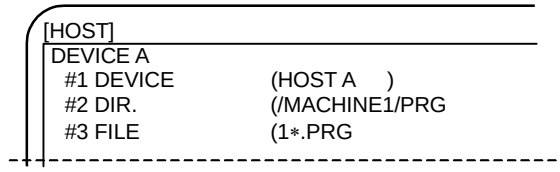
1) Input "3" in the setting area # ( ), and press the cursor key [→].
(Input "6" when setting device B.)



2) Designate the file name in the setting area ( ).
(To select multiple files, use "∗" (asterisk).)
(Example) To select all files starting from "1" and ending at ".PRG" → "1∗.PRG"
(Only one asterisk can be used.)



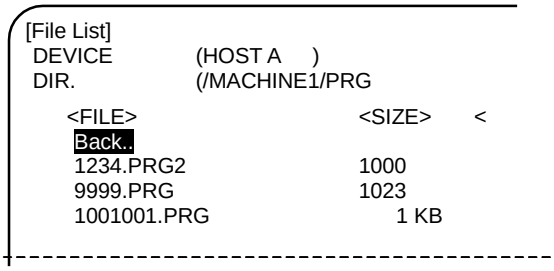
3) Press the [INPUT] key.



The setting area will be cleared, and the "#3 FILE" data will be updated.

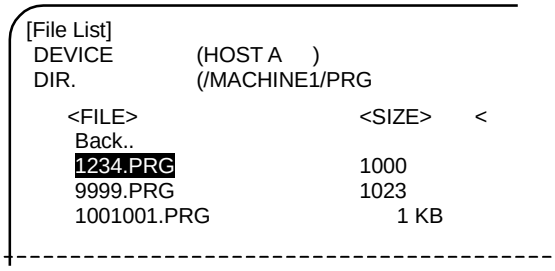
(5) Setting the file 2 (selective input)

1) Press the menu key [A CHOICE].
(Press [B CHOICE] when setting device B.)

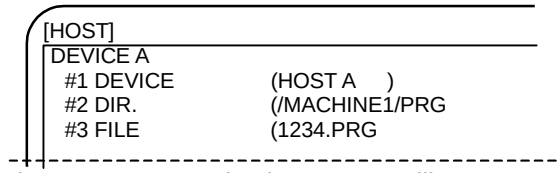


The File Selection screen will open.

2) Press the arrow keys [↑] and [↓], and move to the target file.
(Refer to section "8.3.5 Using the File Selection Screen" for details on operations in the File Selection screen.)



3) Press the [INPUT] key, and then press the [RETURN] key.

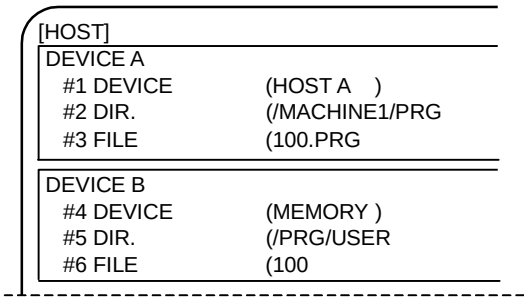


The Host Communication screen will open, and "#3 FILE" will be updated.

(6) Transferring files

1) Set the transfer origin and transfer destination files.
(Refer to steps (1) to (3) in the previous section for the operation methods.)

If the transfer destination file is omitted, the file will be transferred with the same name as the transfer origin file. Note that when inputting and outputting the machining programs in the memory device, the extension (.PRG) will be added or deleted automatically.

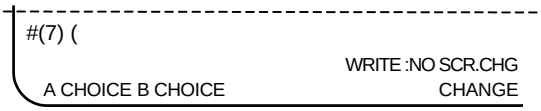


Automatically determining the transfer origin file name

Transfer origin device: /file name	Transfer destination device: /file name	Transfer destination file name	Explanation
Memory: /PRG/USER/100	Host A: /	100.PRG	Transferred as same file name + extension is automatically added
Memory: /PRG/USER/100	IC card: /	100.PRG	Transfer to the IC card is the same as the host
Memory: /PRG/USER/100	Host A: /200	200.PRG	Extension is automatically added
Memory: /PRG/USER/100	Host A: /200.TXT	200.TXT	Transferred with designated file name
Host A: /100.PRG	Memory: /PRG/USER	100	Extension (PRG) is automatically deleted
Memory: /PRG/DAT/TOOL.OFS	Host A: /	TOOL.OFS	Transferred as same file name
Memory: /PRG/DAT/TOOL.OFS	Host A: /OFFSET	OFFSET	Extension is not added automatically
Host A: /100.PRG	IC card: /	100	Extension is not added automatically between host and IC card

2) Input "7" in the setting area # ( ), and press the [INPUT] key.

When using #7 Transmit [A→B], device A will be the transfer origin.
When using #8 Transmit [B→A], device B will be the transfer origin.



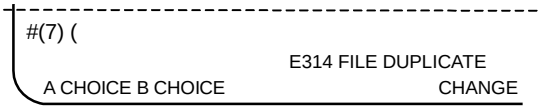
The transfer will start, and the message "WRITE :NO SCR.CHG " will appear.
When the transfer is completed, the message "WRITE COMPLETE" will appear, and then "DATA IN COMPLETE" or "DATA OUT COMPLETE" will appear.

(Note 1) When transferring multiple files with the "∗" designation, the message "WRITE :NO SCR.CHG" and "WRITE COMPLETE" will appear each time a file is transferred. "DATA IN COMPLETE" or "DATA OUT COMPLETE" will appear when the last file has been transferred.

An operation error will occur in the following cases.
- When the transfer origin file cannot be found. (E313)
- When a file with the same name is found at the transfer destination. (E314)
- When transfer was not possible. (The host did not start up, etc.)

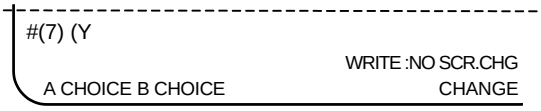
When an operation error (E314) occurs, the file can be overwritten.

[Overwriting a file]



When an operation error (E314) occurs, the cursor will move to the second setting area.

1) Input "Y" in the second setting area, and then press the INPUT key.



Transfer and overwriting of the file will start, and the message "WRITE :NO SCR.CHG" will appear.
When the transfer is completed, the message "WRITE COMPLETE" will appear, and then "DATA IN COMPLETE" or "DATA OUT COMPLETE" will appear.

(Note 1) When transferring multiple files with the "∗" designation, an operation error (E314) will occur each time a duplicate file is found. Note that if "A" is input instead of "Y" when the operation error (E314) occurs, an operation error will not occur even if subsequent files are duplicated, and instead the files will be overwritten.

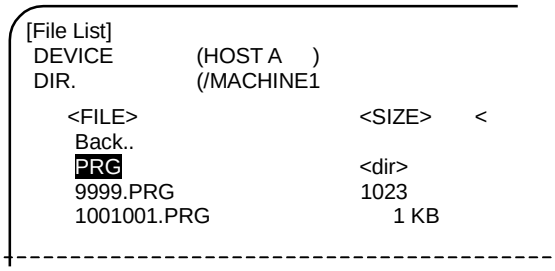
(Note 2) If a character other than "Y" or "A" is input when an operation error (E314) occurs, the file transfer will stop at that point.

8.3.5 Using the File Selection Screen

(1) Moving the directory

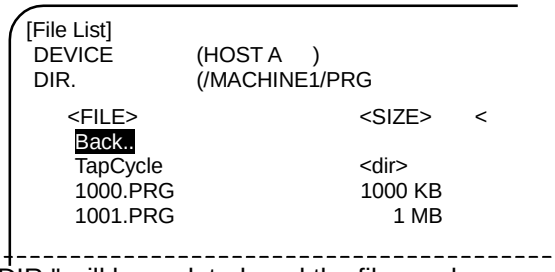
(Example 1) To move from directory "/MACHINE1" to directory "/MACHINE1/PRG/TapCycle"

1) Using the arrow keys [↑] and [↓], select the target directory ("PRG" for this example).
(If there are multiple directories (or files) in "/MACHINE1" and do not appear on one page, the target directory may not be displayed.
Press the page keys ([PREVIOUS PAGE], [NEXT PAGE]) to scroll the pages and display the required information.)



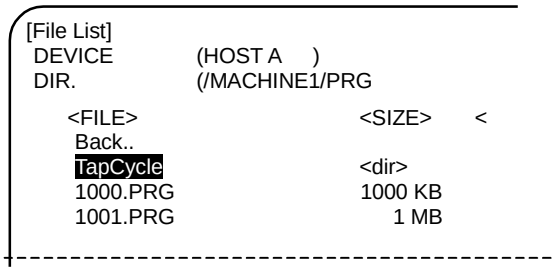
The selected directory (or file) will be highlighted.

2) Press the [INPUT] key.

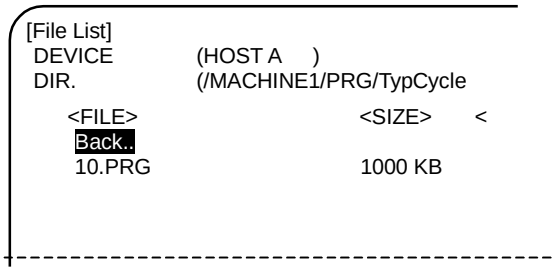


"DIR." will be updated, and the files under "/MACHINE1/PRG" will appear in the file list at the same time.

3) Press the arrow keys [↑] and [↓] again to select the target directory ("TapCycle" in this example).



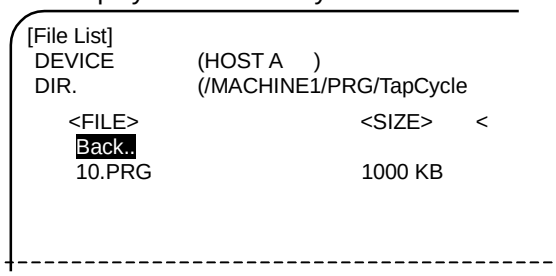
4) Press the [INPUT] key.



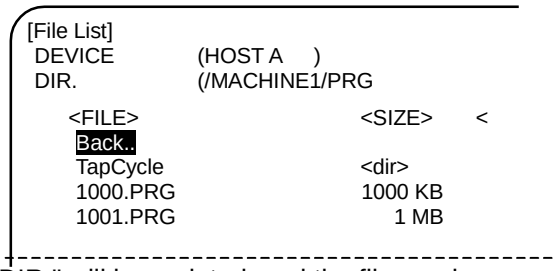
"DIR." will be updated, and the files under "/MACHINE1/PRG/TapCycle" will appear in the file list at the same time.

(Example 2) To move from directory "/MACHINE1/PRG/TapCycle" to directory "/MACHINE1"

1) Press the arrow key [↑], and select "Back..".
(Even if a page other than the first page has been displayed by pressing the page key ([PREVIOUS PAGE], [NEXT PAGE]), "Back.." will always appear at the top line.)

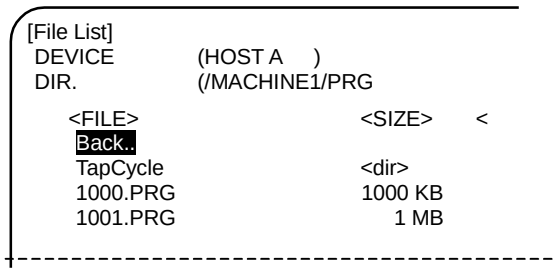


2) Press the [INPUT] key.

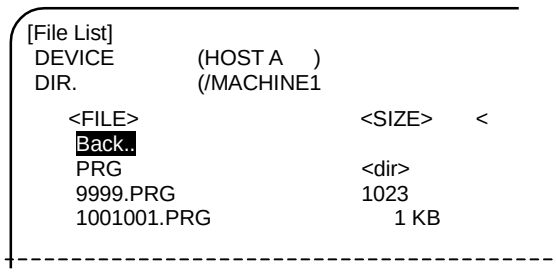


"DIR." will be updated, and the files under "/MACHINE1/PRG" will appear in the file list at the same time.

3) Press the arrow key [↑] again, and select " Back..".
("Back.." will appear immediately after the directory is moved.)



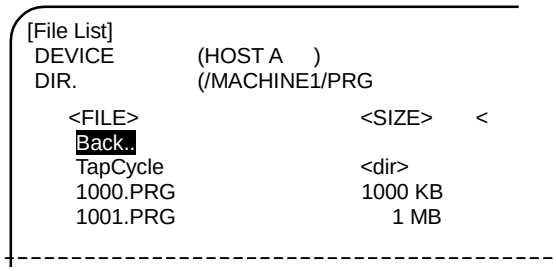
4) Press the [INPUT] key.



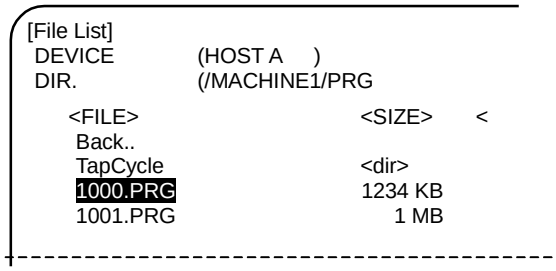
"DIR." will be updated, and the files under "/MACHINE1" will appear in the file list at the same time.

(2) Selecting a file/directory

1) Move to the directory containing the target file.
Refer to "8.3.5 Using the File Selection screen" (1) Moving the directory for details on moving.

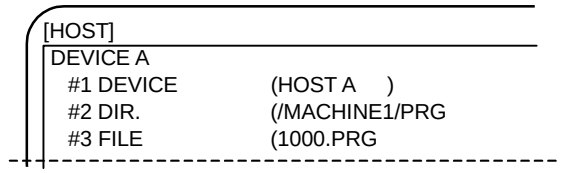


2) Using the arrow keys [↑] and [↓], select the target file.
(If there are multiple directories (or files) and do not appear on one page, the target file may not be displayed.
Press the page keys ([PREVIOUS PAGE], [NEXT PAGE]) to scroll the pages and display the required information.)



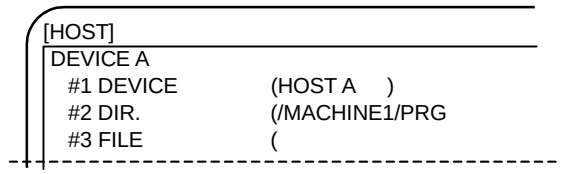
The selected directory (or file) will be highlighted.

3)-1 After pressing the [INPUT] key, press the menu key [RETURN].



The Host Communication screen will open, and "#2 DIR." and "#3 FILE" will be updated.

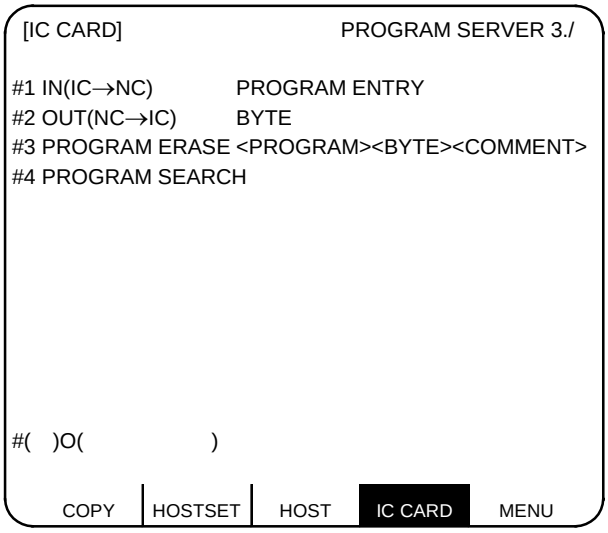
3)-2 Press the menu key [RETURN].



The Host Communication screen will open, and "#2 DIR." will be updated.

8.4 IC Card

When the menu key [IC CARD] is pressed, the IC CARD screen is displayed.
A list of the machining programs in the IC card will appear on the IC Card Input/Output screen.
The machining programs can be input, output, deleted, searched and listed on this screen.



Function	Details
#1 IN (IC → NC)	Selects the machining program input operation (IC card → NC memory).
#2 OUT (NC → IC)	Selects the machining program output operation (NC memory → IC card).
#3 PROGRAM ERASE	Erases the machining program in the IC card.
#4 PROGRAM SEARCH	Searches for machining programs saved in the IC card.

Item	Details
PROGRAM ENTRY	The number of machining programs saved in the IC card is displayed.
BYTE	The remaining number of bytes that can be saved in the IC card is displayed.
<PROGRAM>	The machining program No. (1 to 99,999,999) saved in the IC card is displayed.
<BYTE>	The number of machining program bytes saved in the IC card is displayed.
<COMMENT>	A machining program comment (up to 18 characters) is displayed.

(Note 1) The list is displayed in the order that the IC cards are saved. If the files in the IC card are erased and files are output to the IC card repeatedly, the files will be written into the area opened by erasing the files. Thus, the files may not be displayed in the actually saved order.

(Note 2) When using the 2-part system, the screen operations can be disabled with the parameters.

#1050 MemPrg	#1286 ext22/bit0	Details
0, 2, 4, 6	-	When using the M system, all operations are possible. When using the L system, operations other than operation search are possible. An error (E60) will occur if operation search is attempted.
1, 3, 5, 7	OFF	Input/output of the machining programs registered in the System 1 memory is possible.
	ON	An error (E05) will occur if input/output is attempted.

8.4.1 Inputting a Machining Program from the IC Card (IC to NC)

The following operations are used to input the machining program.

(1) Set #1 and the program No. (1 to 99,999,999).

(Example) To input program 1000 from the IC card to the NC memory.
(1) 0 (1000)

```
# ( 1 ) 0 (   1000)
```

(2) Press the [INPUT] key.
The program input will start, and "DATA IN EXECUTION" will appear in the message area.

```
                                DATA IN EXECUTION
# ( 1 ) 0 (   1000)
```

(3) When the input is completed, "DATA IN COMPLETE" will appear in the message area, and the program No. setting area will become blank.

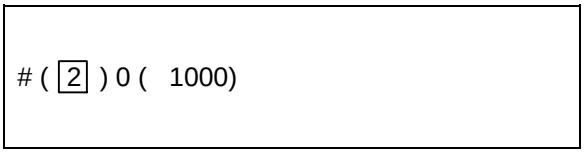
```
                                DATA IN COMPLETE
# (  ) 0 (       )
```

(Note1) Programs being called by the M198 command cannot be input.
(Note2) The screen cannot be changed during data input.

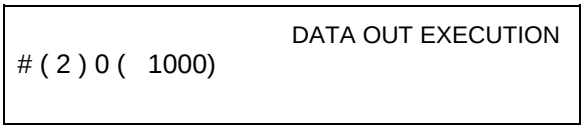
8.4.2 Outputting a Machining Program to the IC Card (NC to IC)

The following operations are used to output the machining program.

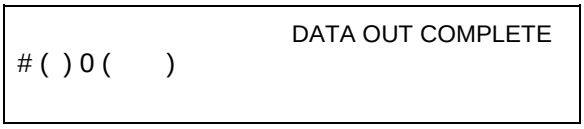
(1) Set #2 and the program No. (1 to 99,999,999).
(Example) To output program 1000 from the NC memory to the IC card.
(2) 0 (1000)



(2) Press the INPUT key.
The program output will start, and "DATA OUT EXECUTION" will appear in the message area.



(3) When the output is completed, "DATA OUT COMPLETE" will appear in the message area, and the program No. setting area will become blank.



(Note1) The screen cannot be changed during data output.

8.4.3 Erasing a Machining Program in the IC Card

The following operations are used to erase a machining program.

(1) Set #3 and the program No. (1 to 99,999,999) to be erased.
(Example) To erase program 1000.
(3) 0 (1000)

```
# ( 3 ) 0 (   1000)
```

(2) Press the [INPUT] key.
The program erasing will start, and "ERASE EXECUTION" will appear in the message area.

```
                          ERASE EXECUTION
# ( 3 ) 0 (   1000)
```

(3) When the erasing is completed, "ERASE COMPLETE" will appear in the message area, and the program No. setting area will become blank.

```
                          ERASE COMPLETE
# (  ) 0 (       )
```

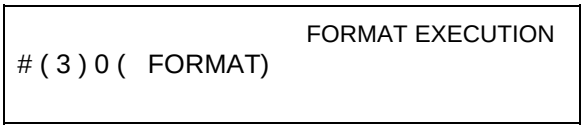
(Note 1) Programs being searched with "#4 PROGRAM SEARCH" cannot be erased.
(Note 2) Programs being called by the M198 command cannot be erased.

8.4.4 Formatting the IC Card

Format the IC card with the following operations to erase all machining programs in the IC card or to format an IC card.

(1) Set #3 and (FORMAT) in the program No. setting area.

(2) Press the INPUT key.
The formatting will start, and the message "FORMAT EXECUTION" will appear in the message area.



(3) When the formatting is completed, the program No. setting area will become blank.

(Note 1) Formatting is not possible while the program is running.

8.4.5 Searching for a Machining Program in the IC Card

The machining program is searched for with the following operations.

(1) Set #4 and the program No. (1 to 99,999,999) to be searched.
(Example) To search for program 1000.
(4) 0 (1000)

```
# ( 4 ) 0 (   1000)
```

(2) Press the [INPUT] key.
The program search will start, and "SEARCH EXECUTION" will appear in the message area.

```
                              SEARCH EXECUTION
# ( [4] ) 0 (   1000)
```

(3) When the search is completed, "SEARCH COMPLETE" will appear in the message area, and the program No. setting area will become blank.

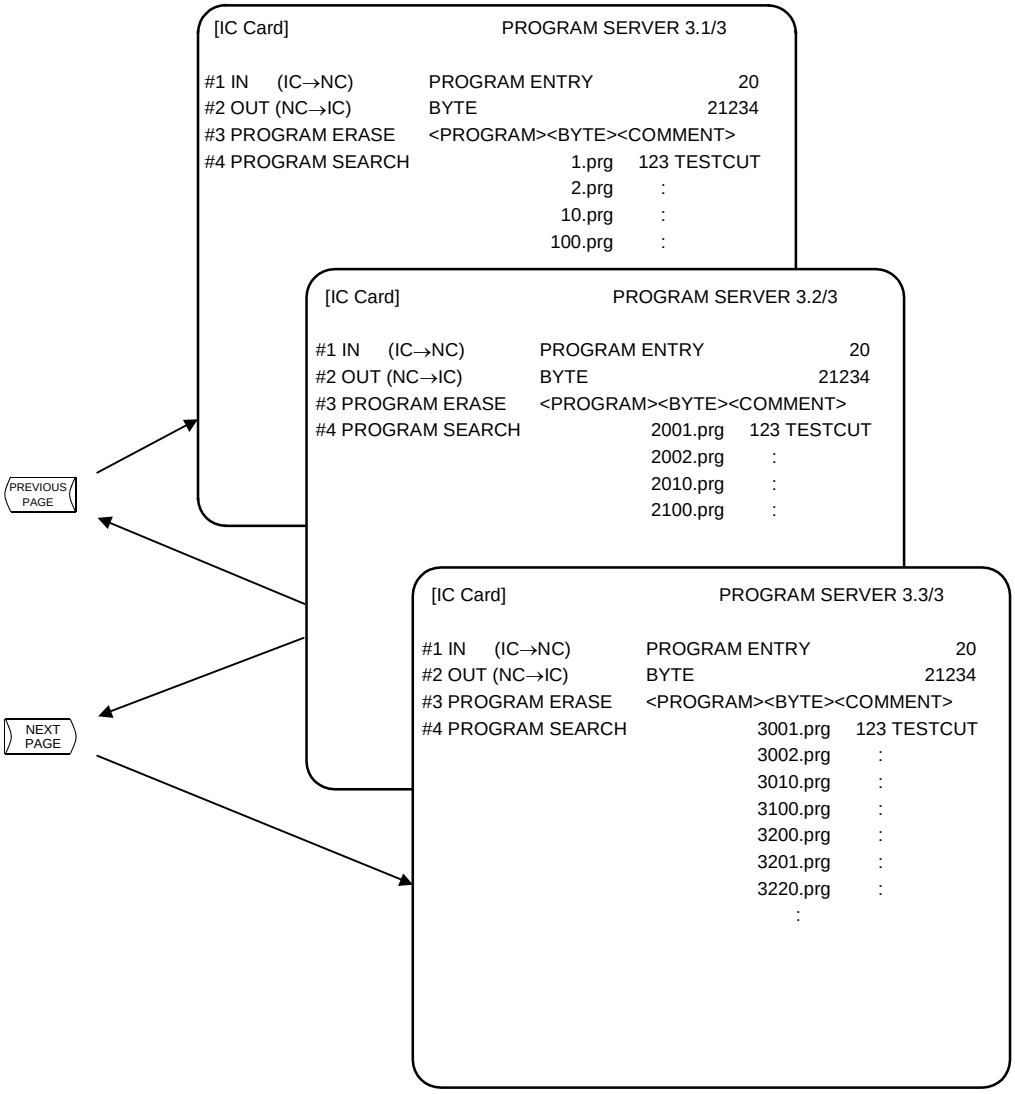
```
                              SEARCH COMPLETE
# (  ) 0 (       )
```

(4) To start the program that was searched and found, press the Automatic Start switch in the tape mode.
The search state is held until reset is input. (When reset, the program must be searched for again.)

(Note) Searching is not possible unless the machining program format is correct. (The machining program format is explained later.)

8.4.6 Listing the Machining Programs in the IC Card

The machining programs can be listed with the following operations.
If all of the saved machining programs cannot be displayed on one page, the list will cover several pages.
Display the previous and following pages by pressing the page keys (PREVIOUS PAGE, NEXT PAGE).



9. Graphics

9.1 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.

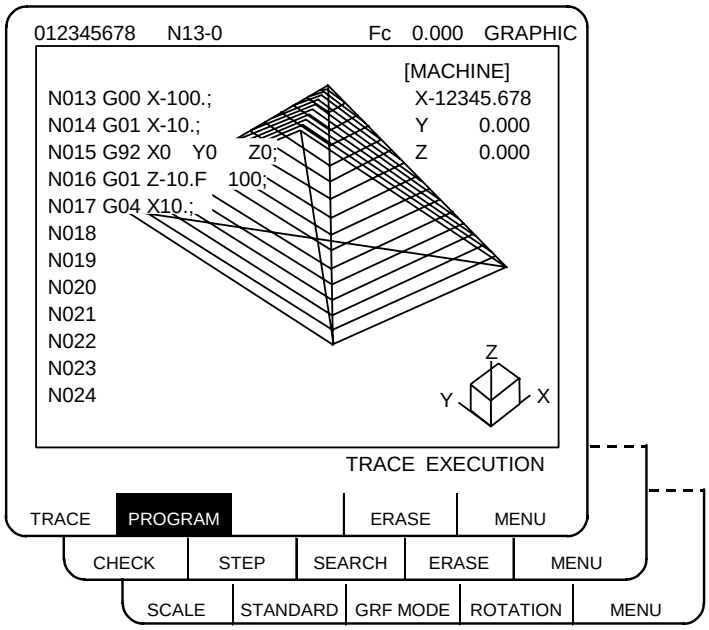
(1) Trace function
The trace function illustrates the actual machine movement path and draws the machine motion itself. The machine operation can be monitored during machining.

(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 option. The GRAPHIC screen will not be displayed if this specification is not added.
(Note 2) If the machining program uses a hypothetical axis, such as in milling interpolation, the figure may not be drawn correctly with the program check function.

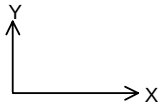
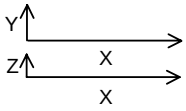
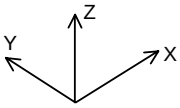
9.2 Menu Function

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 rounding/chamfering 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.

Menu	Function
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)

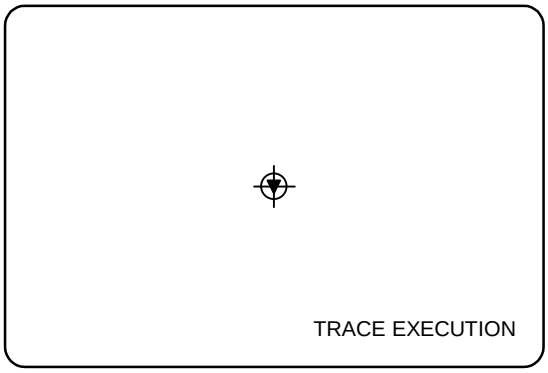
9.3 Use of the Trace Mode (TRACE)

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.

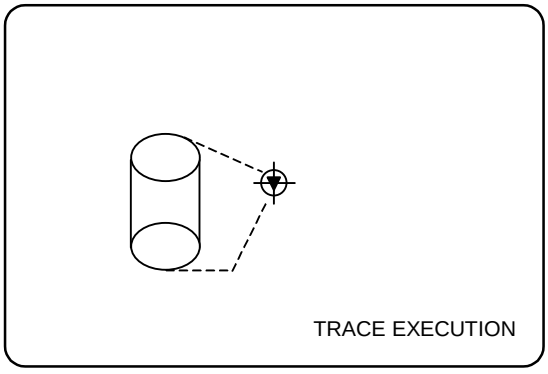
(1) Trace start

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.

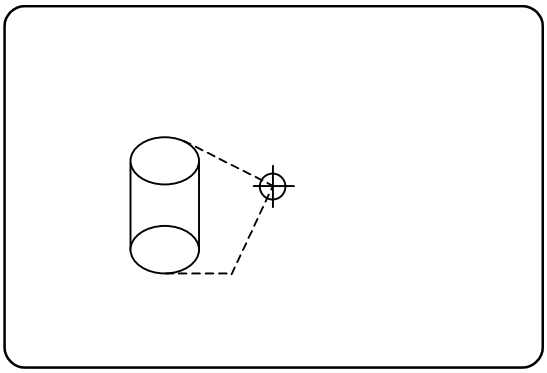


(2) Trace release

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.

9.4 Use of the Check Modes

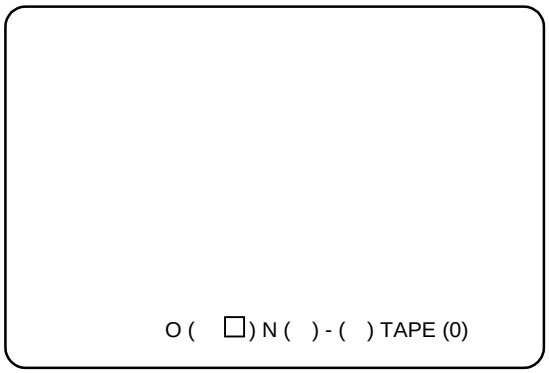
To select a check mode, press menu key [CHECK] or [STEP].
The check mode appears immediately when the check mode is selected.

(1) Preparation for checking

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

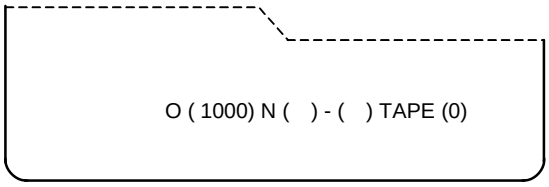
Press the menu key [SEARCH].

The data setting area appears.



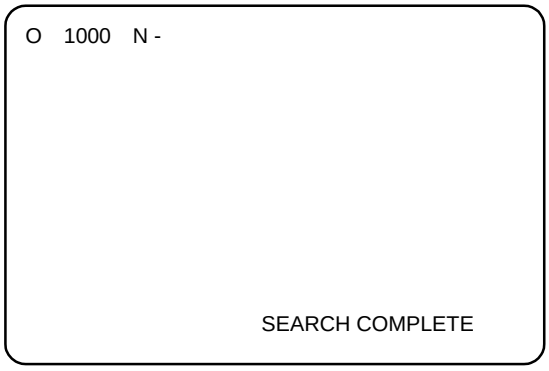
(Example) To check O1000, proceed as follows:

O (1 0 0 0)
N ( ) – ( )
Tape (0)



Press [INPUT] key.

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

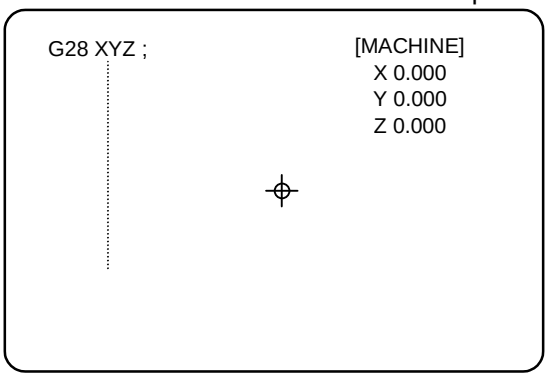


(Note 1) This operation search is exactly the same as that of the MONITOR function. Pressing the automatic start button after the search is completed starts the automatic operation.
The operation mode is designated by the operation mode selection switch on the machine operation panel.

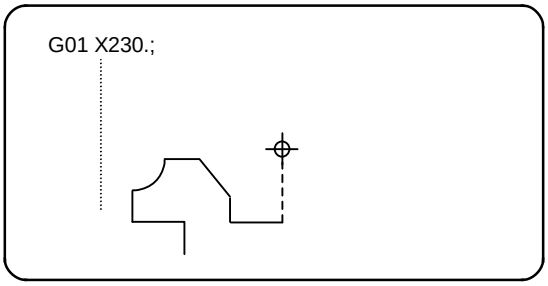
(2) Checking start

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

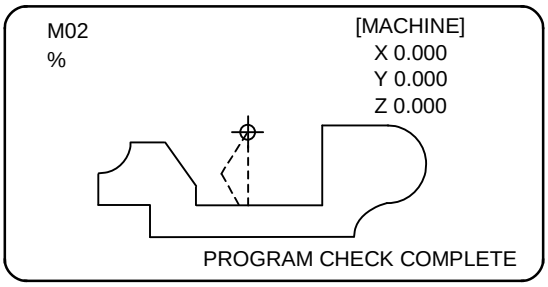
Press the menu key CHECK on the GRAPHIC screen.



The program check is executed.



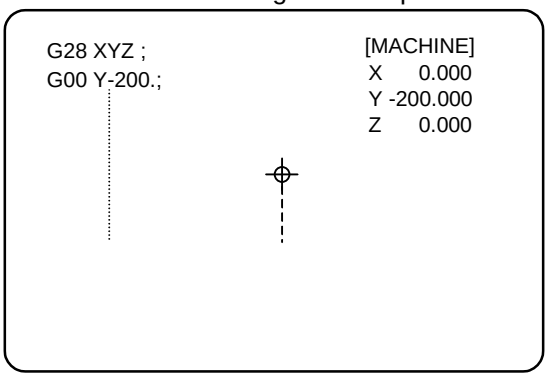
The program check stops at the program end.



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

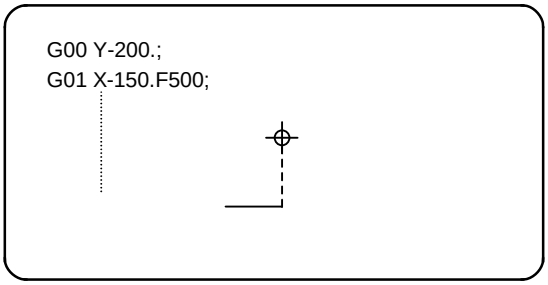
Press the menu key STEP on the GRAPHIC screen.

The system executes one block and stops.



Press the menu key STEP on the GRAPHIC screen.

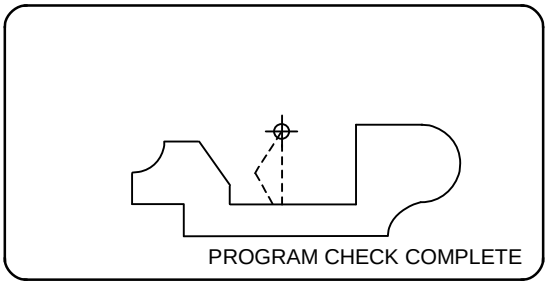
The system executes the next block and stops.



Press the menu key STEP successively.

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

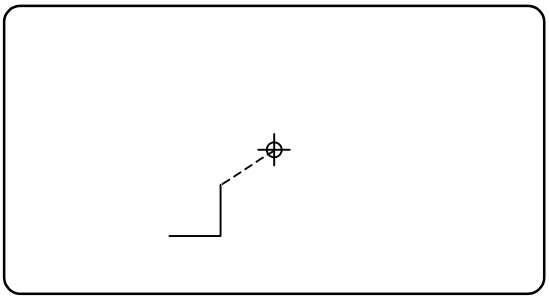
The program check stops at the program end.



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

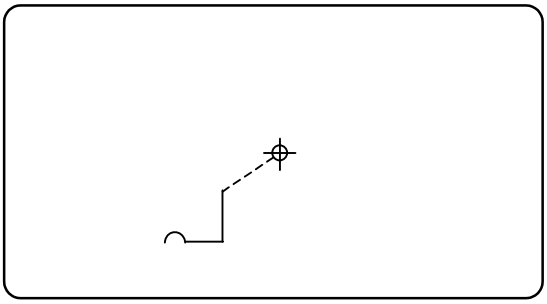
3) Changeover between continuous check and step check modes
Pressing the menu key [STEP] during continuous check execution changes the mode to the step-check mode, in which the system checks subsequent blocks of the program one at a time.
Similarly, the mode can be also changed from the step-check mode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

Execute the continuous check.



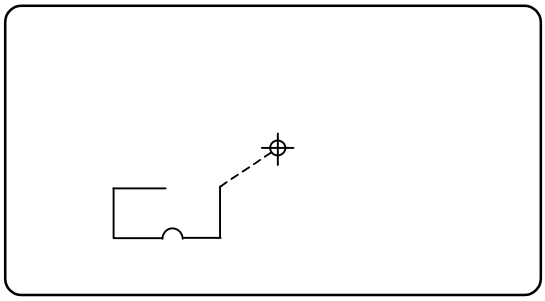
Press the menu key [STEP].

The system executes next one block and stops.



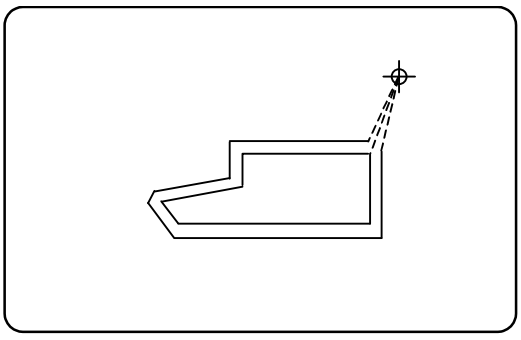
Press the menu key [STEP].

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

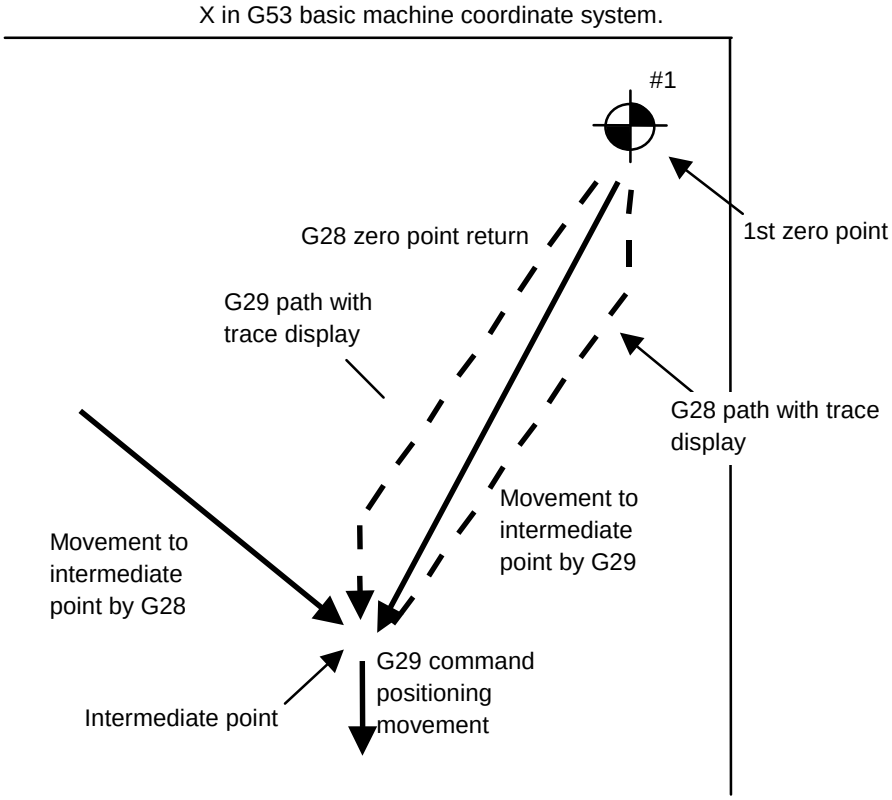
		During tool nose radius compensation	Not during tool nose radius compensation
Manual feed		—	—
Rapid traverse	Program path	Broken line	—
	Tool center path	Broken line	Broken line
Cutting feed	Program path	Solid line	—
	Tool center path	Solid line	Solid line

(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.

Function name	Graphic check	Remarks
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	×	
Zero point return/start position return command	×	Refer to (Note 1).
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 • Message display and stop • 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)** When an NC alarm occurs, the alarm state will be entered without displaying a message for the operator.

(Note 1) Zero 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 zero 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 zero 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, "M01 OPERATION ERROR 0112" will occur while the button is held down, and automatic start will not be possible. Press the automatic start button after the message "PROGRAM CHECK END" has been displayed or the operation has been canceled by 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.
(10) If the mode is changed to the manual operation during the checking, "M01 OPERATION ERROR 0112" will occur and the axis movement will be disabled. End the checking with reset before moving the axis.
(11) If the PLC interruption is commanded during the checking, "M01 OPERATION ERROR 0112" will occur. End the checking with reset before executing the PLC interruption.
(12) If the tap retract is attempted during the checking, "M01 OPERATION ERROR 0112" will occur. Cancel the checking with reset before executing the tap retract.
(13) Neither the counter set nor the origin set can be executed during the checking. If an axis address key is pressed on the POSITION screen, "E69 PROG. CHECK MODE" will occur.
(14) Before returning to the top to restart the checking, cancel the program check in execution with reset. Do not restart the continuous check during the execution (before the "PROGRAM CHECK END" is displayed).

9.5 GRF MODE (GRF MODE)

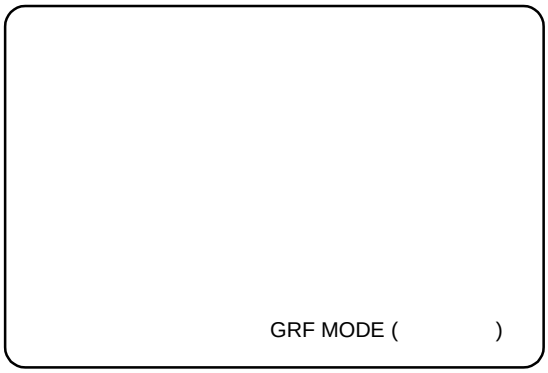
(1) GRF mode types

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.



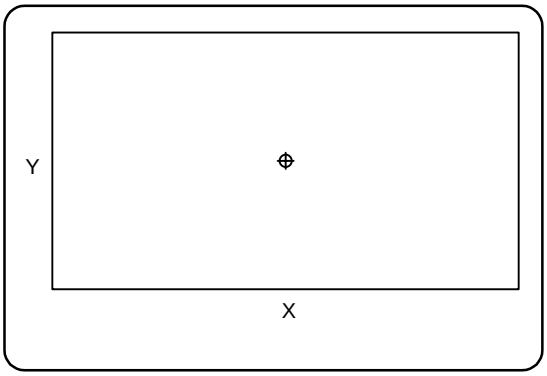
(Example 1) Setting the 1-plane display mode

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)

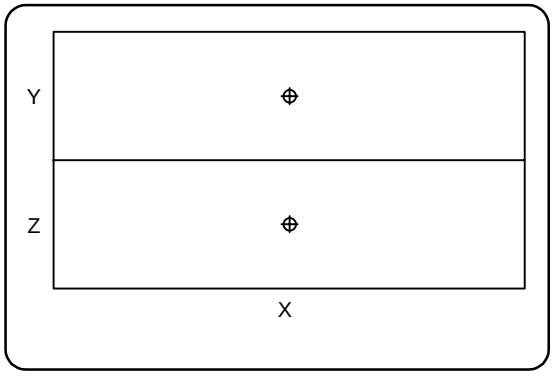
Set the horizontal and vertical axis names of the upper plane and those of the lower plane in order, and press the INPUT key.

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. ☞

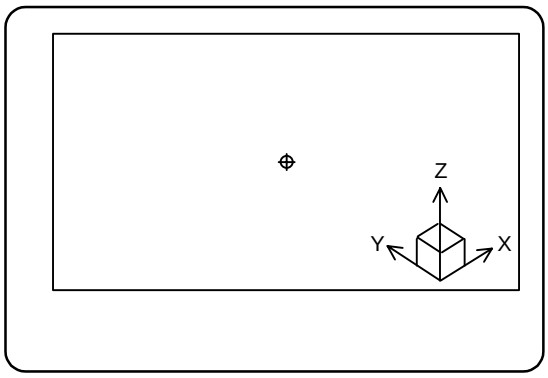
1) When the GRF mode changes over, all already drawn graphics are erased, and the new coordinate axes, etc., appear.



(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.

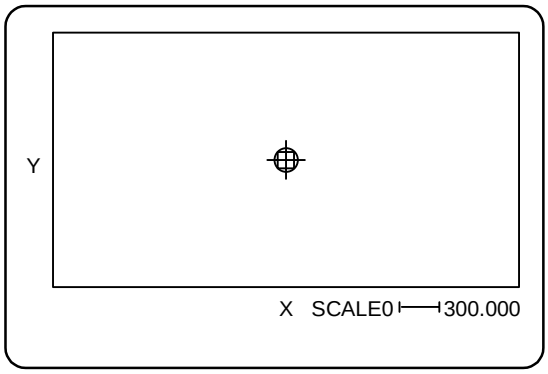
9.6 SCALE (SCALE)

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

9.6.1 Changing the Scale

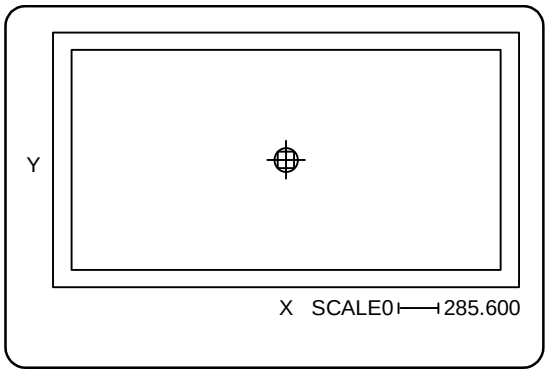
Press the menu key SCALE.

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.



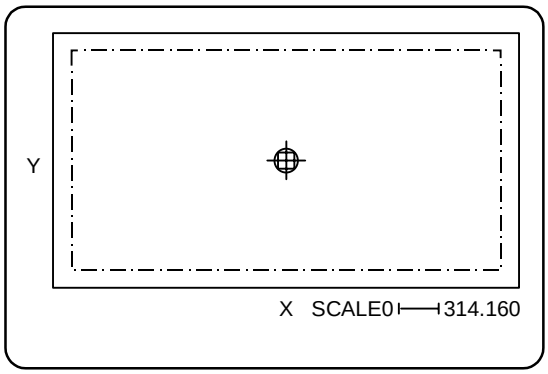
To enlarge the graphics, press the − key to decrement the scale value.

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.



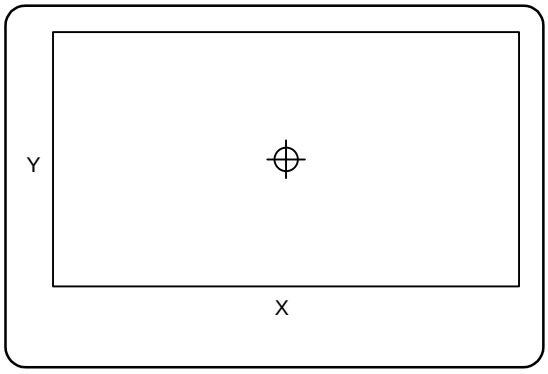
To reduce the graphics, press the + key to increment the scale value.

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.



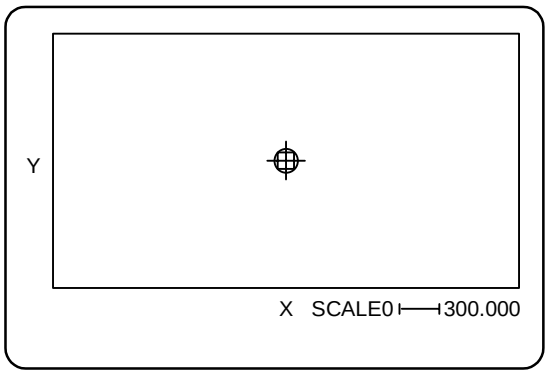
(Note 1) The scale value is incremented/decremented by approx. 5% each time the [+] or [−] key is pressed.
(Note 2) The scale value can be changed in the range of 0.100 to 9999.999.

9.6.2 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 [C.B CAN] key.

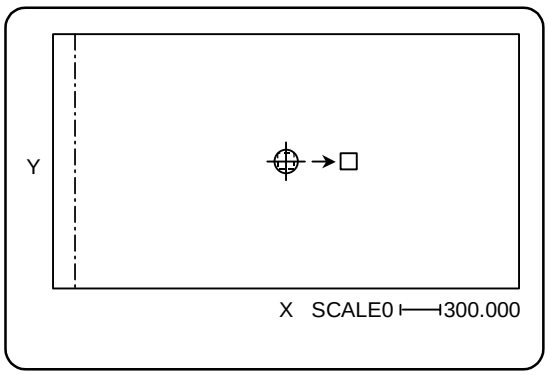
(1) Designating the display range center

Press the menu key [SCALE].

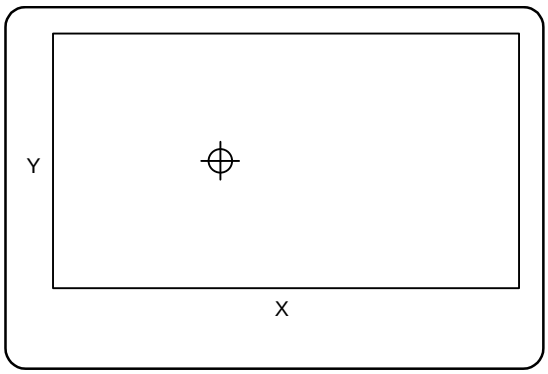


1) The cursor indicating the display range center appears.

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



Press the [INPUT] key.

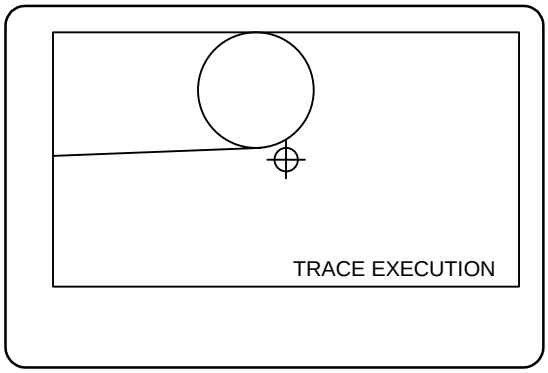


1) The coordinate system changes to the new coordinate system (having the cursor position as the screen center).
2) All already drawn graphics (if present) are erased, and the screen returns to the initial screen.

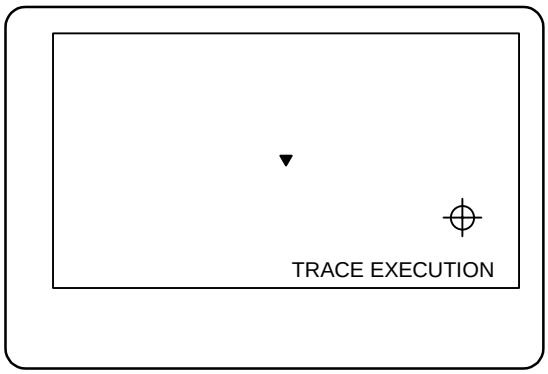
(Note 1) The amount the center moves for one cursor shift varies depending on the current scale value.

(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.



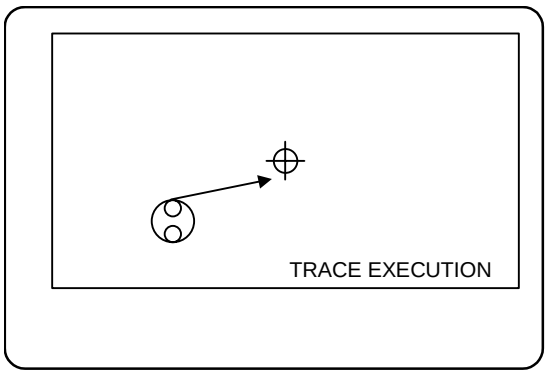
Press the keys [SHIFT] and [C.B CAN].



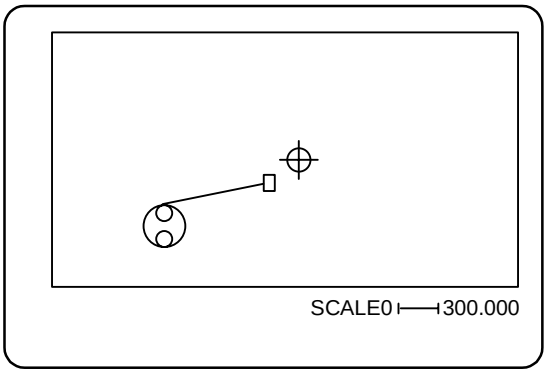
1) The tool mark appears in the screen center.
2) If the message "TRACE EXECUTION" appears, consecutive drawing is made.
   (Note) [SHIFT] + [C.B CAN] function
   This function displays the current machine position in the screen center.

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.

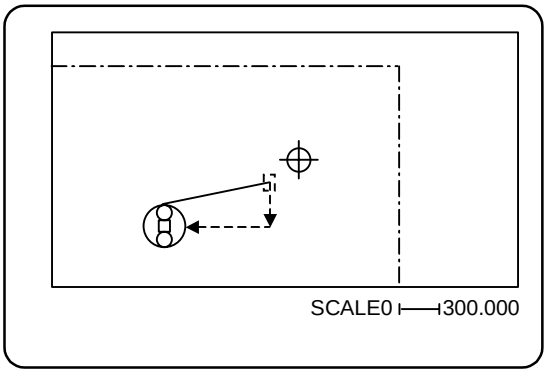
(Example) Changing the scale and display position simultaneously
To enlarge the graphics (shown at the right) in the screen center, carry out the following operations:



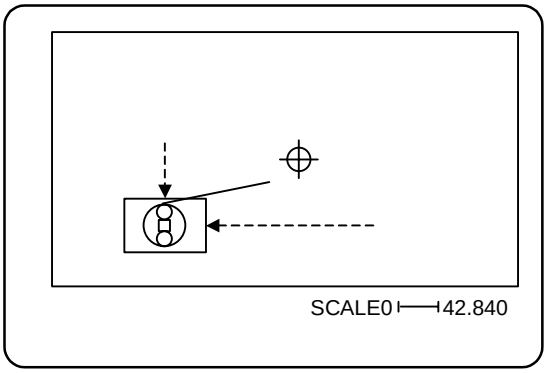
Press the menu key [SCALE].



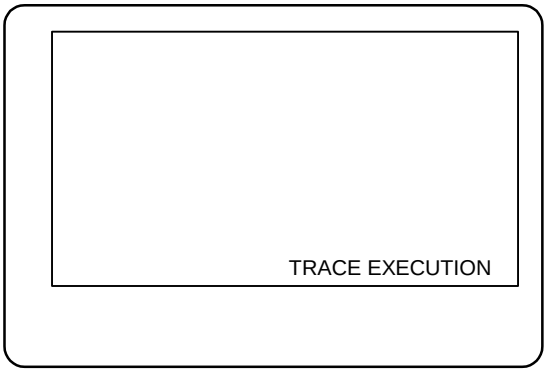
Using the cursor shift keys, move the cursor near the graphics center.



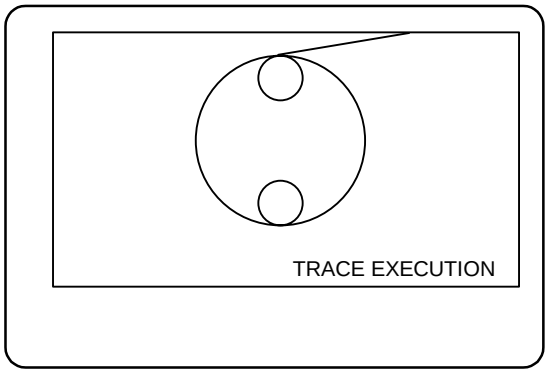
Using the [−] key, fully enclose the graphics within the frame.



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

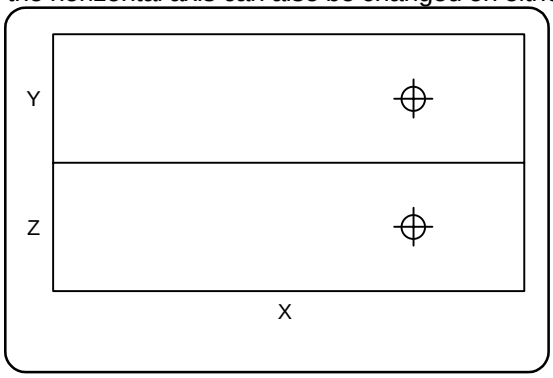


Draw the graphics again with the same program.



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

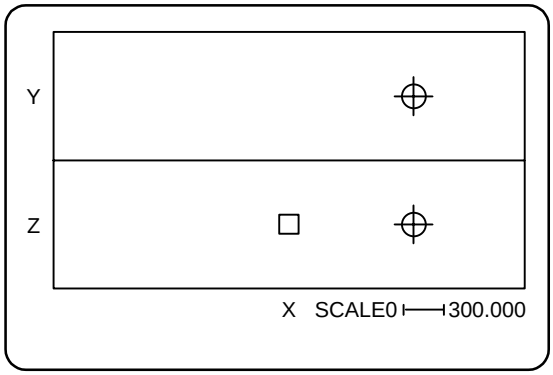
1-plane and 3-dimensional display modes are as explained before, but in the 2-plane display mode the display positions of the upper and lower planes can be changed separately.
Although the scale of either plane can be changed in the 2-plane display mode, the same scale is always applied to the upper and lower planes. Similarly, the horizontal axis can also be changed on either plane.



Changeover between upper and lower planes

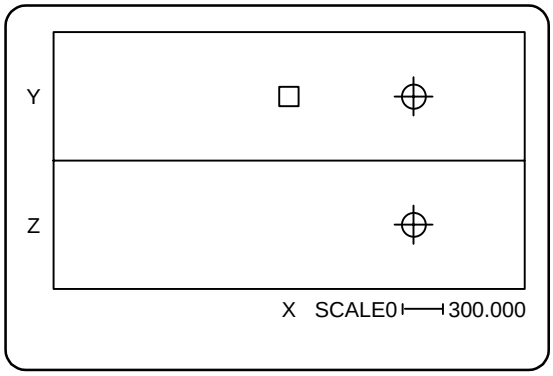
Press the menu key SCALE.

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.

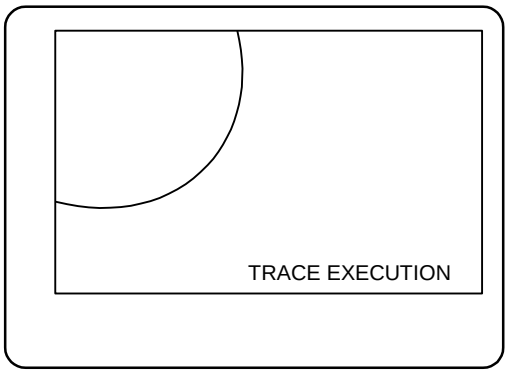
1) The cursor moves to the upper plane. The upper plane display range can now be changed.



9.7 STANDARD (STANDARD)

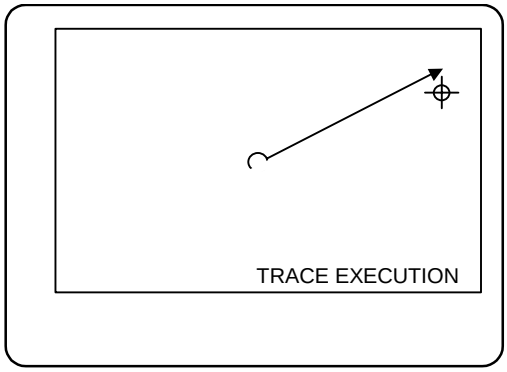
When the menu key STANDARD is pressed, the machine's moveable range determined in setup parameters OT+ and OT– (stored stroke limit range) becomes the display range. The scale and display position are automatically changed. This function is useful if the drawing graphics abnormally shift out of the screen.

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



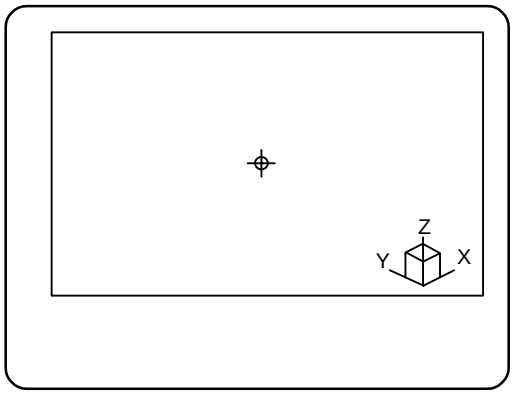
Press the menu key STANDARD.

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.

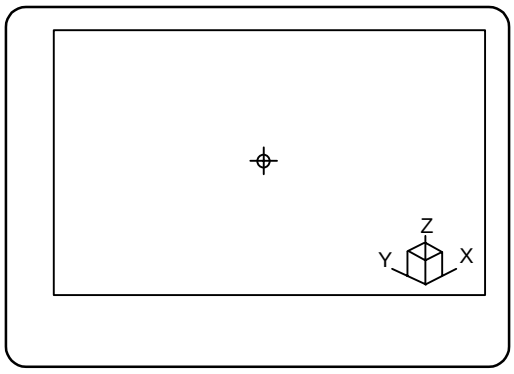


9.8 ROTATION (ROTATION) (M system)

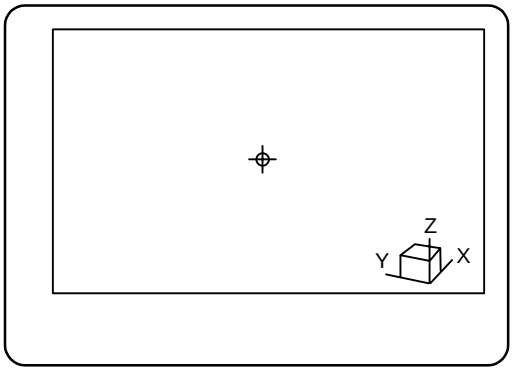
A 3-dimensional display can be rotated in any direction.



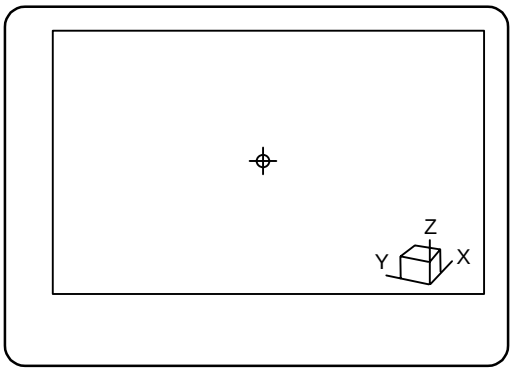
Press the menu key ROTATION.



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



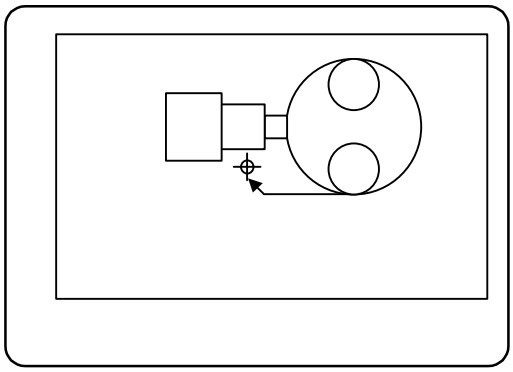
Press the INPUT key, or press the ROTATION key again.



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

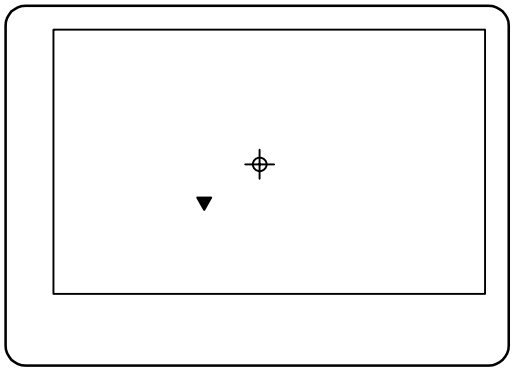
9.9 ERASE (ERASE)

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.

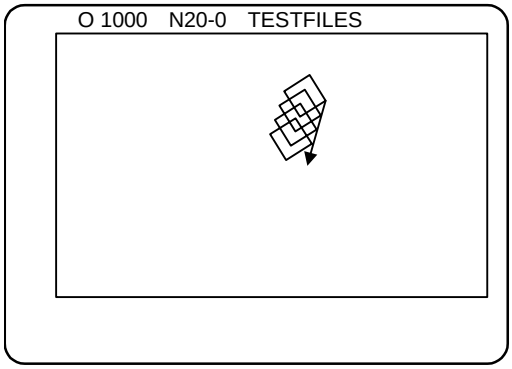
1) All already drawn graphics are deleted.



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

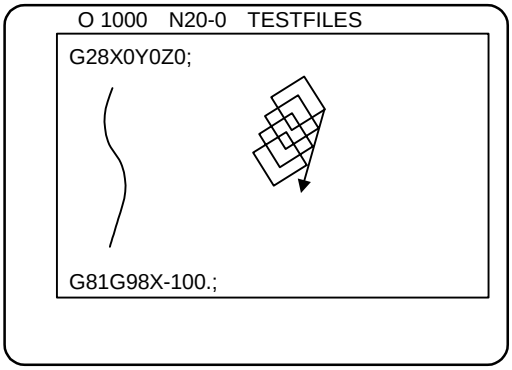
9.10 PROGRAM (PROGRAM)

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.



10. Ladder Circuit Monitor [for PLC built-in specification only]

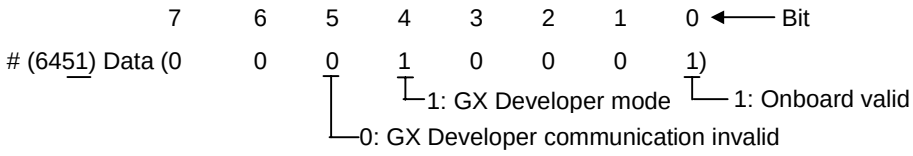
By pressing the function selection key [F0], the menu key [LADDER] appears and the sequence circuit operation status can be confirmed.
The following monitor functions are available:
(1) Circuit monitor
(2) Screen stop using a monitor stop trigger point
(3) Registration monitor
(4) Current value monitoring changeover between decimal notation and hexadecimal notation

10.1 Parameter Setting

Parameter setting for the ladder circuit monitor function is carried on the PLC parameter BIT SELECT screen.

The ladder circuit monitor can be selected by turning the "Onboard valid" parameter ON.
When GX-Developer communication is valid, the on-board screen will not appear and the ladder circuit monitor cannot be used.

[PLC bit selection]

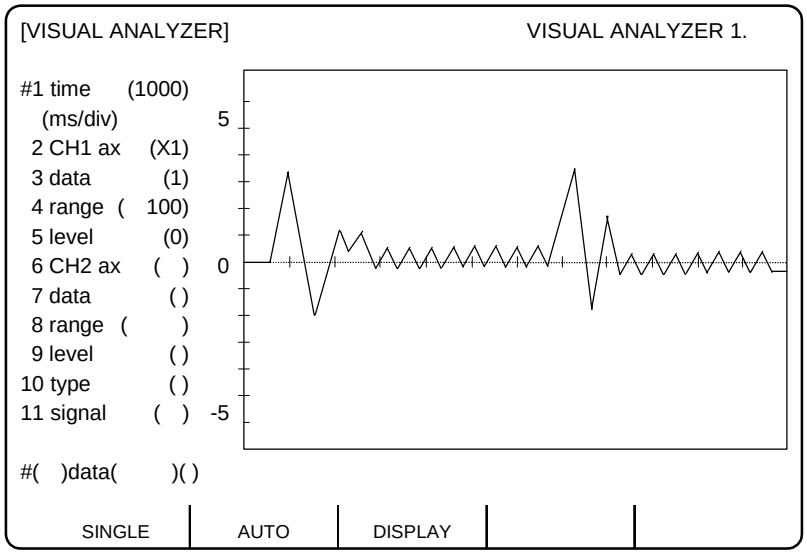


This function is used for user PLC development.
Refer to the PLC Onboard Instruction Manual (BNP-B2213) for details.

11. Visual Analyzer (Waveform display)

Pressing the function selection key [F0] displays the VISUAL ANALYZER screen. On this screen, changes in the spindle and servo operation status can be displayed in time units simultaneously for both channels.
During synchronous tap, the difference of the spindle and tap axis position deviation (synchronous error amount) can be displayed as a waveform.
The waveform data can be output. Refer to section "6.2.6 Outputting Waveform Data" for details.

(Note) The VISUAL ANALYZER screen is valid when the setup parameter "#1222 aux06/bit2" is set to 1.



#	Parameter	Description	Setting range (units)
1	time (ms/div)	Specify the unit of a time axis (horizontal axis) gradation in milli-second (ms).	1 to 9999
2	CH1 ax	Specify the name of the axis to be displayed for CH1. No waveform data is displayed if ( ) is left blank. Input "/" to erase the displayed waveform.	Servo axis: Axis name [2 characters] (X1, Y1, Z1, etc.) Spindle: S1 to S4 (Just S is sufficient for the first spindle.) "/" will cancel the axis name.

#	Parameter	Description	Setting range (units)
3	data	Specify data to be sampled. (Servo axis) (Spindle) 1: Current FB (%) 1: Motor load (%) 2: Current command (%) 2: Motor rotation speed (speed command value) (r/min) 3: Position deviation (μm) 3: Position deviation (μm) 4: Position command (μm) 4: Position command (μm) 5: Speed FB (r/min) 5: Srpm (speed FB) (r/min) 6: Synchronous error width (μm) 7: Synchronous error angle (0.001◦) The position command and deviation and synchronous tap error width are displayed with an mm unit even when inch specifications are set.	1 to 7
4	range	Specify the size of the vertical axis. That is, specify the size of six gradations equivalent to the half of the screen. (Example 1) In case of current FB When #4= 400, the upper half of the screen is 400% and the lower half of the screen is –400%. (Example 2) In case of position deviation When #4= 18000, 18,000μm (18mm) (Example 3) In case of synchronous error width (μm) When #4 = 6000, the scale increment for the vertical axis will be 1mm (1000μm). (Example 4) In case of synchronous error angle (0.001◦) When #4 = 6000, the scale increment for the vertical axis will be 1◦.	1 to 99999
5	level	Specify the position of level 0 on the vertical axis. That is, specify the position of the current and speed 0 level on the scale. In the case of a position command or position deviation, the display start position is displayed at the specified gradation.	–6 to 6
6	CH2 ax	Specify the name of the axis to be displayed for CH2. No waveform data is displayed if ( ) is left blank. Input "/" to erase the displayed waveform.	Refer to the setting range for "#2 CH1 axis".
7	data	Refer to "#3 data". (#7 is used for CH2.)	1 to 7
8	range	Refer to "#4 range". (#8 is used for CH2.)	1 to 99999
9	level	Refer to "#5 level". (#9 is used for CH2.)	–6 to 6
10	type	Specify the condition to stop waveform display in continuous mode. 0: Continuous menu OFF 2: #11 signal ON 3: #11 signal OFF Display will stop when the continuous menu is turned OFF regardless of the stopping conditions.	0, 2, 3
11	signal	Specify the stop signal name in hexadecimal number preceded by X or Y.	X00 to XFF Y00 to YFF

(Note 1) Only the servo axis name set in "#1022 axname2" can be set in #2 and #6.
(Note 2) Only bus connected spindle (#3024 sout=1) can be set in #2 and #6.

11.1 Menu Function

Menu	Function
SINGLE	When menu key SINGLE is pressed, data is collected and displayed for "#1 setting value ∗ 12 scale [ms]".
AUTO	When menu key AUTO is pressed, data is continuously collected and displayed.
DISPLAY	The waveform data last page is displayed.

(Note 1) For continuous waveform display, the waveform display stop condition can be selected using #10.

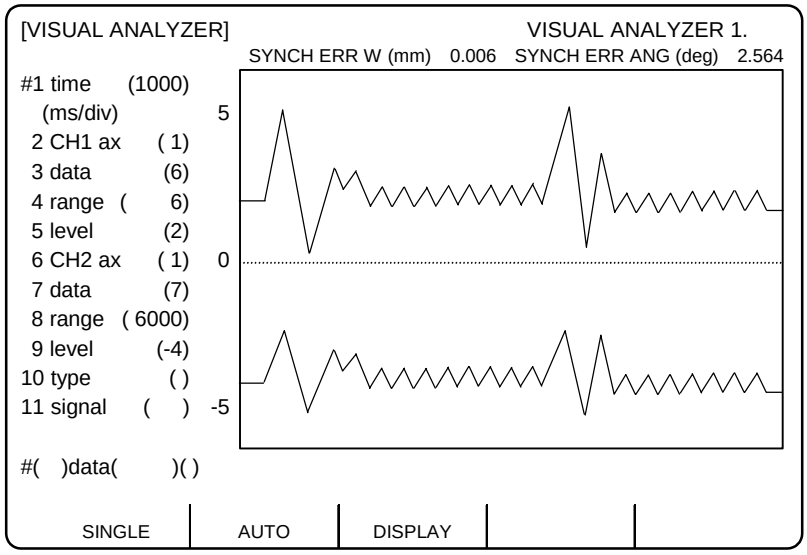
(Note 2) To refer to data exceeding one screen with the continuous waveform display, press the menu key AUTO to stop the waveform display, and then press the page key.

(Note 3) The parameter settings cannot be changed during continuous waveform display. Stop waveform display and change them.

(Note 4) The data for up to 4226 points can be retrieved for the waveform data. If data exceeding 4226 points is retrieved, the data will be deleted from the oldest data.

11.2 Synchronous Tap Error Display

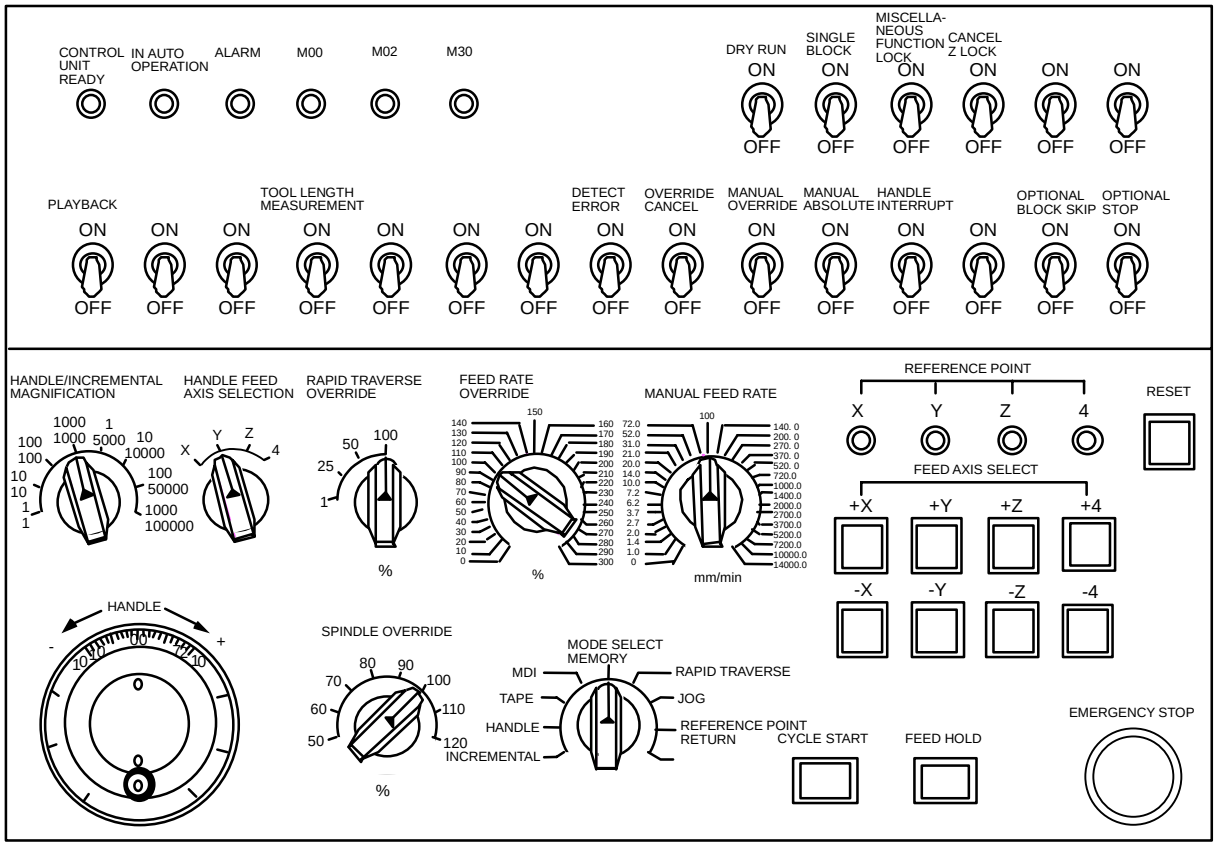
To display the error amount during synchronous tap, set either "6" or "7" in "#3 data" or "#7 data" and display the waveform.
The operation methods and other setting items are the same as the normal waveform display.



(Note 1) With the synchronous tap error display, only the waveform during cutting feed is displayed.
(Note 2) Data displayed in "SYNCH ERR W (mm)" and "SYNCH ERR ANG (deg)" on the screen indicates the maximum value of the synchronous tap error width obtained from the data displayed as a waveform.
(Note 3) The "SYNCH ERR W (mm)" is indicated as a value that allows for the gear ratio.
(Note 4) The "SYNCH ERR W (mm)" and "SYNCH ERR ANG (deg)" are displayed on the screen only when the spindle No. is set in "#2 CH1 axis" (or #6 CH2 axis), and 6 (synchronous tap error width) or 7 (synchronous tap error angle) is set in "#3 data" (or #7 data).

II. MACHINE OPERATION MANUAL

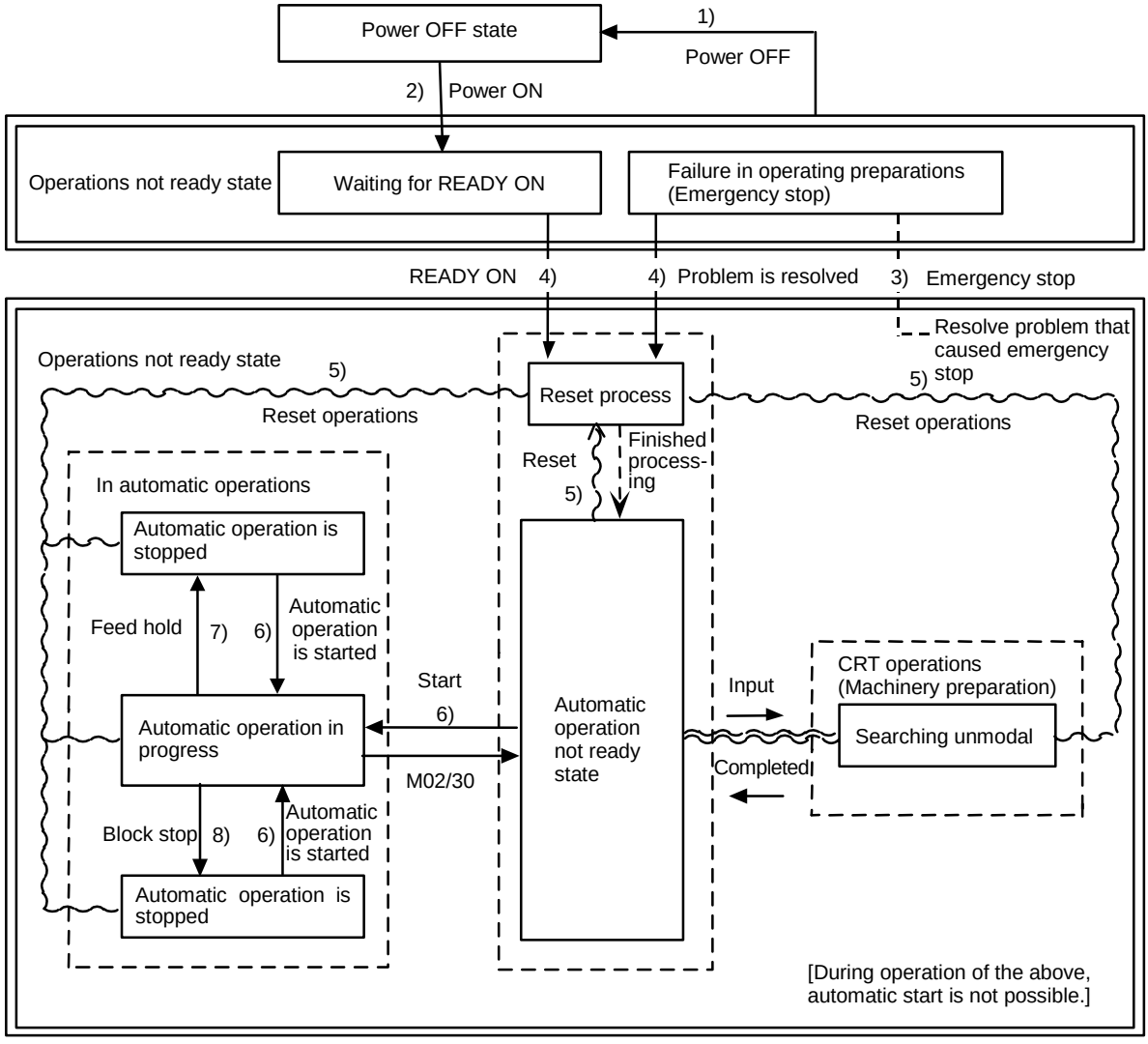
This chapter explains the functions and operation method of the machine operation switches for operation (automatic operation and manual operation) by using the illustration of the machine operation panel.
The actual machine operation and motion vary from one minute to another. Refer to the operation manual issued by the machine manufacturer. Use this chapter for reference.



1. Operation State

1.1 Operation State Transition Diagram

The controller 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.
Operation is enabled only in the operating preparation ready state. The operating preparation ready state is furthermore classified into reset, automatic operation stop, automatic operation start, and automatic operation pause as shown below:
Manual mode operation is enabled in the operation complete state.



Operation state transition diagram

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 POWER OFF signal is input from the machine;
- When power supplied from the machine to controller is turned OFF; or
- When power unit fuse in the controller blows.

1.3 Run Not Ready

The operation preparation not ready state means that the system is not ready to run because of the controller or because of the machine, even though power is supplied to the NC unit is control circuit. The RUN READY lamp on the setting and display unit is OFF.

(1) From power OFF to not ready (transition 2))
- 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 and display unit; or
- When any of the following alarms is displayed on the setting and display unit;
  Servo alarm, feedback alarm, excessive error alarm, watch dog, MCP 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 and display unit is ON.
The state is furthermore classified into the following four:

1.4.1 Reset

The reset state means that the controller is reset.

(1) From not ready to reset (transition 2))
This state is also called initial state.
- When about four seconds have passed after the power is turned ON.

(2) From another ready state to reset (transition 5))
- When the RESET key on the setting and display unit is turned ON;
- When external reset signal is input from the machine; or
- 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 IN-AUTO OPERATION lamp on the machine operation panel is ON.

(1) From another ready state 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 pause (transition 7))
- When the FEED HOLD switch on the machine operation panel is turned ON; or
- 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 complete; or
When automatic mode input is changed to another automatic mode input.

2. Indicator Lamps

2.1 Control Unit Ready

The CONTROL UNIT READY lamp indicates that the control unit is ready to run. It goes ON in about one second after the power is turned ON. The lamp goes OFF at emergency stop or when an alarm occurs in the drive or operation block.

2.2 Automatic Operation Busy

The AUTO BUSY lamp is ON from CYCLE START switch turning ON in the automatic operation mode (memory, tape, or MDI) to the program end after M02 or M30 execution, reset, or emergency stop.

2.3 Automatic Operation Start Busy

The AUTO START BUSY lamp indicates that the controller 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 (tape, 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 AUTO PAUSE switch turning on to AUTO START switch turning ON or when the mode selection switch is changed from the automatic to manual mode during the automatic operation.

2.5 Return to Reference Position

Output is executed when the controlled axis arrives at the reference position when manual return to reference position, automatic return to reference position, or reference position collation (check).

2.6 Alarm

The ALARM lamp goes ON when an alarm occurs during NC running.

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 controller is reset by turning ON the RESET switch on the machine operation panel or the RESET key on the setting and 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 move command is being executed, move 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 ALARM lamp goes OFF.
(5) If the reset switch is turned ON while the input/output device is running, the power will be cut off.
(6) When the reset switch is turned ON, the modal state will return to its original state.

3.2 Emergency Stop Button

The EMERGENCY STOP button is a red mushroom-shape pushbutton. The unready state is set by pressing the EMERGENCY STOP button. During emergency stop, the READY lamp goes OFF and automatic operation and manual operation do not work. The controller is reset.
If the EMERGENCY STOP button is pressed when a move command is executed, the moving axis stops and all other machine motions also stop.
When the EMERGENCY STOP button is released, the READY lamp goes 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 pressed, 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

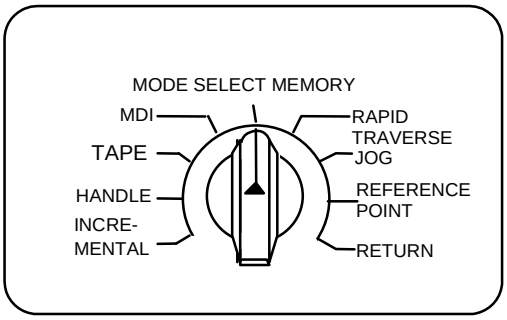
❗ If the axis overruns or emits an abnormal noise, immediately press the emergency stop button and stop the axis.

4. Operation Mode

(Note) Do not execute the tap retract simultaneously with the operation in the manual (JOG feed, rapid traverse feed, reference position return, incremental and handle) mode or in the automatic (memory, tape and MDI) mode.
The simultaneous operation causes the all axes interlock.

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.
Tape mode:
Select the mode for tape 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 maker 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.

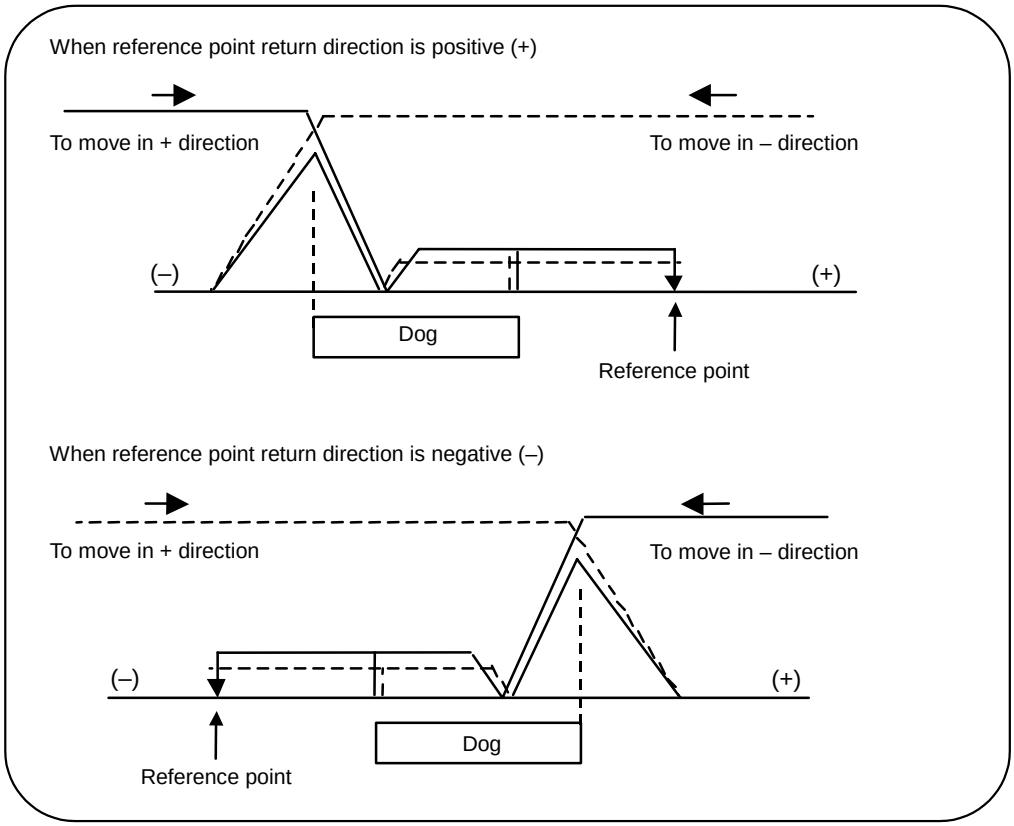
Using the RAPID TRAVERSE OVERRIDE switch, set any desired override value.

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

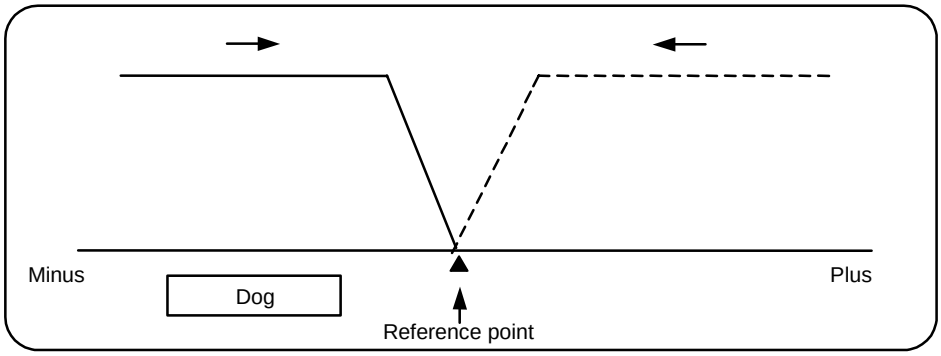
The steps below describe what happens to the controlled axis when it returns to the reference point for the first time with the power ON and with the machine in an "operations not ready state" (emergency stop is engaged or the servo alarm is ON) or when the parameters are selected in the dog mode.

(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.

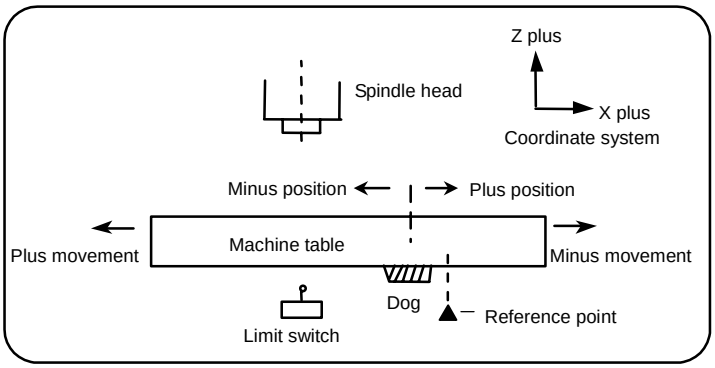


Operation procedure

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

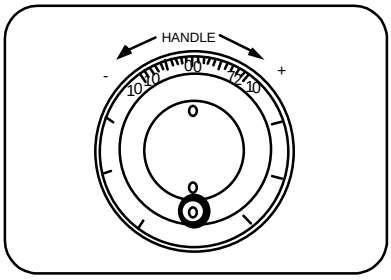
Using the MODE SELECT switch, select the handle feed mode.

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.

☞



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.

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 run 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 setting and display unit MDI screen.

Operation procedure

The MDI operation follows the memory operation.

Set data on the 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. Refer to 4.7.

(Note 1) When using the 2-part system, the MDI data operation can be switched with the parameters.

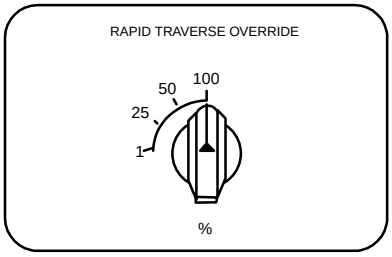
#1050 MemPrg	Details
0, 2, 4, 6	The common MDI program for the system is run in each system.
1, 3, 5, 7	MDI programs for each system are run in each system.

5. Operation Panel Switches in Operation Mode

5.1 Rapid Traverse Override

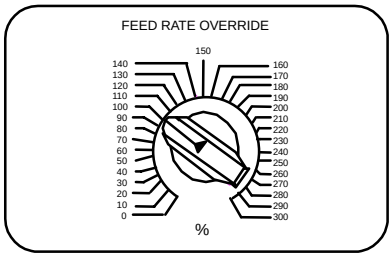
Use the RAPID TRAVERSE OVERRIDE switch to override the rapid traverse feedrate in automatic or manual operation.
RAPID TRAVERSE OVERRIDE is applicable to the following:

Automatic operation: G00, G27, G28, G29, G30
Manual operation: Rapid traverse, return to reference position, incremental feed



5.2 Cutting Feed Override

Use the FEED RATE OVERRIDE switch to override the feedrate in automatic operation (G01, G02, or G03 F command) or the manual feedrate of jog feed in manual operation in 10% units in the range of 0% to 300%. FEED RATE OVERRIDE is also applicable to the dry run rate in automatic operation.

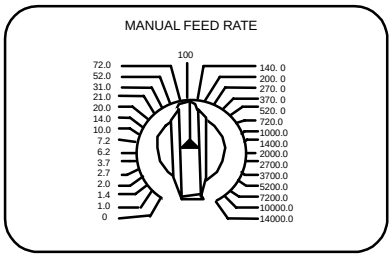


(Note 1) The dry run rate is the movement rate set by using the MANUAL FEED RATE switch by overriding the programmed feedrate in automatic operation.
(Note 2) See Section 6.5 for feedrate override applied to manual feedrate.

5.3 Manual Feedrate

Use the MANUAL FEED RATE 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 FEED RATE OVERRIDE switch takes precedence over the value set by using the MANUAL FEED RATE switch.

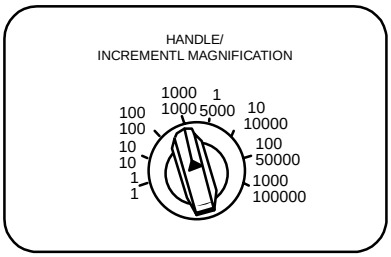
Manual feedrate (mm/min)				
0.	7.2	72	720	7200
1.0	10.0	100	1000	10000
1.4	14.0	140	1400	14000
2.0	20.0	200	2000	
2.7	27.0	270	2700	
3.7	37.0	370	3700	
5.2	52.0	520	5200	



5.4 Handle/Incremental Feed Magnification Factor

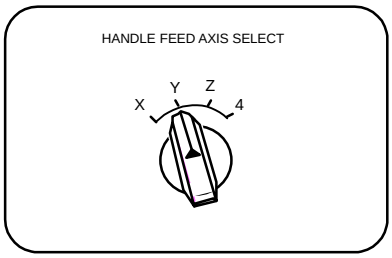
Use the HANDLE/INCREMENTAL MAGNIFICATION switch to set the travel distance specified when manual handle feed or incremental feed is made.
The travel distances for each axis are listed below:
(Up to 1000 can be set for handle feed.)

Handle	Incremental
1	1
10	10
100	100
1000	1000
1	5000
10	10000
100	50000
1000	100000



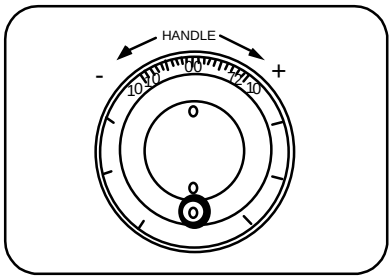
5.5 Handle Feed Axis Selection

Use the HANDLE FEED AXIS SELECT switch to select the axis moved by handle operation when the handle mode is selected.



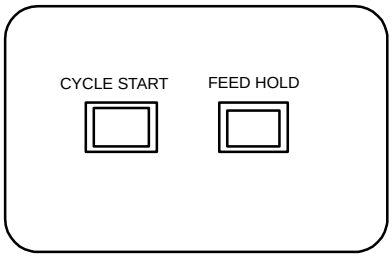
5.6 Manual Pulse Generator

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

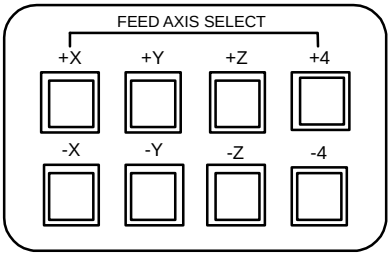
Use the CYCLE START switch to start automatic operation (memory, tape, or MDI). Automatic operation is executed by turning ON the switch. Use also the switch for restart from stop by the FEED HOLD switch or the automatic operation stop state.



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 Chamfering

Chamfering can be validated/invalidated in the thread cutting cycle using an external switch.

6.2 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.

6.3 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.

6.4 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).

6.5 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.

6.6 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.

6.7 Optional Stop

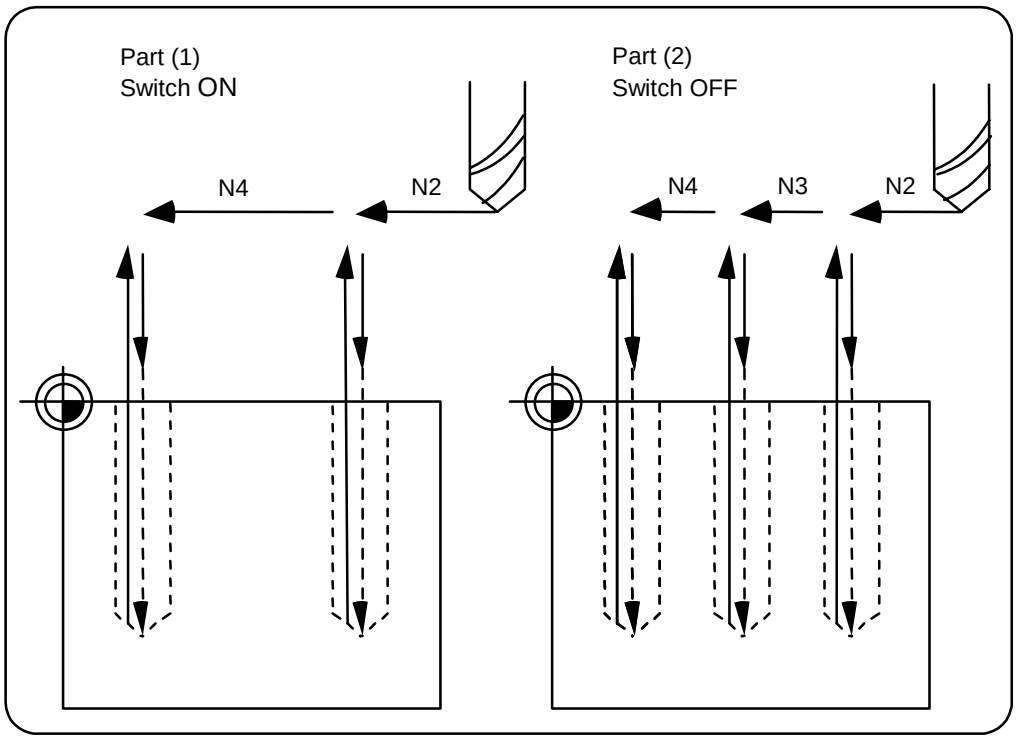
(1) If M01 is programmed, the machine automatically stops by turning ON the OPTIONAL STOP switch. When the switch is OFF, M01 is ignored and the machine does not stop.
(2) The machine stops after the M01 block is executed.

6.8 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.

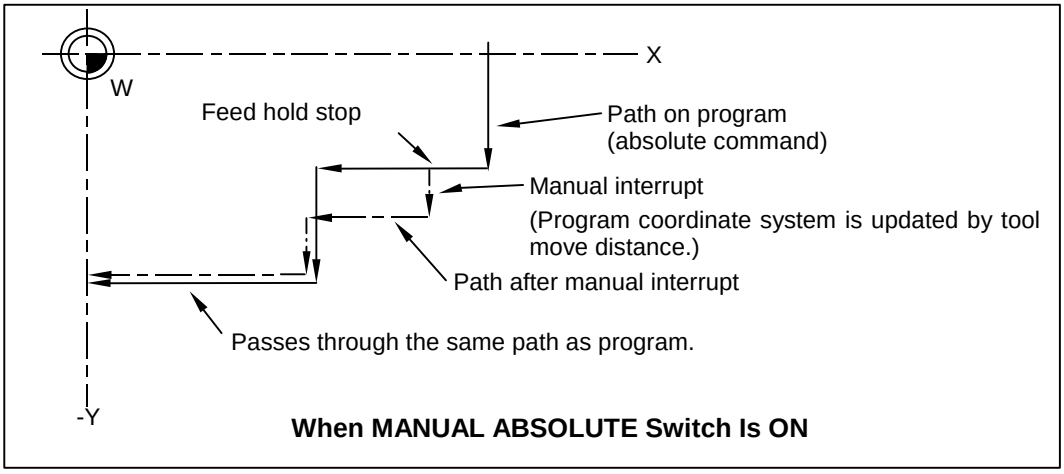
(Example) To work two parts as illustrated below, if the following program is prepared and work is made by turning ON the OPTIONAL BLOCK SKIP switch, part (1) is provided; if work is made by turning OFF the switch, part (2) is provided:

```
Program  N1G54 ;
         N2G90G81X50. Z-20. R3. F100 ;
         /N3X30. ;
         N4X10. ;
         N5G80 ;
         M02 ;
```

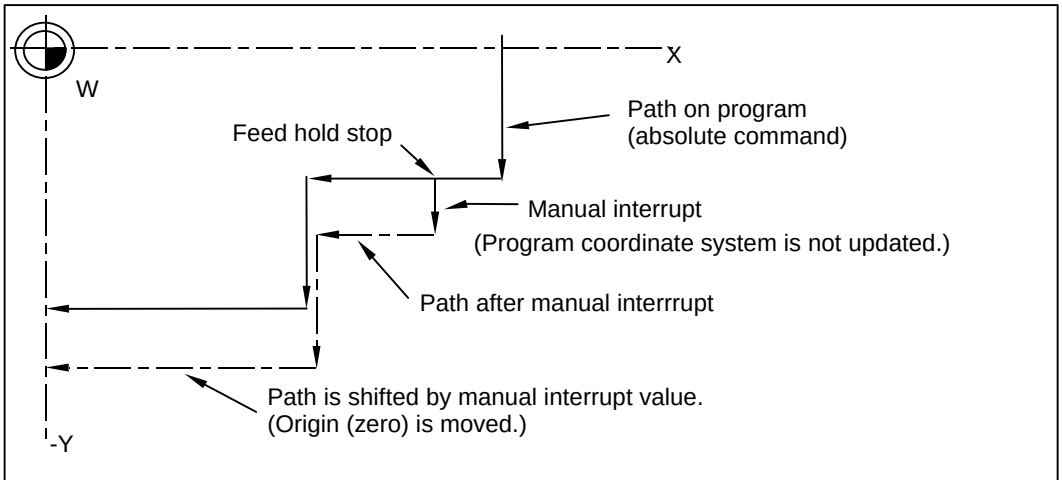


6.9 Manual Absolute

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.



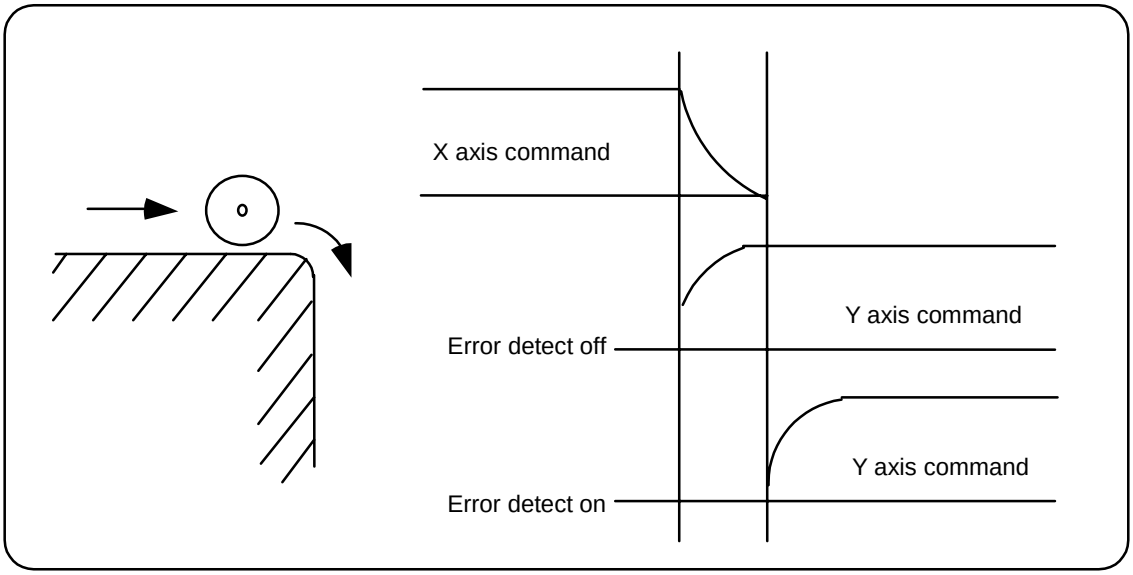
When MANUAL ABSOLUTE Switch Is ON



When MANUAL ABSOLUTE Switch Is OFF

6.10 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.



6.11 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.

6.12 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.

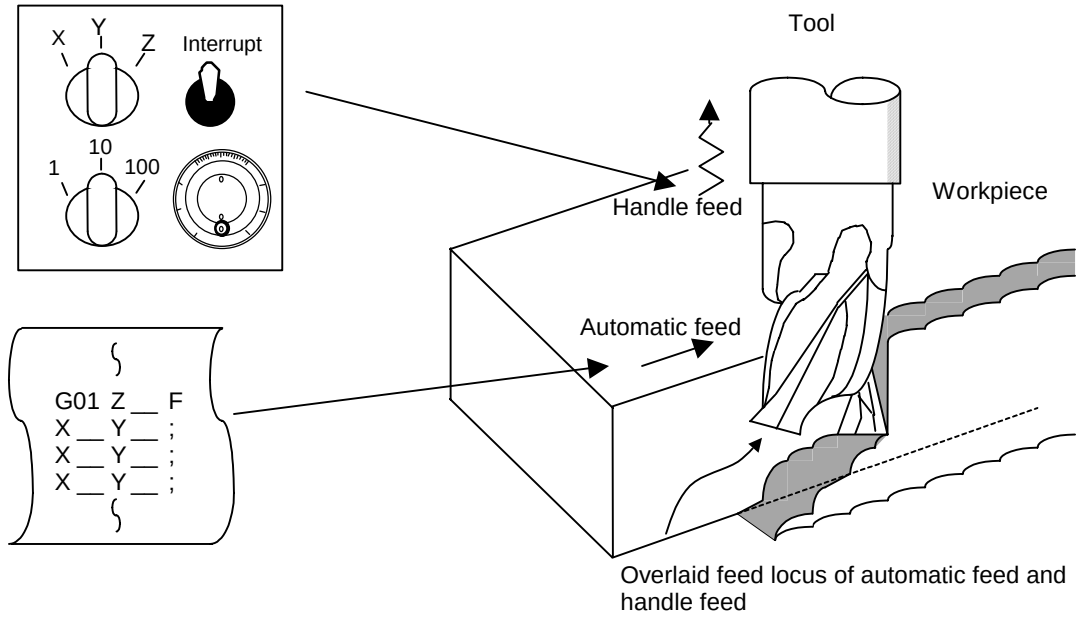
6.13 Manual/Automatic Synchronous Feed

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.

6.14 Handle Interruption

6.14.1 Outline

Section 6.14 explains automatic handle interruption, which enables the operator to interrupt movement using the manual handle in automatic modes (tape, memory, MDI).



6.14.2 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.

6.14.3 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.)

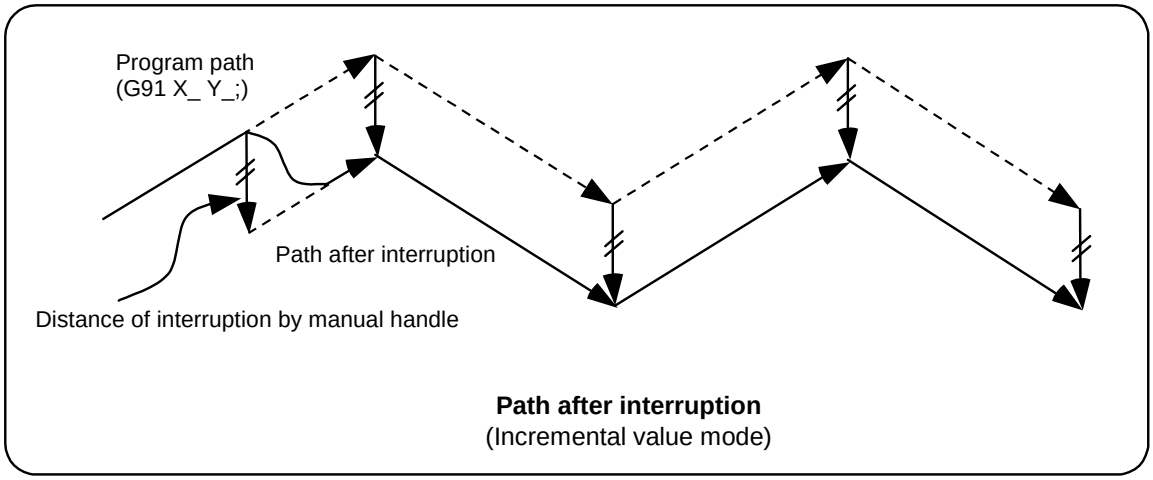
6.14.4 Axis Movement Speed Resulting from Interruption

(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.

6.14.5 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.

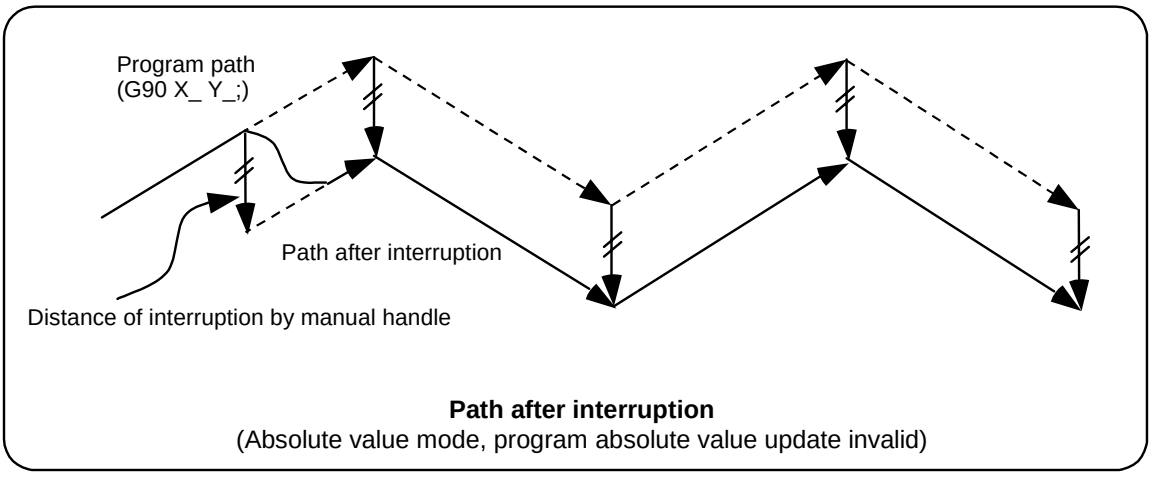
(Note) Each of the POSITION and MACHINE display includes the distance of handle interruption.

For automatic handle interruption, select whether absolute data is updated or not, as follows:

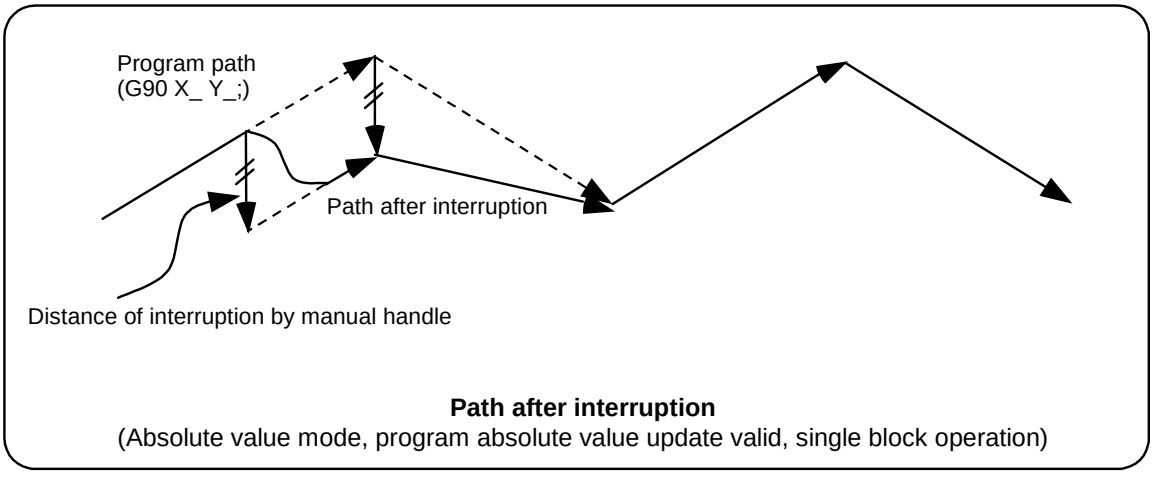
1) Using machine parameters, set whether the manual absolute changeover switch or parameters are used.
2) If the parameters are used, set whether or not absolute data is updated for each axis, by using machine parameters other than those in item.
3) If the manual absolute changeover switch is used, use the machine operation panel switch for selection.

Absolute value update conditions for automatic handle interruption

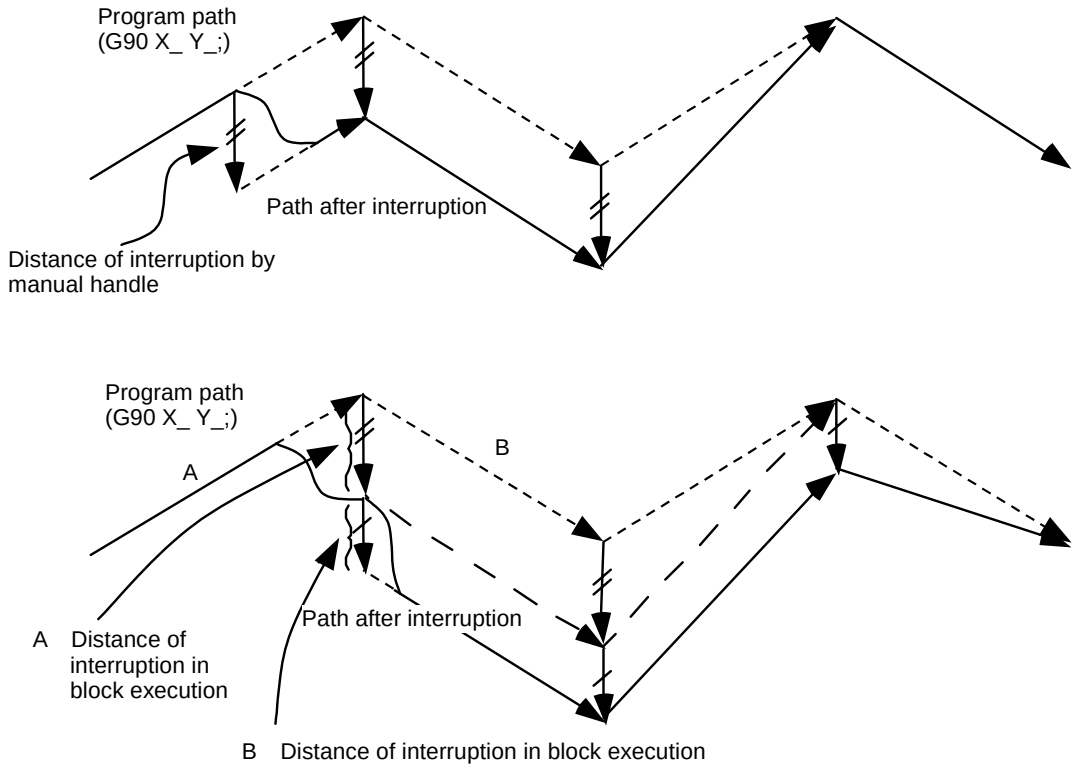
			Absolute data update		MONITOR 2 screen manual interruption distance display
Parameter #1145 I_abs	ON "1"	Parameter #1061 intabs (Every axis)	ON "1"	Absolute value is updated.	Not updated.
			OFF "0"	Absolute value is not updated.	Updated.
	OFF "0"	PLC interface manual absolute switching	ON	Absolute value 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)



Path after interruption
(Absolute value mode, program absolute value update valid, continuous operation)

6.14.6 Handle Interruption in Tool Radius Compensation

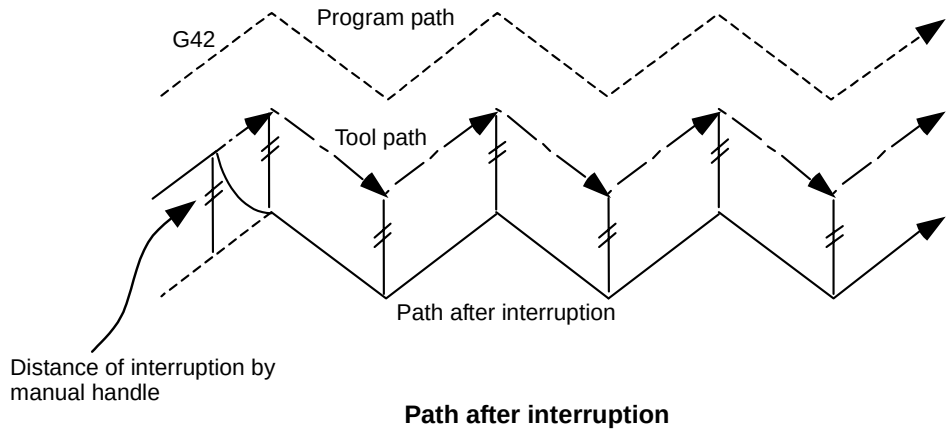
Special movement described below relates only to the tool radius compensation plane axis. It has no influence on the other axes.
At time of tool radius compensation (G41, G42):
In incremental value mode (G91) — The quantity of deviation equals the distance of interruption.
In absolute value mode (G90) — 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.

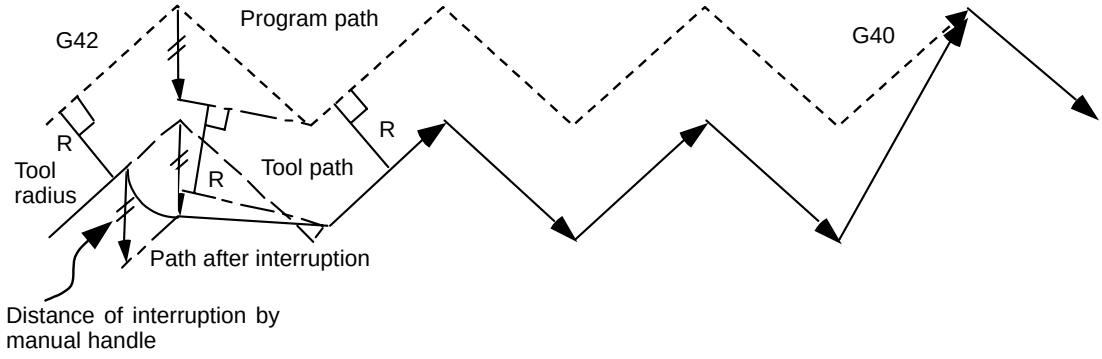
At this time, four or more blocks may not exist between

- Block for which the interruption has been completed and
- Block that contains the nose radius compensation cancel command (G40).

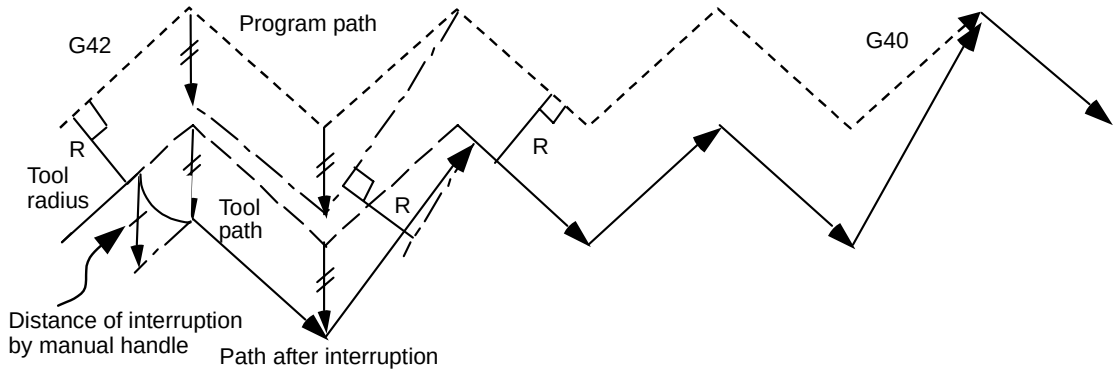
In this case, the proper tool path will return at the block next to the tool radius compensation cancel command (G40).



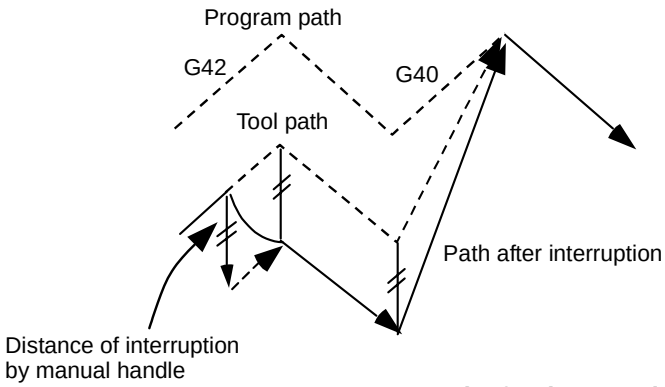
Path after interruption
(Incremental value mode, continuous operation)



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)



Path after interruption
(Absolute value mode, program absolute value update valid, continuous operation)

6.14.7 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 setup parameter "#1151 rstint"

6.14.8 Operation Sequence

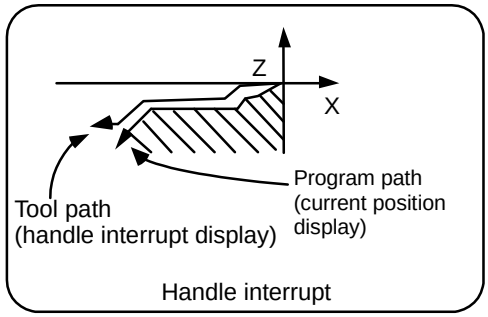
An operation example is given where auto operation of XYZ axes is executed in the memory operation mode and the Z axis is used as a handle interrupt axis.

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/ INCREMENTAL RATIO switch.

Turn the manual handle in the + or – direction and change the Z axis cut depth amount.



When M02 or M30 is executed, automatic operation will stop.

6.15 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.

6.16 Deceleration Check

6.16.1 Functions

The purpose of the deceleration check is to reduce the machine shock that occurs when the control axis feedrate is suddenly changed, and prevent corner roundness. The check is carried out at block joints.

(1) Deceleration check during rapid traverse
The deceleration check is always carried out at the block joints during rapid traverse, before executing the next block.

(2) Deceleration check during cutting feed
The deceleration check is carried out at the block joints (before executing the next block) during cutting feed when any one of the following conditions is valid.
1) When the error detect switch is ON.
2) When G09 (exact stop check) is commanded in the same block.
3) When G61 (exact stop check mode) has been selected.

(3) Designating deceleration check
The deceleration check by designating a parameter includes “deceleration check specification type 1” and “deceleration check specification type 2”. The setting is selected with the parameter “#1306 InpsTyp”.

(a) Deceleration check specification type 1 (“#1306 InpsTyp” = 0)

The G0 and G1 deceleration check method can be selected with the base specification parameter deceleration check method 1 (#1193 inpos) and “deceleration check method 2” (#1223 aux07/bit1).

Parameter	Rapid traverse command
Inpos (#1193)	G0→XX (G0+G9→XX)
0	Command deceleration check
1	In-position check

Parameter	Other than rapid traverse command (G1 : other than G0 command)	
AUX07/BIT-1 (#1223/BIT-1)	G1+G9→XX	G0→XX
0	Command deceleration check	No deceleration check
1	In-position check	

(Note 1) XX expresses all commands.
(Note 2) “#1223 aux07” is the system common parameter.

(b) Deceleration check specification type 2 (“#1306 InpsTyp” = 1)

Rapid traverse and cutting in-position are designated with the “inpos” parameter.

Parameter	Command block		
#1193 Inpos	**G0**	**G1+G9**	**G1**
0	Command deceleration check	Command deceleration check	No deceleration check
1	In-position check	In-position check	No deceleration check

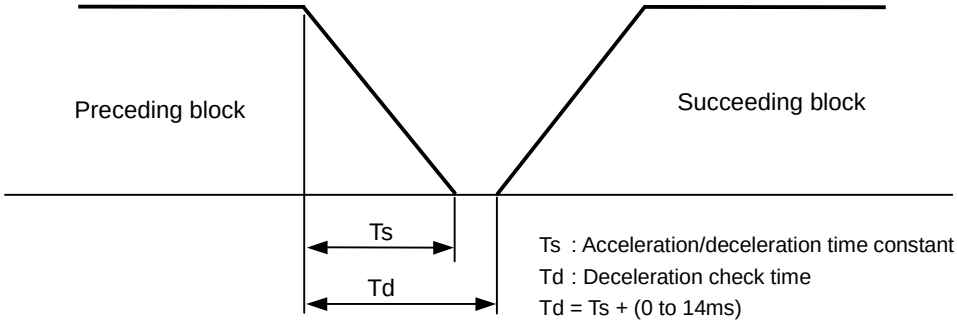
(Note 1) “#1193 inpos” is the parameter per system.
(Note 2) “G0” means the rapid traverse, and “G1” means the cutting feed.

6.16.2 Deceleration Check Method

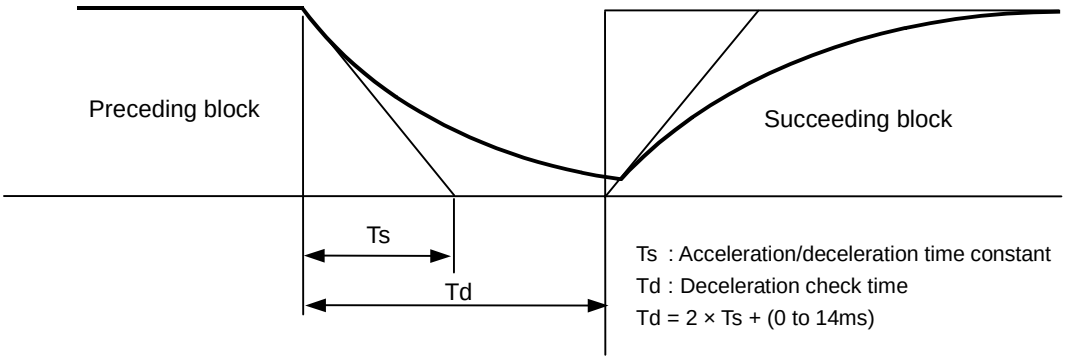
(1) Command deceleration check

After interpolation for one block has been completed, the completion of the command system deceleration is confirmed before execution of the next block.
The time required for the deceleration check is determined according to the acceleration/ deceleration mode and acceleration/deceleration time constant.

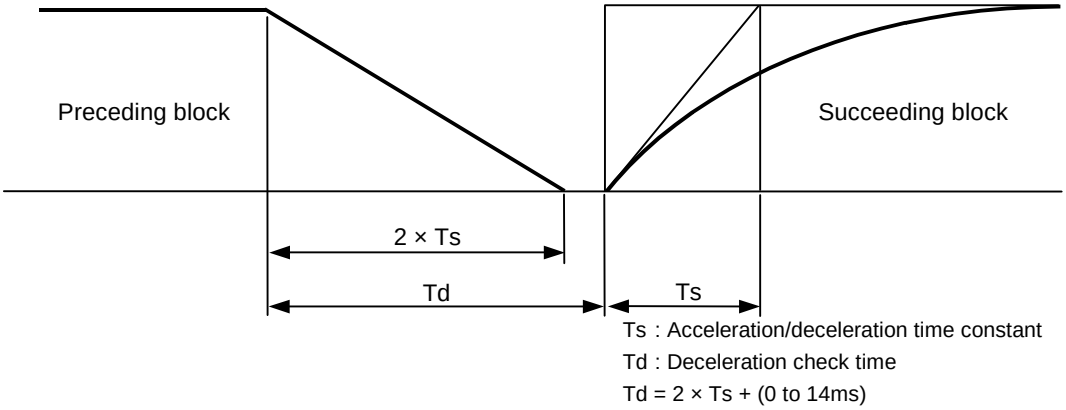
1) For linear acceleration/deceleration



2) For exponential acceleration/deceleration



3) For exponential acceleration and linear deceleration

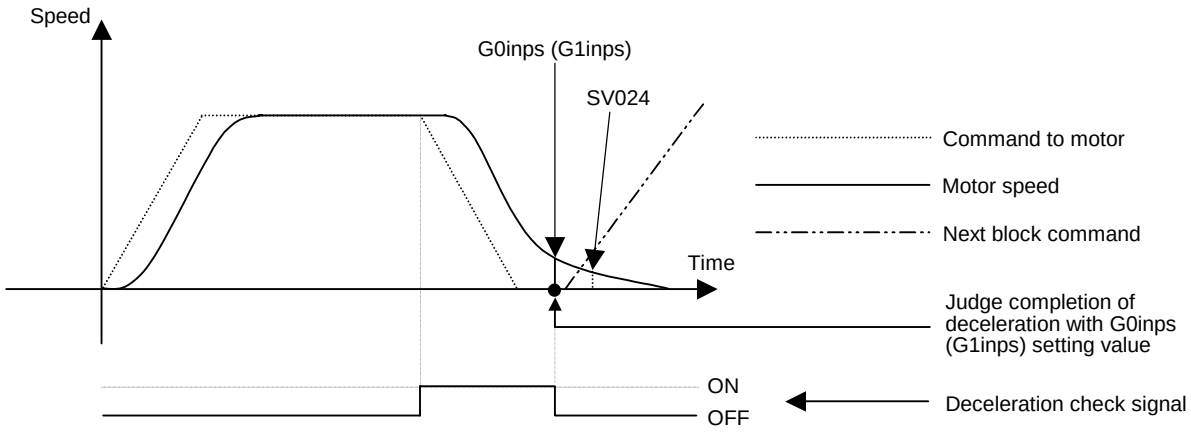


The deceleration check time required during rapid traverse is the longest rapid traverse deceleration check time of all axes. This check time is determined by the rapid traverse acceleration/deceleration mode and rapid traverse acceleration/deceleration time constant of simultaneously commanded axes.
The deceleration check time required during cutting feed is determined in the same manner. It is the longest rapid traverse deceleration check time on all axes. This check time is determined by the cutting feed acceleration/deceleration mode and cutting feed acceleration/ deceleration time constant of simultaneously commanded axes.

(2) In-position check

When the in-position check is valid, the command deceleration check is carried out. After that, it is confirmed that the servo system positional error is less than the parameter setting value, and the next block is executed.

The in-position check width can be designated with the servo parameter in-position width (SV024). However, G0 and G1 can be designated independently with the axis specification parameter G0 in-position check width (G0inps) and G1 in-position check width (G1inps). If both the servo parameter and axis specification parameter are set, the larger value will have the priority.



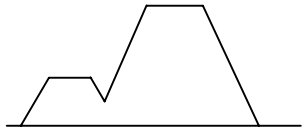
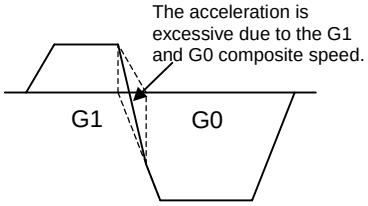
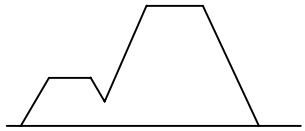
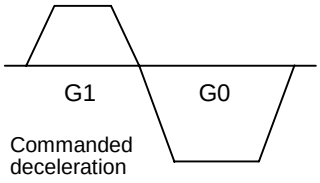
If the SV024 setting value is larger, in-position check will end when the SV024 setting value is established.

6.16.3 Deceleration Check when Opposite Direction Movement is Reversed

Deceleration check cannot be designated for G1 → G0 or G1 →G1, but it can be designated in the following manner only when the movement reverses to the opposite direction in successive blocks. Deceleration check can also be executed if even one axis is moving in the opposite direction while several axes are interpolating.

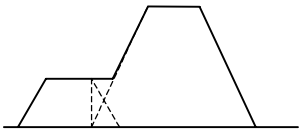
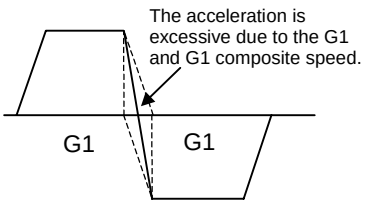
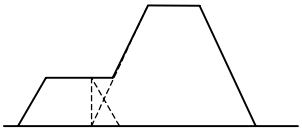
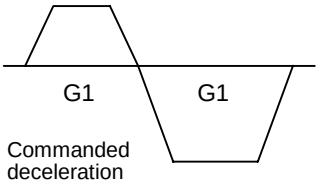
(1) Designating deceleration check for G1 → G0 opposite direction movement reversal

If the axis movement reverse to the opposite direction in a G1 to G0 successive block, the deceleration check for the movement in the opposite direction can be changed with the base specification parameter G1 → G0 deceleration check (#1502 G0Ipfg).

	Same direction	Opposite direction
G0Ipfg:0		
G0Ipfg:1		

(2) Designating deceleration check for G1 → G1 opposite direction movement reversal

If the axis movement reverse to the opposite direction in a G1 to G1 successive block, the deceleration check for the movement in the opposite direction can be changed with the base specification parameter G1 → G1 deceleration check (#1503 G1Ipfg).

	Same direction	Opposite direction
G0Ipfg:0		
G0Ipfg:1		

6.16.4 Parameters

(1) Designation of deceleration check

Base specification parameters

#	Items		Details	Setting range
1193	inpos	Change with the "#1306 InpsType deceleration check designation type". 0: Deceleration check method 1: Validate in-position check		
		Deceleration check method 1	Select the G0 deceleration check method. 0: Command deceleration check 1: In-position check	0/1
		Validate in-position check	Designate the deceleration confirmation method for the positioning and cutting command. 0: G0, G1+G9 Command deceleration check 1: G0, G1+G9 In-position check	0/1
1223	aux07 (bit1)	Deceleration check method 2	Select the deceleration check method for G1 + G9. 0: Command deceleration check when G1+G9 1: In-position check when G1+G9 Deceleration check is executed only for G1+G9. If the #1306 InpsType deceleration check designation type is "1" (deceleration check designation type 2), this parameter will be invalid.	0/1
1306	InpsTyp	Deceleration check specification type	Select the parameter specification type for the G0 or G1 deceleration check. 0: Deceleration check specification type 1 G0 is specified with “#1193 inpos”, and G1+G9 with “#1223 aux07/BIT1”. 1: Deceleration check specification type 2 G0 or G1+G9 is specified with “#1193 inpos”.	0/1

(2) Deceleration check during opposite direction travel

Base specification parameters

#	Items		Details	Setting range
1502	G0Ipfg	G1 → G0 deceleration check	0: Do not perform a deceleration check when the move direction is changed from G1 to G0. 1: Perform a deceleration check when the move direction is changed from G1 to G0.	0/1
1503	G1Ipfg	G1 → G1 deceleration check	0: Do not perform a deceleration check when the move direction is changed from G1 to G1. 1: Perform a deceleration check when the move direction is changed from G1 to G1.	0/1

(3) Designation of in-position check width

(a) Servo parameter

#	Items		Details	Setting range
2224	SV024 INP	In-position width	Set the in-position detection width value. Set the accuracy required of the machine. The positioning accuracy increases when the setting value is lowered, but the cycle time (settling time) will increase. Set "50" for ordinary operation.	0 to 32767 (μm)

(b) Axis specification parameter

#	Items		Details	Setting range
2077	G0inps	G0 in-position width	Specify the G0 in-position width (each axis) when an in-position check is performed separately for G0 and G1. (SV024 or this parameter, whichever is larger, is applied.) The standard value is 100 (50 μm). Value 0 indicates 1 μm. (Applied only for SV024)	0 to 32767 (in 0.5 μm steps)
2078	G1inps	G1 in-position width	Specify the G1 in-position width (each axis) when an in-position check is performed separately for G0 and G1. (SV024 or this parameter, whichever is larger, is applied.) The standard value is 100 (50 μm). Value 0 indicates 1 μm. (Applied only for SV024)	0 to 32767 (in 0.5 μm steps)

6.16.5 Precautions

(1) Designating deceleration check

- When in-position check is valid, set the in-position width in the servo parameters.

(2) Deceleration check for opposite direction movement reversal

- When deceleration check is valid (G0Ipfg=1), deceleration check will be executed when the axis reverses its movement to the opposite direction at the G1 → G0 successive block regardless of whether G0 non-interpolation is ON or OFF.
- When deceleration check is valid (G0Ipfg=1), deceleration check will be executed when the axis reverses its movement to the opposite direction at the G1 → G0 successive block even in the fixed cycle.
- In the G1 → G28, G1 → G29 or G1 → 30 successive blocks, deceleration check will always be executed when the G1 movement is completed, when movement to the intermediate point is completed and when movement to the return point is completed. Note that if the base specification parameter simple zero point return (#1222 aux06/bit7) is valid, the base specification parameter G1 → G0 deceleration check (G0Ipfg) will be followed when the G1 movement is completed and when movement to the intermediate point is completed. (Deceleration check will always be executed when movement to the return point is completed even in this case.)

(3) Designating in-position width

- The in-position width (programmable in-position check width) designated in the machining program has a priority over the in-position width set in the parameters (SV024, G0inps, G1inps).
- When error detect is ON, in-position check will be forcibly carried out.

(4) Deceleration check in G1 → G0, G1 → G1 opposite direction movement reversal during high-speed machining mode

- When the axis movement reverses to the opposite direction in a G1 → G1 successive block during the high-speed machining mode, the commanded deceleration will not take place even if G1Ipfg is set to 1. Note that the G0Ipfg setting will be followed if the axis direction reverse to the opposite direction in a G1 → G0 successive block.

(5) Deceleration check in movement including spindle/C-axis

- The deceleration check in a movement command including the spindle/C-axis is as the table described below when the following condition is fulfilled.

  When the different values are set for the position loop gain in non-cutting mode (spindle parameter #3203 PGCO) and the position loop gain in cutting mode (spindle parameter #3330 PGC1 to #3333 PGC4).

  That is because a vibration and so on occurs in the machine when the gain is changed during the axis movement.

Parameter	Rapid traverse command
Inpos (#1193)	G0→XX (G0+G9→XX)
0	Command deceleration check
1	In-position check

Parameter	Other than rapid traverse command (G1 : other than G0 command)	
AUX07/BIT-1 (#1223/BIT-1)	G1→G0 (G1+G9→XX)	G1→G1
0	In-position check (Applicable only to SV024)	No deceleration check
1		

(Note 1) When G1 command is issued, the in-position check is performed regardless of the deceleration check parameter.

(Note 2) XX expresses all commands.

(6) Deceleration check in polar coordinate interpolation / milling interpolation / cylindrical start / cancel command

- The deceleration check in polar coordinate interpolation / milling interpolation / cylindrical start / cancel command are as follows.

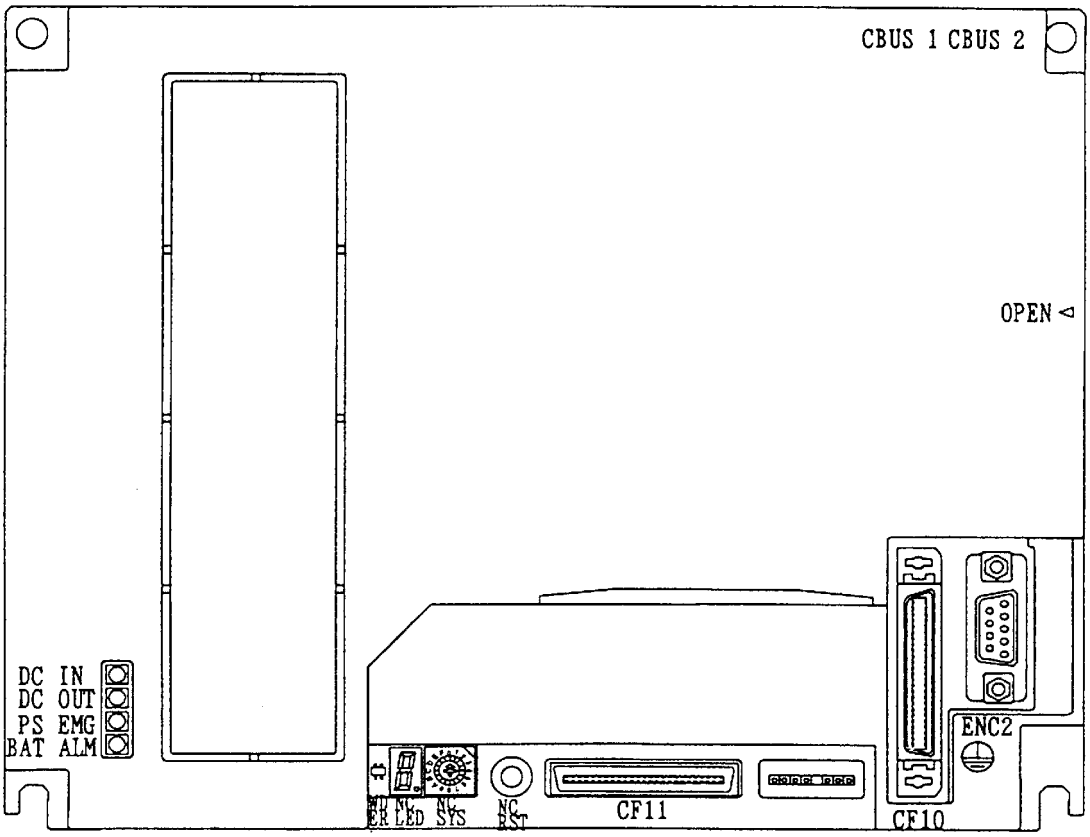
Parameter: #1223 aux07 BIT1	Deceleration check method
0	Command deceleration check
1	In-position check

III. SETUP

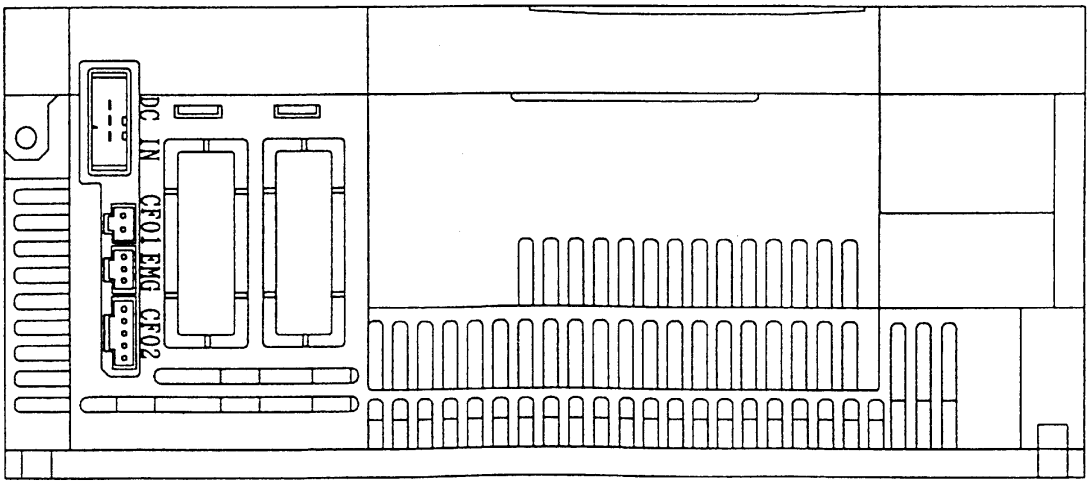
1. Switches

1.1 Layout Diagram of the Control Unit Rotary Switch

<M64A/M64 main unit>

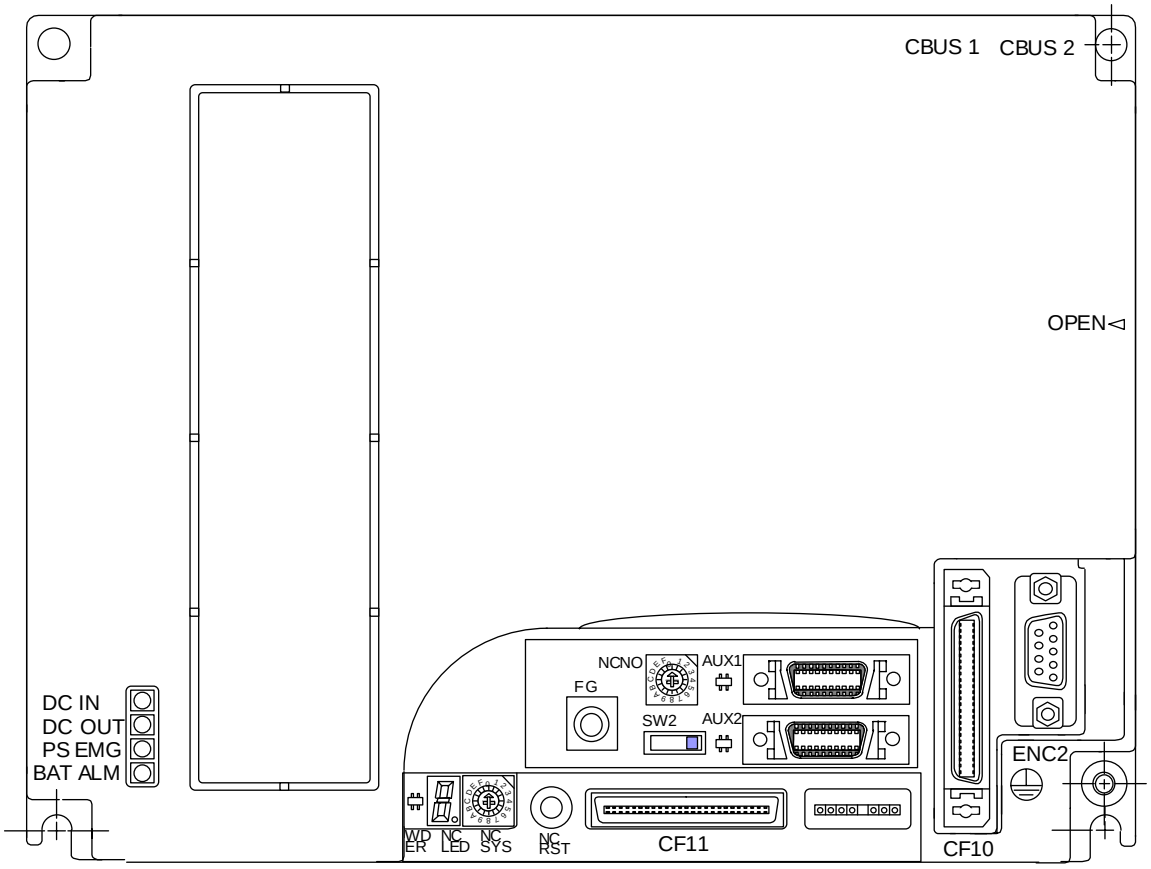


Front

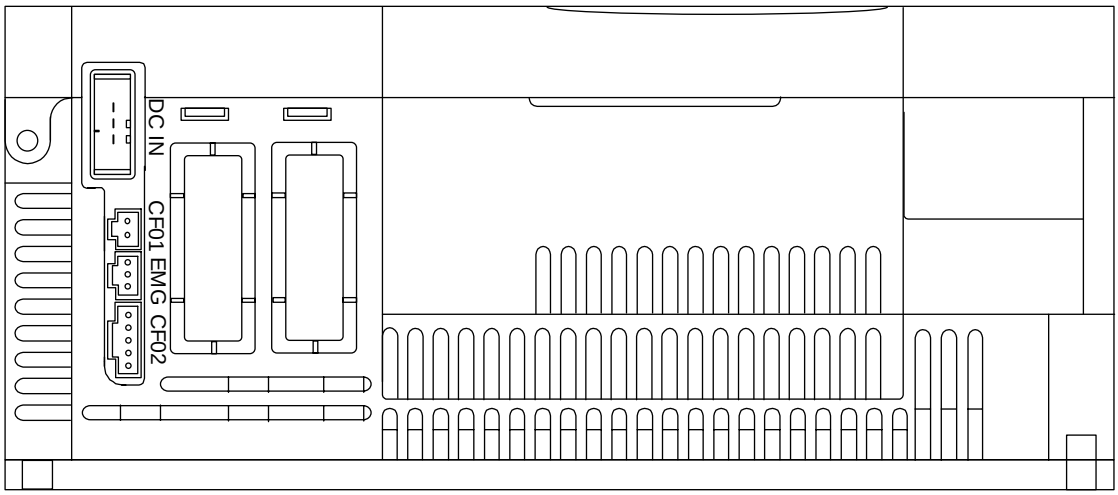


Base

<M65/M66/M64AS/M64S/M65S/M66S main unit>



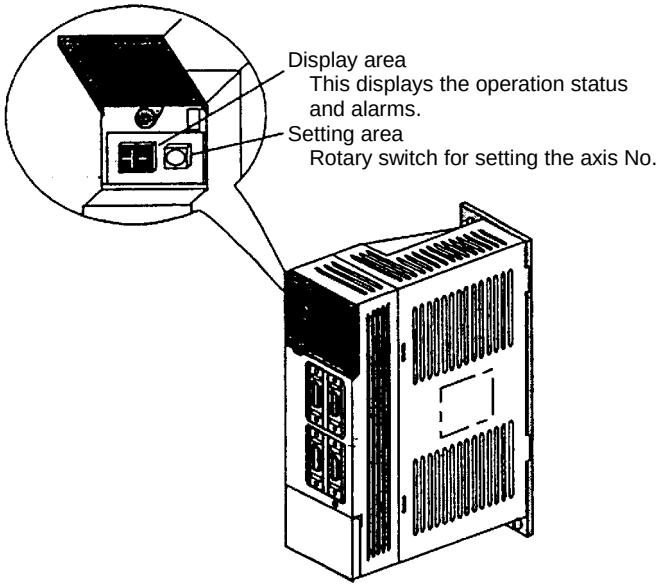
Front



Base

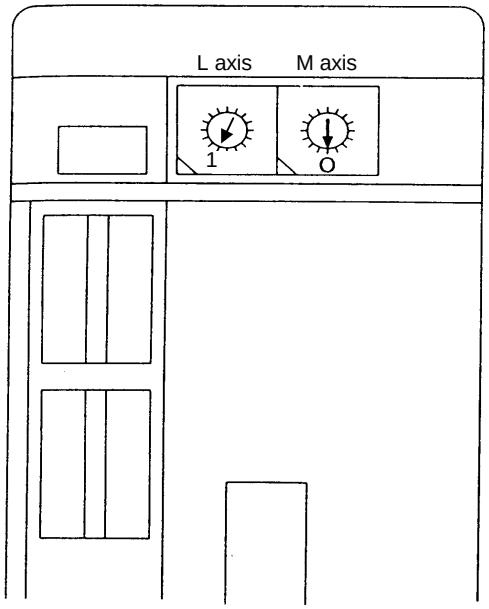
<SVJ2 Series Amplifier>

The axis No. must be set with the rotary switch before turning the power ON. The rotary switch setting is validated when the amplifier power is turned ON.



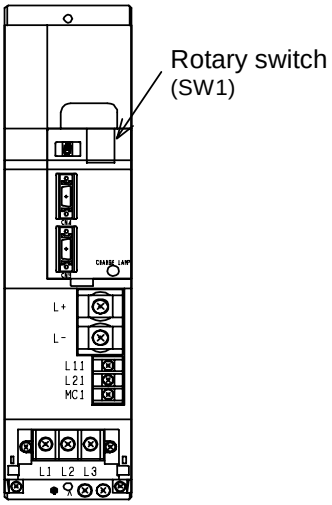
Rotary switch setting	Setting axis No.
0	1st axis
1	2nd axis
2	3rd axis
3	4th axis
4	5th axis
5	6th axis
6	7th axis
7	Not used
8	
9	
A	
B	
C	
D	
E	
F	Unused axis

<V2 Series Amplifier> 2-axis integrated amplifier



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 ~ E	Not used
	F	Unused axis selection

<CV Series Power Supply>



Set the rotary switch (SW1) as shown below.

SW1 setting	C1-CV specifications (Notes)	
0	During operation with contactor (deposits are detected)	External emergency stop
1	During operation with no contactor	When not used
2	Setting prohibited	
3		
4	During operation with contactor (deposits are detected)	External emergency stop
5	During operation with no contactor	When used
6	Setting prohibited	
7		
8		
9		
10		
11		
12		
13		
14		
15		

2. Start up and Adjustment Procedure

2.1 Confirmation of Connections

Refer to Connection Manual (BNP-B2183) to confirm the connection of each unit and communication terminals, etc.
Especially confirm the position of the input power and connection connectors, etc.
It is recommended to leave the servomotor and spindle motor drive lines disconnected until the settings of the parameters, etc., is completed.

⚠ CAUTION

⏚ Ground the signal cables to ensure stable system operation. Also ground the control unit main frame, power distribution panel and machine to one point, so they all have the same potential.

2.2 Setting of Various Switches

1) Control unit setting switch

A slide switch (SW1) and rotary switches (CS1, CS2) are located in the window on the upper front of the control unit.
Set SW1 to the "lower side" and CS1 to "0". CS2 is normally set to "0". Refer to the following settings if needed.

Switch	Operation	Application
0	Normal operation mode	Normal operation
1	PLC program stop	For PLC development work
2	Not used	
3	Maintenance mode for maker	Do not use.
4	Not used	
5	Maintenance mode for maker	Do not use.
6	Not used	
7	The entire memory will be erased, so do not set this.	Do not use.
8~F	Maintenance mode for maker	Do not use.

⚠ CAUTION

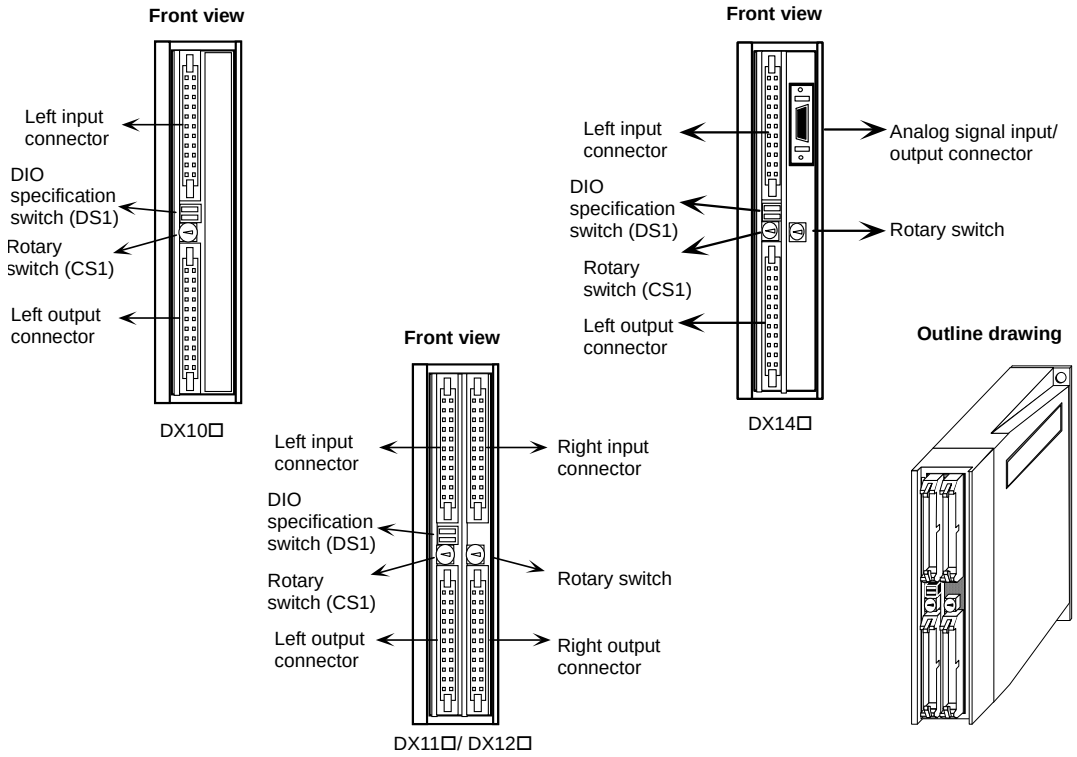
⚠ If the control unit's rotary switch is set to "7", all data in the NC will be erased and the system will not start up.

2) Remote I/O unit setting switches

There is a lever-type switch (DS1) and rotary switch (CS1) in the center from of the remote I/O unit. The DX10□ model has one of each switch, and DX11□/12□/14□ has two of each switch.
All DS1 levers must be set and fixed to "OFF: left side". The CS1 setting is "0" to "7", and should be set while referring to the following table. However, the PCB output (DO) on the right side of DX11□/12□/14□ looking from the front is a 16-point PCB, so take care.
Make sure that the CS1 setting No., is different from other CS1 setting Nos.

Remote I/O unit CS1 setting

Rotary switch CS1 No.	Read in device No.		Output device No.			
	On both units		Left side of unit		Right side of unit	
	1st system	2nd system	1st system	2nd system	1st system	2nd system
0	X00~X1F	I00~I1F	Y00~Y1F	J00~J1F	Y00~Y0F	J00~J0F
1	X20~X3F	I20~I3F	Y20~Y3F	J20~J3F	Y20~Y2F	J20~J2F
2	X40~X5F	I40~I5F	Y40~Y5F	J40~J5F	Y40~Y4F	J40~J4F
3	X60~X7F	I60~I7F	Y60~Y7F	J60~J7F	Y60~Y8F	J60~J6F
4	X80~X9F	I80~I9F	Y80~Y9F	J80~J9F	Y80~Y8F	J80~J8F
5	XA0~XBF	IA0~IBF	YA0~YBF	JA0~JBF	YA0~YAF	JA0~JAF
6	XC0~XDF	IC0~IDF	YC0~YDF	JC0~JDF	YC0~YCF	JC0~JCF
7	XE0~XFF	IE0~IFF	YE0~YFF	JE0~JFF	YE0~YEF	JE0~JEF



2.3 Turning Power ON, Memory Initialization and Parameter Settings

1) Turning power ON

Confirm the cable connections, etc., before turning the power ON. Especially take care to the power system connections.

Confirm that the communication terminal screen can be changed over after turning the power ON.

2) Setting of basic specification parameters and initialization of memory

(1) Press the [TOOL/PARAM] function selector key on the communication terminal.

(2) Press the "Menu" key corresponding to the "Menu" displayed on the bottom of the screen. Change the menu display, find the menu display "SETUP", and press that "Menu" key.

(3) The message "Open the setup parameter?" will display on the screen. Press [Y] key, and then [INPUT] key.

(4) The first page of the basic parameters for setup will display, so refer to the section on the parameters and set as required. (Parameters #1001 to 1019)

(5) Press the "NEXT PAGE" key on the right of the "Menu" keys, and display the second page of the basic parameters.

(6) Refer to the section on the parameters and set the items as with the first page. (Parameters #1025 to 1043)

(7) Set parameter #1060 SETUP on the second page to "1" (normally it is set to "0"), and perform one-touch setup.

[Procedures for one-touch setup]

(a) Set "#1060 SETUP" to "1".	→	The operation message "BASE PARA. SET? (Y/N)" will appear.
(b) Set "Y".	→	The standard parameter will be set. The operation message "FORMAT? (Y/N)" will appear.
(c) Set "Y".	→	Formatting will start.

The basic parameters and memory will be initialized.

(8) Turn the control unit's input power OFF and ON once.

(9) This completes the initialization of the memory.

3) Setting of axis specification parameters

(1) Press the [TOOL/PARAM] function selector key on the communication terminal.

(2) Press the "Menu" key corresponding to the "Menu" displayed on the bottom of the screen. Change the menu display, find the menu display "SETUP", and press that menu key.

(3) The message "Display setup parameters?" will display on the screen. Press [Y] key, and then [INPUT] key.

(4) Press the "Axis Parameter" menu key. (Five pages in all.)

(5) Refer to "Setup parameters" section and set each item.

⚠ CAUTION

⃠ Do not change setup parameters without prior approval from the machine maker.

3. Adjustment of Dog-type Reference Point Return

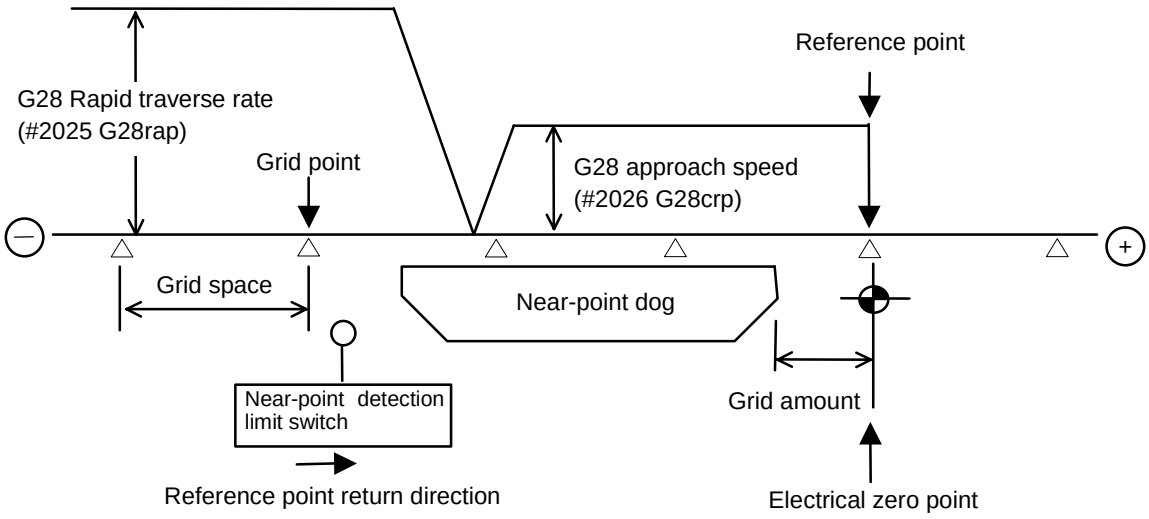
3.1 Outline

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

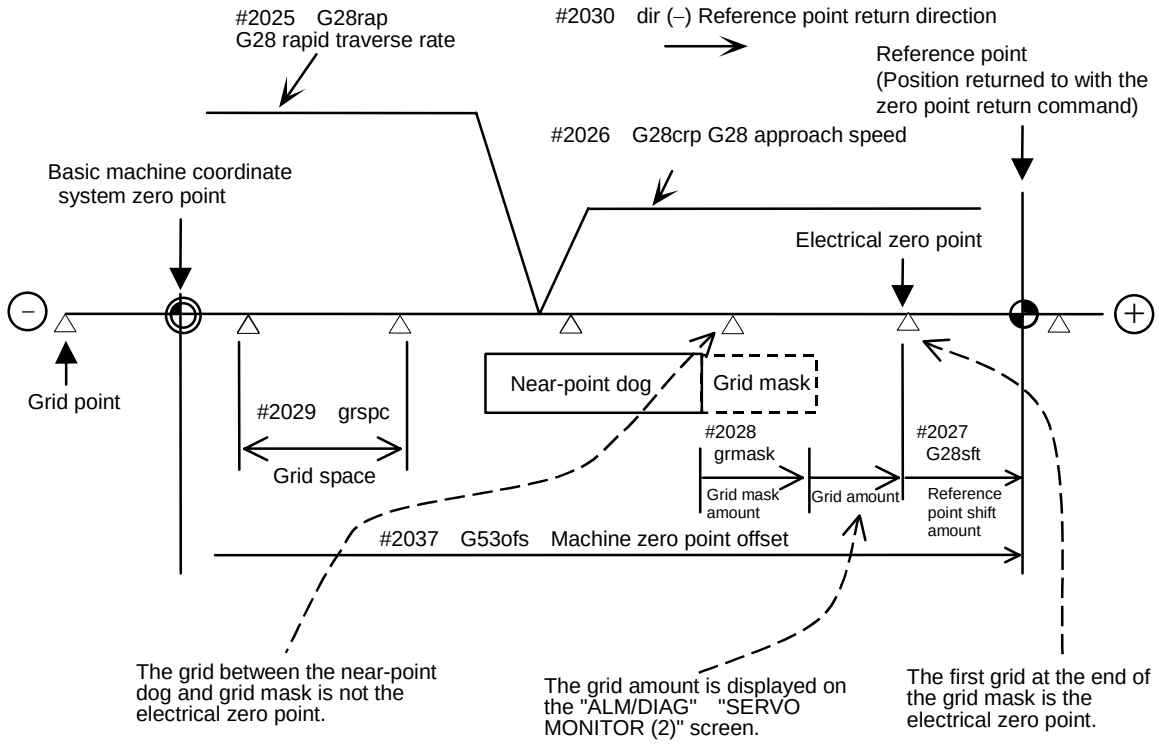
The distance between the grid points is the grid space. The grid space can be set in mm units with the SETUP PARAM. "#2029 grspc".

(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.

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)

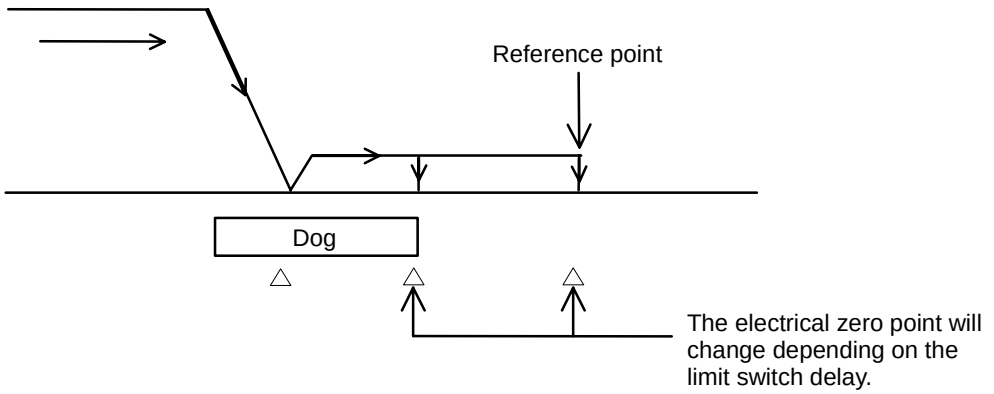
This parameter sets the approach speed to the reference point after decelerating to a stop by the near-dog detection. The G28 approach speed is accelerated and decelerated in steps (acceleration/deceleration zero), so if the speed is fast, mechanical shock, etc., could occur. The G28 approach speed should 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)

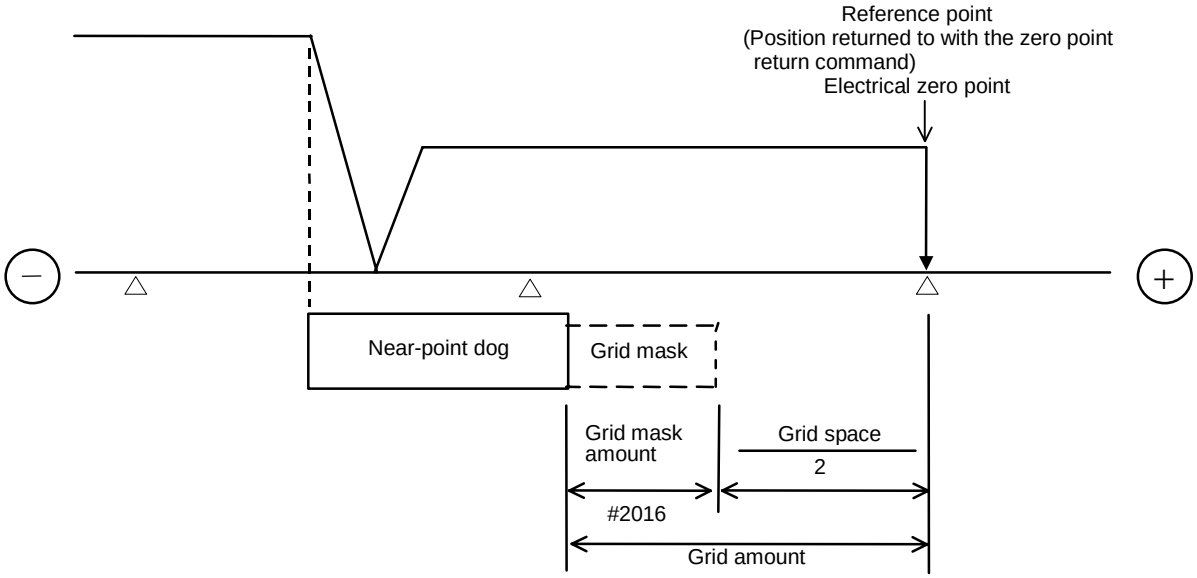
The electrical zero point is the first grid point after the dog is kicked.
If the grid point is at the position where the near-point dog is kicked, the electrical zero point will be the grid point at the position where the dog is kicked because of the delay of the limit switch operation. Thus, the next 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.



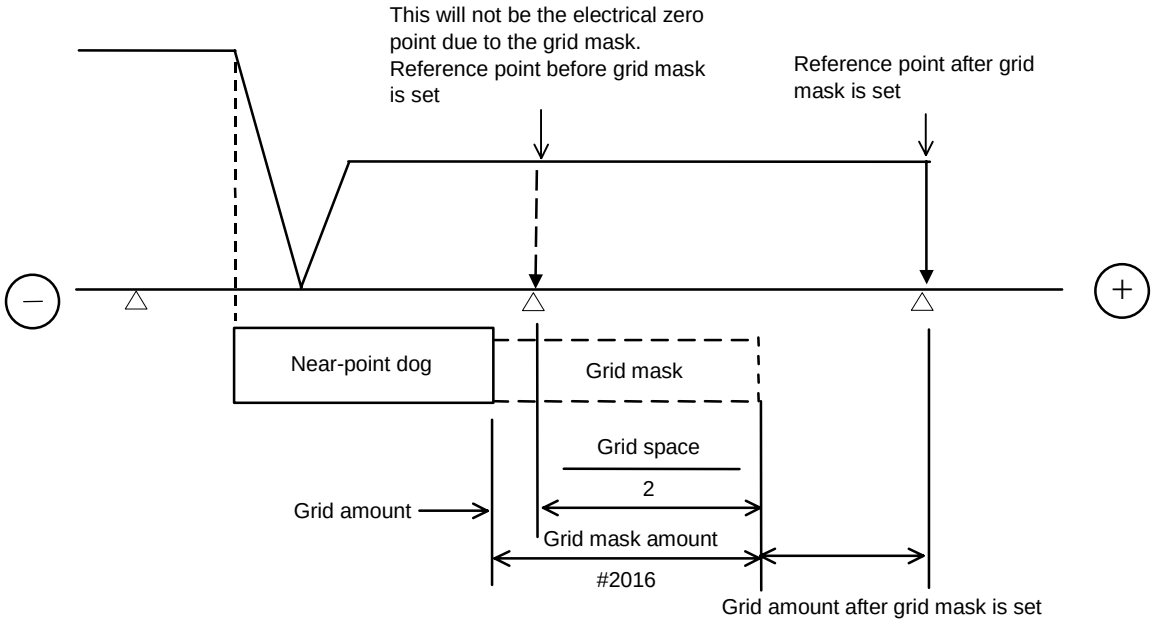
Adjustments can be made by changing the near-point dog or by setting the grid mask amount.
Setting the grid mask has the same effect as lengthening the near-point dog.
If the grid amount is near the grid space or 0, the grid point will be at the position that the near-point dog is kicked, so set a grid 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 (2)" 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 rotation axis

$$\text{Movement angle per motor rotation} = \frac{\text{Motor side gear ratio}}{\text{Machine side gear ratio}} * 360$$

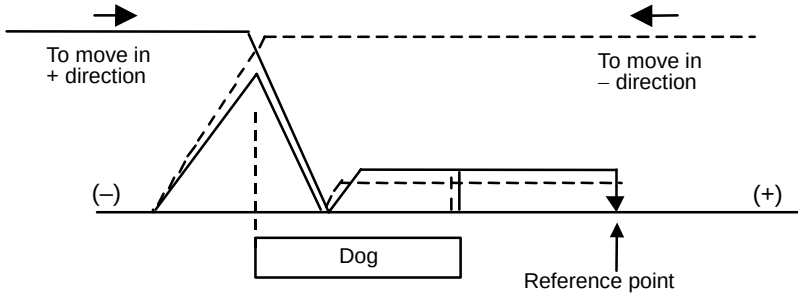
$$N = \frac{PC1}{PC2} * PIT$$

N = Movement amount per motor rotation
PC1 = Motor side gear ratio
PC2 = Machine side gear ratio
PIT = Ball screw pitch

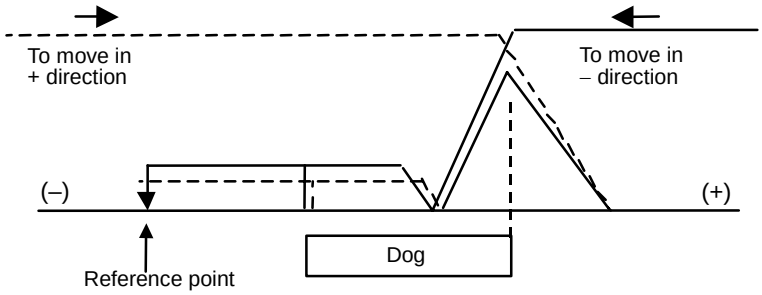
(7) Reference point return direction (#2030 dir (–))

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.

(10) Grid display selection (#1229 set01/bit6)

Select a grid type to be displayed on the servo monitor screen during dog type reference point return.
0: Selects the distance between dog OFF and zero point (including a grid mask amount).
1: Selects a value given by reducing a grid mask amount from the distance between dog OFF and zero point.

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

By detecting the machine movement amount even when the power is turned OFF, the absolute position detection function allows automatic operation without executing zero point return after the power is turned ON. The reliability of this function is extremely high as it performs various data checks such as a mutual checks of the feedback amount from the detector, and checks of the absolute position on the machine's characteristic point.
The machine zero point must be determined and the absolute position established for the absolute position detection function. The two following methods can be used depending on how the absolute position is established.

(1) Dog-less type absolute position detection
The absolute position is established by setting a random coordinate on a random position without using the dog.
The basic position can be determined with the following two methods in this method.
• Machine end stopper method
• Marked point alignment method
Manual initialization or automatic initialization can be used for the machine end stopper method.

(2) Dog type absolute position detection
The absolute position is established by executed dog type zero point return.

The validity of the absolute position detection system and the method can be set for each axis with parameters. However, the servo amplifier and detector must have the absolute position detection specifications.

4.2 Coordinate System of Absolute Position System

The basic machine coordinate zero point is created at the mechanical basic position (machine end or basic point) or at the position determined by inverting the sign of the "ZERO" parameter using the electric basic position (grid point just before the machine end or basic point) as the absolute position basic point. The reference point is created at the position of the "G53ofs" parameter using the zero point of the basic machine coordinate.

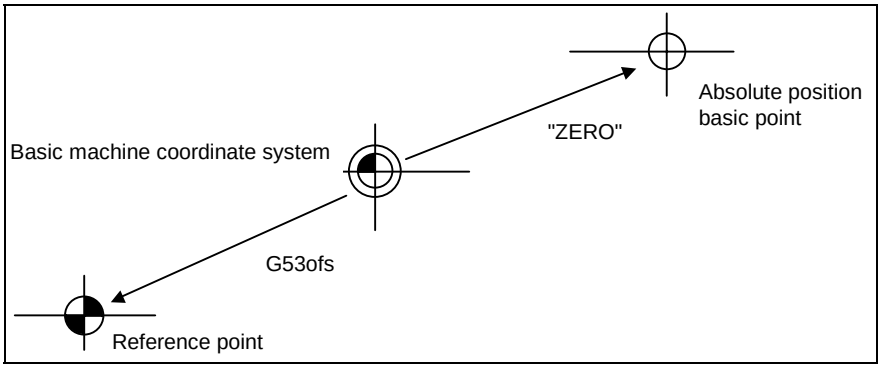


Fig. 2.2 Absolute position (dogless setting system)

"ZERO" : Coordinate value of the absolute position basic position looking from the basic machine coordinate zero point. ([ABS. POSITION SET] "#2 ZERO")
"G53ofs" : Coordinate value of the reference point looking from the basic machine coordinate zero point. ([AXIS SPEC] "#2037 G53ofs")

(Note) Select whether to use the machine basic position or electrical basic position as the absolute position basic point with parameter "#2059 zerbas".

4.3 Starting up Absolute Position Detection System

It is necessary to perform zero-point initialization before starting up the absolute position detection system. Zero-point initialization sets up the coordinate system to be ready for operation.

(1) Operation before establishment of absolute position

If an attempt is made to start up the system while zero-point initialization has never been performed or the absolute position is lost, an NC alarm occurs with the following alarm displayed and non-initialized axis name. Under this state, the NC coordinate system is undefined and subjected to the restrictions in the following table in individual modes. If this occurs, perform zero-point initialization to set up the coordinate system.

Alarm: Z70 ABS.ILLEGAL
Z71 DETECTOR EEROR

Operation in individual modes

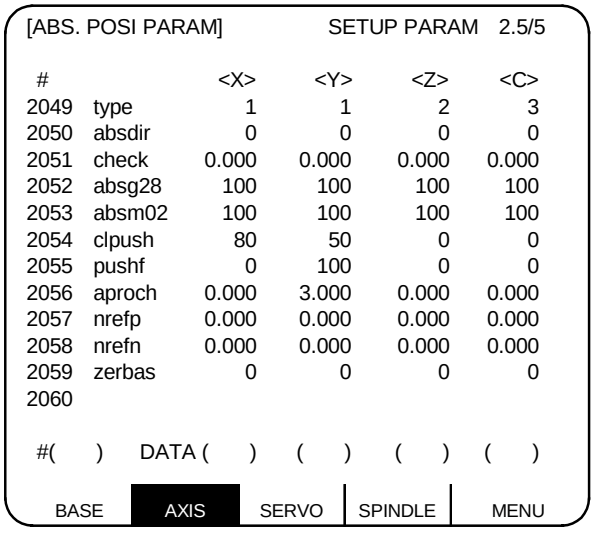
Operation mode	Absolute position detection method	
	Dog-less type	Dog type
Memory/MDI	Move command invalid (including G28) **(Note 1)**	Move command invalid (G28 is valid) **(Note 1)**
Jog feed	Valid	Valid
Rapid traverse	Valid	Valid
Handle	Valid	Valid
Step	Valid	Valid
Zero return	Start invalid **(Note 2)**	Start valid

(Note 1) A program error (P430) occurs.
(Note 2) An attempt to start the axis for which the absolute position has not been set up causes "M01 OPERATION ERROR 0024." (It is effective for the axis for which the absolute position has been set up.)

(2) Selecting zero-point initialization methods

The zero-point initialization methods can be selected on the "ABS. POSI PARAM" screen.

#2049 type 1: Dog-less machine end stopper method
2: Dog-less marked point alignment method
3: Dog type



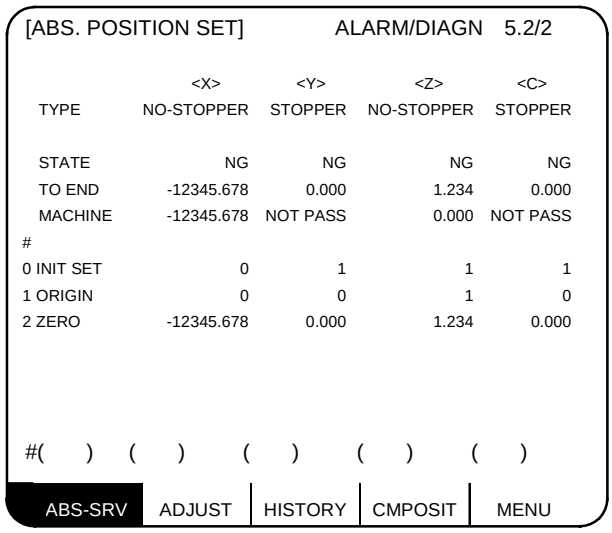
ABS. POSI PARAM screen

(Note) Refer to "MELDAS60/60S Series Alarm/Parameter Manual" (BNP-B2201) for details on the parameters.

(3) Zero-point initialization

The jog or handle is used with the "ABS. POSITION SET" screen to perform zero-point initialization. The progress of the initialization operation is displayed in the STATE field of the same screen.

The operation procedure will differ depending on the zero point initialization method. Each procedure is explained below.



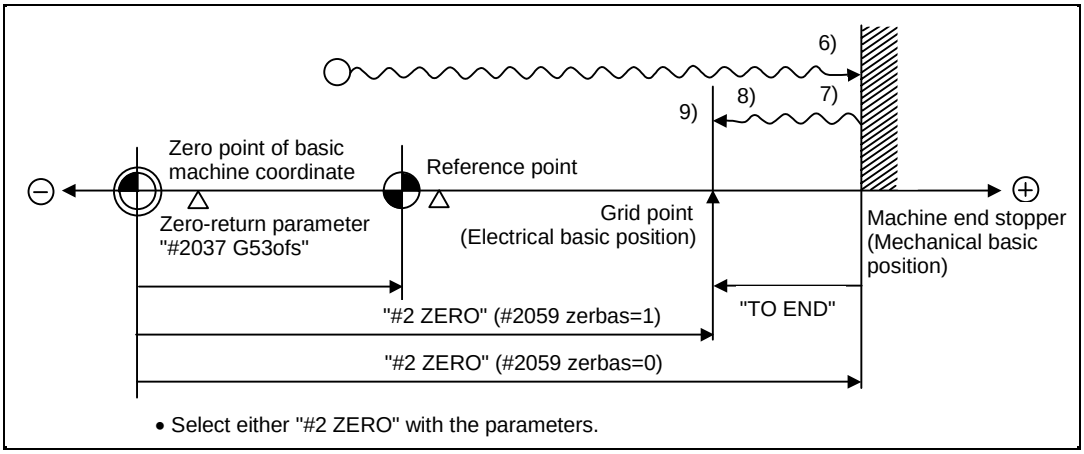
(a) Machine end stopper method (#2049 type: 1)
The machine end stopper method includes two initialization modes: manual and automatic.

(i) Manual initialization
With this method, the axis is pressed against the machine end stopper using handle or JOG. Before performing zero-point initialization, specify the following parameter (for details, see Alarm/ Parameter Manual):
#2054 clpush (Current limit)

[Operation procedure]

Operation procedure	STATE display
1. Select the "ABS. POSITION SET" screen. 2. Select the handle or jog mode. 3. Ensure that the stopper method is applied for the axis for which zero-point initialization is to be performed. ("TYPE" of [ABS. POSITION SET] screen indicates [STOPPER])	• [NG] if the absolute position is lost. • [OK] if the absolute position has been established.
4. Specify "1" to "#0 INIT. SET" for the axis for which zero-point initialization is to be performed. 5. Specify data for "#2 ZERO". 6. Press the axis against the stopper at the machine end.	[STOPPER]
7. Check that "STATE" indicates that the axis is being pressed. (After the axis is pressed against the stopper and the current limit is kept reached for a given time, "STATE" indicates [RELEASE] and "TO END" indicates the distance between the machine end and the grid point just before it.)	[RELEASE]
8. The axis moves in the opposite direction.	[ORIG-RTN]
9. The axis automatically stops at the grid point just before the stopper. • The basic machine coordinate system is automatically set. This sets up the absolute position.	[OK]
10. This completes zero-point initialization. After completion of zero-point initialization for all axes, turn power OFF and ON again.	
11. Output parameter tape.	

To change just the basic machine coordinate zero point, perform steps 4 and 5 above, and then turn the power OFF and ON.



Manual zero-point initialization

(Note 1) If pressing against the machine end is executed without passing the grip point once after turning the power ON, the message "NOT PASS" will appear. Return to a point before the last grid, and then repeat from step 6.

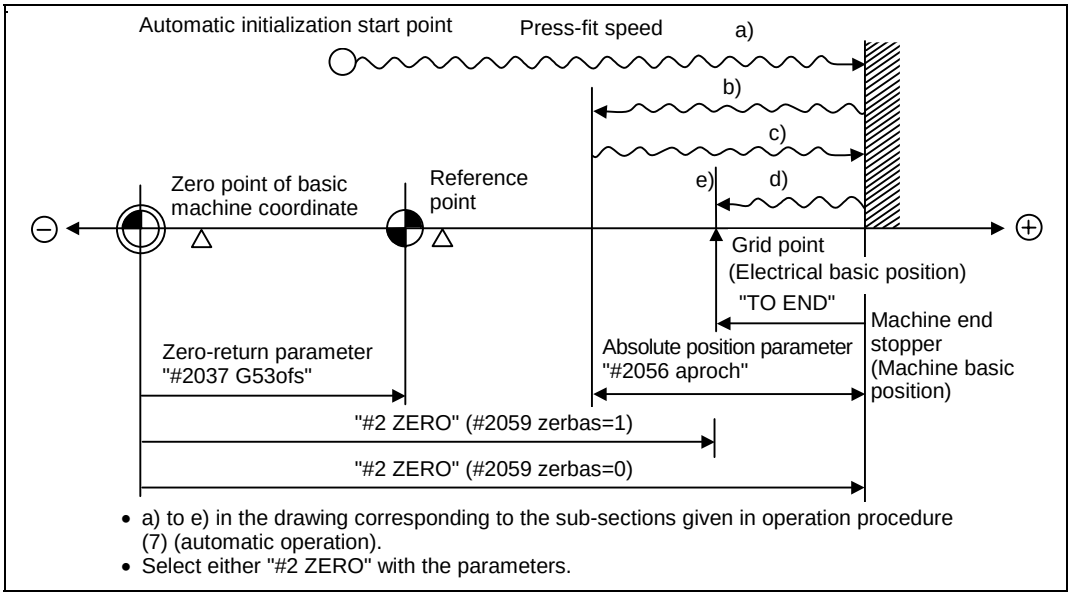
(Note 2) If the first grid point is covered by the grid mask ("#2028 grmask" on [ZERO-RTN PARAM] screen) as a result of marked point return at step 9 (returning to the basic point), the axis stops the next grid point.
Note that zero-point shift ("#2027 G28sft" on [ZERO-RTN PARAM] screen) is invalid.

(ii) 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.
It has the following merits as compared to manual initialization.
1) Pressing is always carried out under the same conditions (feed rate and distance), so displacements of zero points can be reduced.
2) Part of operation can be automated to ease zero-point initialization.
Before performing zero-point initialization, specify the following parameter on the [ABS. POSI PARAM] screen.
(for details, see Alarm/Parameter Manual):
#2054 clpush : Current limit
#2055 pushf : Push speed
#2056 aproch : Approach point

[Operation procedure]

Operation procedure		STATE display
1. Select the "ABS. POSITION SET" screen.		• NG if the absolute position is lost. • OK if the absolute position has been established.
2. Select the "INIT-SET" mode.		
3. Ensure that the stopper method is applied for the axis for which zero-point initialization is to be performed. ("TYPE" of [ABS. POSITION SET] screen indicates STOPPER)		
4. Specify "1" to "#0 INIT. SET" for the axis for which zero-point initialization is to be performed.		JOG-START
5. Specify data for "#2 ZERO".		
6. Perform jog start for the axis for which zero-point initialization is to be performed. • The jog start is available only in the direction of the sign specified for "#2 ZERO" (toward machine end stopper). (An attempt to perform the jog start in a wrong direction encounters "invalid start direction".)		STOPPER 1
7. Auto-matic opera-tion	a) The axis moves toward the machine end stopper at the "press-fit speed".	
	b) The axis hits against the machine end stopper. After the current limit is kept reached for a given time, the axis moves toward the approach point at the "press-fit speed".	ZP-RTN
	c) After it reaches the approach point, it again moves toward the machine end stopper at the "press-fit speed".	STOPPER 2
	d) The axis hits against the machine end stopper. After the current limit is kept reached for a given time, the axis moves toward the grid point just before the stopper at the "press-fit speed".	ORIG-RTN
	e) The axis stops at that first grid point. • The basic machine coordinate system is automatically set. This sets up the absolute position.	OK
8. This completes zero-point initialization. After completion of zero-point initialization for all axes, turn power OFF and ON again.		
9. Output parameter tape.		

To change just the basic machine coordinate zero point, perform steps 4 and 5 above, and then turn the power OFF and ON.



Automatic zero-point initialization

(Note 1) When the approach point is reached after the first press-fit, if the grid has not been passed once after turning the power ON, the message "NOT PASS" will display. Return to a point before the last grid, and then repeat from step 6.

(Note 2) Acceleration/deceleration during movement at the specified press-fit speed is performed in smoothing-off (step feed) mode.

(Note 3) If 0 is specified for "#2056 aproch" of the [ABS. POSI PARAM] screen, the machine zero point becomes the approach point.

(Note 4) Automatic initialization is interrupted if one of the following events occurs. If it is interrupted, "STATE" indicates JOG-START (after selection of the "INIT-SET" if it is caused by mode change), so restart operation from step 6.

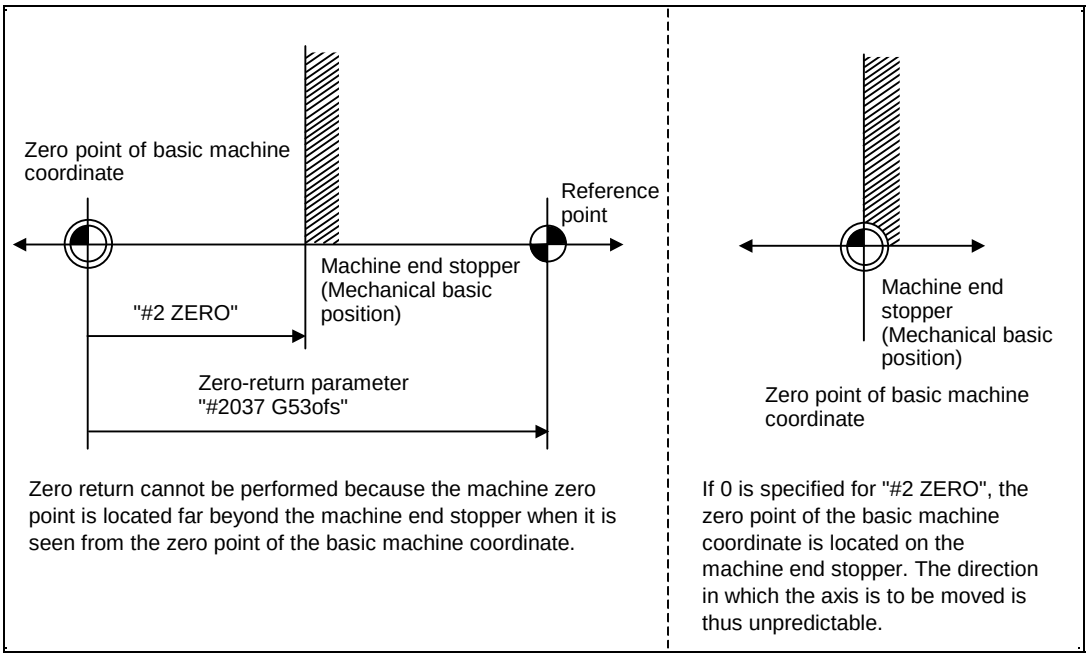
- An absolute position detection alarm occurs.
- Operation preparation turns OFF.
- The mode is changed.
- The system is reset.

If "STATE" is OK before automatic initialization is started, "STATE" returns to OK if power is turned OFF and ON again without restarting the operation.

(Note 5) If the first grid point is covered by the grid mask ("#2028 grmask" on [ZERO-RTN PARAM] screen) as a result of marked point return at step 7 e), the axis stops at the next grid point.
Note that zero-point shift ("#2027 G28sft" on [ZERO-RTN PARAM] screen) is invalid.

(Note 6) Automatic initialization cannot be started if one of the following event occurs. An attempt to start it encounters the message "T01 CAN'T CYCLE ST".
- Data has not been specified for "#0 INIT. SET" parameter.
- Invalid data has been specified for the "#2 ZERO" parameter.
- Data has not been specified for "#2055 pushf" of the [ABS. POSI PARAM] screen.
- "Z71 DETECTOR ERROR 0005" has occurred.

Invalid data for the "#2 ZERO" parameter indicates that the relationship between "#2 ZERO" and zero return parameter "#2037 G53ofs" is inadequate. That is, if the former is smaller than the latter, the machine end stopper would be located between the zero point of the basic machine coordinate and machine zero point; this disables automatic initialization. If the former is 0, the direction of the machine end stopper is unpredictable; this also disables automatic initialization. See the figure below.



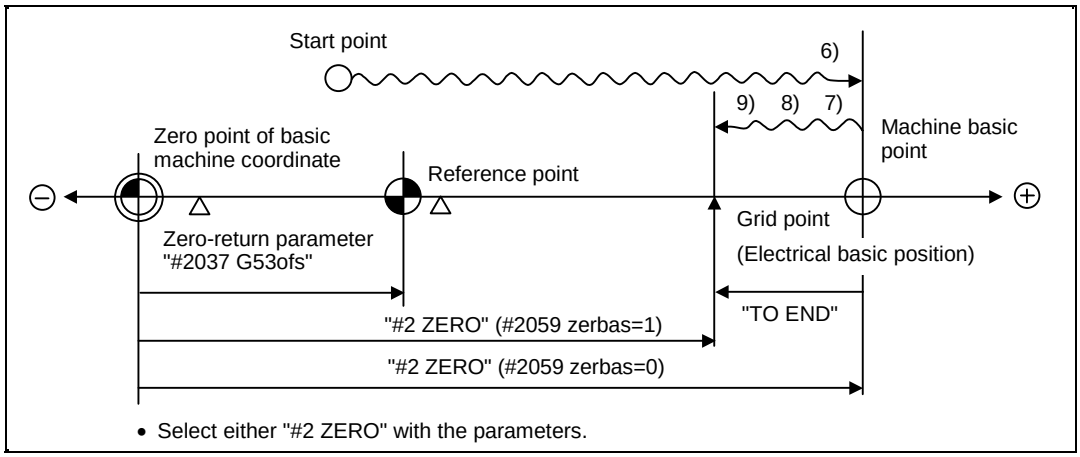
Explanation of causes that disable automatic initialization

(b) Marked point alignment method
The handle or jog is operated to align the axis with the basic point of the machine.
Before performing zero-point initialization, specify the following parameter of the [ABS. POSI PARAM] screen (for details, see Alarm/Parameter Manual):
#2050 absdir

[Operation procedure]

Operation procedure	STATE display
1. Select the "ABS. POSITION SET" screen.	• NG if the absolute position is lost. • OK if the absolute position has been established.
2. Select the handle mode, handle axis or jog mode.	
3. Ensure that the origin point alignment method is applied for the axis for which zero-point initialization is to be performed. ("TYPE" of [ABS. POSITION SET] screen indicates NO-STOPPER.)	
4. Specify "1" to "#0 INIT. SET" for the axis for which zero-point initialization is to be performed.	ORIGIN SET
5. Specify data for "#2 ZERO".	
6. Move the axis to the machine basic position and align it with the basic point.	
7. Specify "1" to "#1 ORIGIN".	
8. The machine will move in the direction specified with [ABS. POSI PARAM] "#2050 absdir".	ORIG-RTN
9. The axis automatically stops at the first grid point. • The basic machine coordinate system is automatically set. This sets up the absolute position.	OK
10. This completes zero-point initialization. After completion of zero-point initialization for all axes, turn power OFF and ON again.	
11. Output parameter tape.	

To change just the basic machine coordinate zero point, perform steps 4 and 5 above, and then turn the power OFF and ON.



Zero-point initialization by origin point alignment method

(Note 1) If the basic point is set when the grid has not been passed once after turning the power ON, the message "NOT PASS" will display. Return to a point before the last grid, and then repeat from step 6.

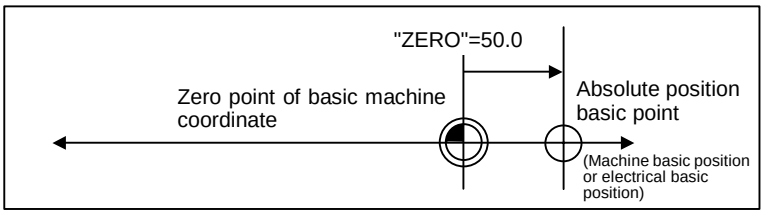
(Note 2) If the first grid point is covered by the grid mask ("#2028 grmask" on [zero-return parameter] screen) as a result of marked point return at step 9, the axis stops at the next grid point.
Note that zero-point shift ("#2027 G28sft" on [ZERO-RTN PARAM] screen) is invalid.

(Note 3) Reconfirm the absdir direction if the machine does not move in step 8. The machine will move only in the positive direction when set to 0, and the negative direction when set to 1.

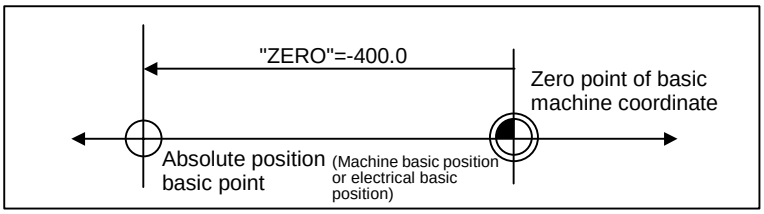
(c) Common precautions for dogless type absolute position detection

(i) Examples of setting "#2 ZERO" parameter
The coordinate value of the absolute basic point (mechanical basic position or electrical basic position) looking from the basic machine coordinate zero point is set for "#2 ZERO" parameter.

(Example 1)
The zero point is determined at the position 50.0 to the front of the absolute position basic position on the positive side.

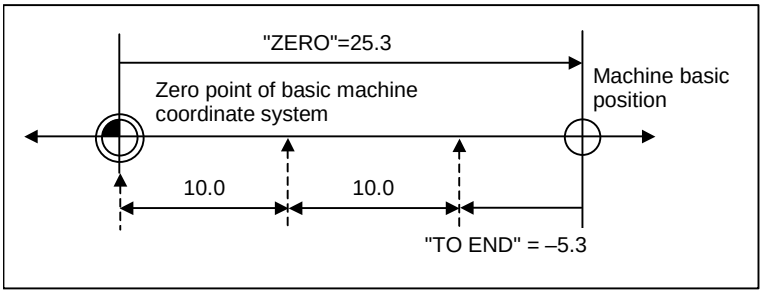


(Example 2)
The zero point is determined at the position 400.0mm to the front from the machine basic position or absolute position basic point at the negative side.



(Example 3)
If it is desired to create the zero point of the basic machine coordinate on a grid point, the value indicated in "TO END" is used to calculate the value to be set to the "#2 ZERO" parameter as shown in the example below. The value indicated in "TO END" is the distance from the machine basic position to the grid point right before the end. (If the coordinates of the absolute position basic point are used for "#2 ZERO", TO END does not need to be considered.)

The zero point is determined at the third grid point (10.0mm grid-point intervals) when "TO END" indicates –5.3 at the basic position at the positive side. (Example for 10.0mm grid interval.)

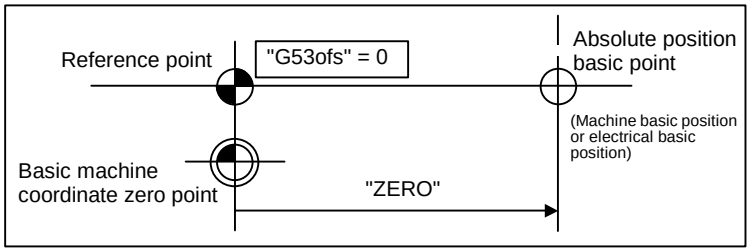


(ii) Setting of reference point

The reference point can be created in the following method by setting the "#2037 G53ofs" of the [ZERO-RTN PARAM] screen.

(Example 1)

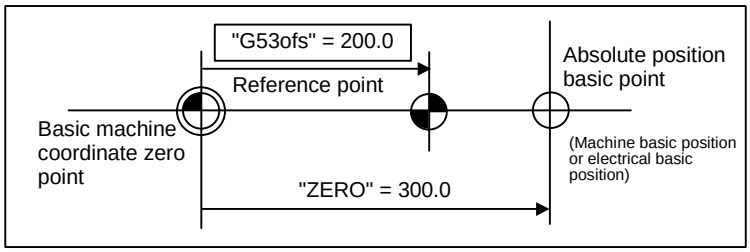
To set the reference point at the same point as the basic machine coordinate zero point.



(Example 2)

To set the reference point at a position 200.0 to the front of the basic machine coordinate zero position on the positive side.

(When the basic machine coordinate zero point is 300.0mm to the front of the absolute position basic position.)



(iii) Common precautions for initialization

(Note 1) "#0 INIT. SET" parameter setting (axis for which zero point is to be initialized) can be set for all axes or for one axis.

(Note 2) "#0 INIT. SET" parameter cannot be turned OFF with the keys. The power must be turned OFF once. If even one axis is ON for this parameter, the message "INITIAL SET (ABS)" will display on the operation status section of all screens, and the axis in initialization will be output. The automatic, MDI and manual zero point return operation will be interlocked at this time.

(Note 3) The "#2 ZERO" setting can be made at any level is "#0 INIT. SET" parameter is set to "1".

(Note 4) The grid point must be passed once after the power is turned ON to execute zero point initialization. If the grid point has not been passed once, "NOT PASS" will appear at "MAC POS".

(Note 5) After the absolute position has been established, save the necessary data in the memory.

(5) Zero point initialization for dog type absolute position detection

By executing dog type zero point return with the manual zero point return mode or automatic zero 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]

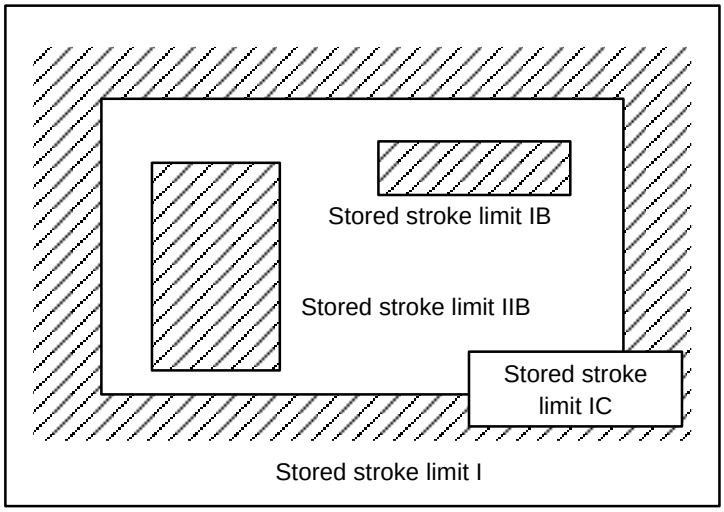
Operation procedure	STATE display
1. Select the [ABS. POSITION SET] screen.	• NG if the absolute position is lost. • OK if the absolute position is established.
2. Confirm that the axis to be zero point initialized is a "dog type" axis. (See "TYPE" on the [ABS. POSITION SET] screen.)	
3. Perform manual or automatic dog type zero point return.	ZP-RTN
4. The basic machine coordinate system will be established when the zero point is reached. The absolute position is established at this stage, and the zero point initialization is completed.	COMPLETE
5. Output the parameters after zero point initialization is completed for all axes.	

(Note 1) If the dog type zero 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 zero point return, zero point return can be executed again even if the "STATE" is OK.

5. Stored Stroke Limit

(1) Outline

Three tool entry prohibited ranges can be set with stored stroke limit I, stored stroke limit II, 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.
The stored stroke limit II and IIB select the outer side (II) and inner side (IIB) of the entry prohibited range with parameters.



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.

[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.
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

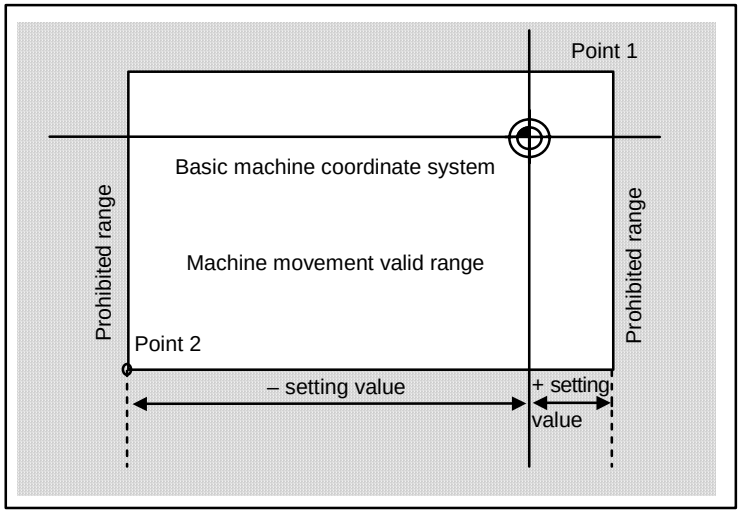
The stored stroke limit sets a prohibited range with the parameters or program command. The maximum and minimum values of the prohibited range are set as coordinate values (radius values) on the basic machine coordinate system for each axis. If the parameters (#8204 OT-CHECK-N and #8205 OT-CHECK-P), for setting the prohibited range, are set to the same value, the stroke will not be checked. This function is valid only for the axis for which the reference point is established.
If 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.

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 narrow range designated by the two types becomes the movement valid range.		#2013 OT– #2014 OT+	• Zero 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.	• #8210 OT-INSIDE:0 • Used with I.	#8204 OT-CHECK-N #8205 OT-CHECK-P	• Zero 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+	• Zero 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_1C type: 2

5.1 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 narrow range designated by the two types becomes the movement valid range.

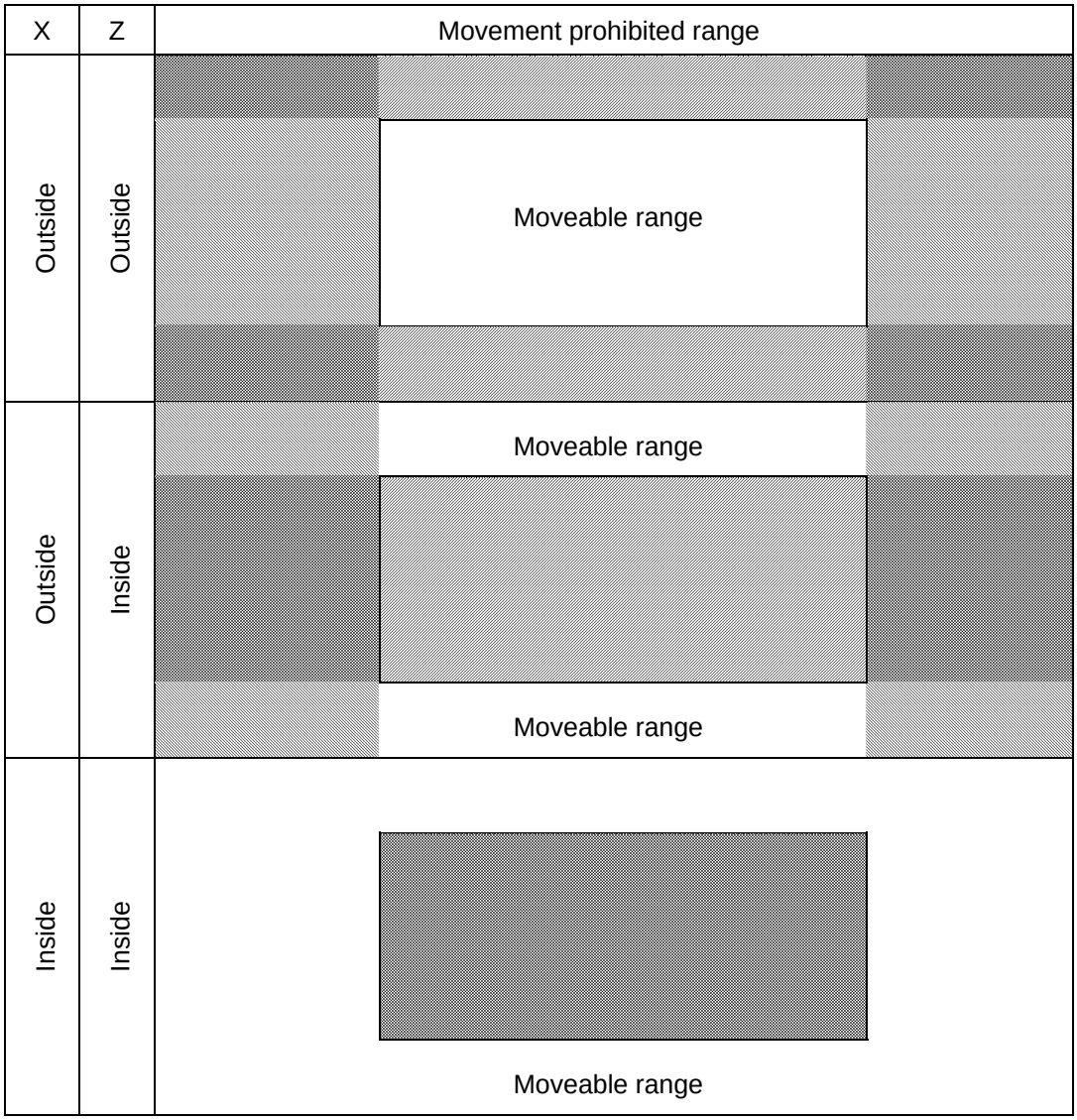


The following values are set with the coordinate values in the basic machine coordinate system.
Point 1: #2014 OT+
Point 2: #2013 OT–

5.2 Stored stroke limit II

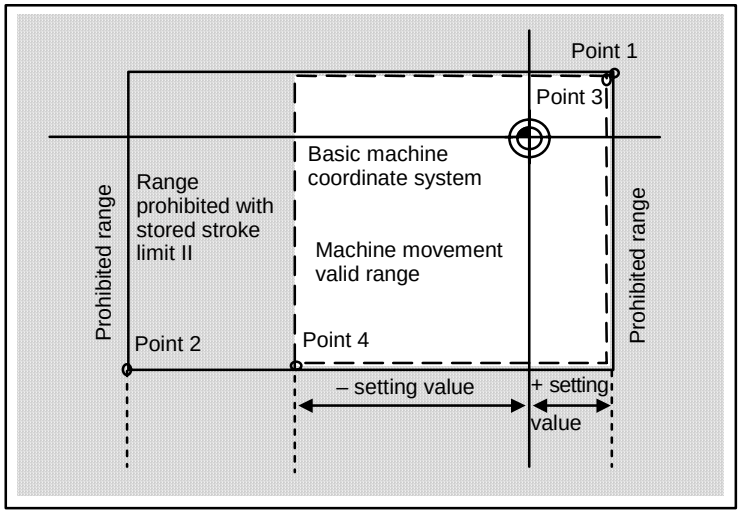
The boundary is set with the parameters (axis parameters #8204 OT-CHECK-N, #8205 OT-CHECK-P) or with the program commands. Either the inside or the outside of the set boundary is the prohibited range. Whether the inside or outside of the range is prohibited is determined by parameter (#8210 OT-INSIDE). 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 narrow range designated by the two types becomes the movement valid range.

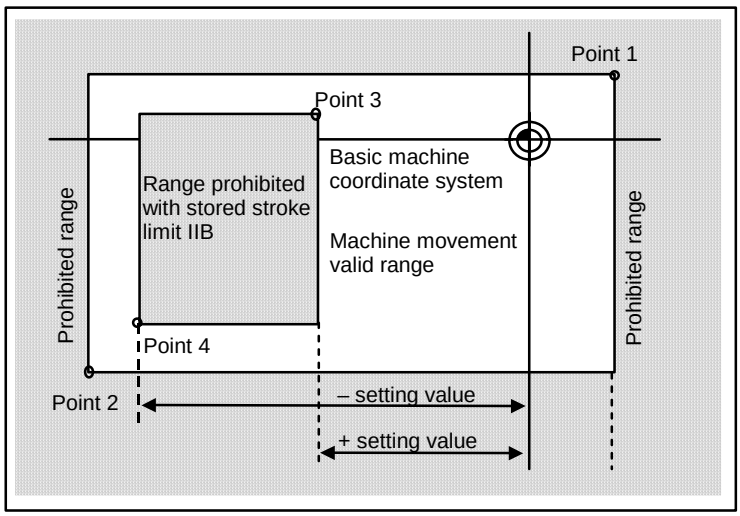


The following values are set with the coordinate values in the basic machine coordinate system.
Point 3 and 4:
#8205 OT-CHECK-P
#8204 OT-CHECK-N

Points 1 and 2 are the prohibited range set with stored stroke limit I.

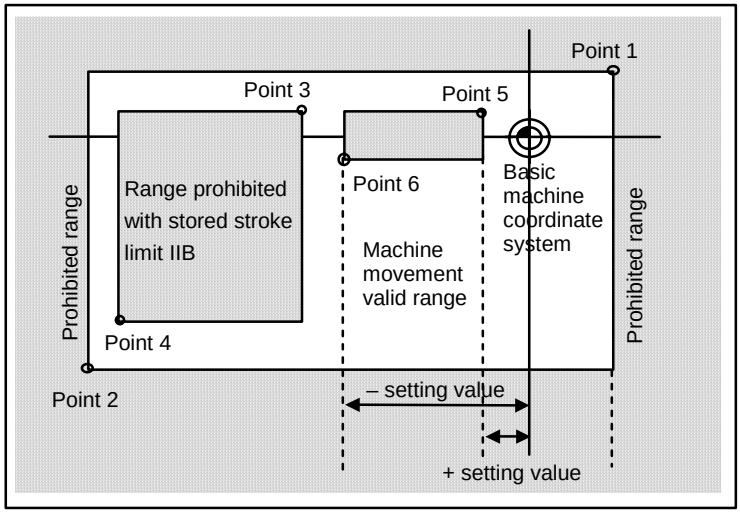
(2) Stored stroke limit IIB (When prohibited range is on inside)

A range except for that of the stored stroke limit I becomes the movement prohibited range.



5.3 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.

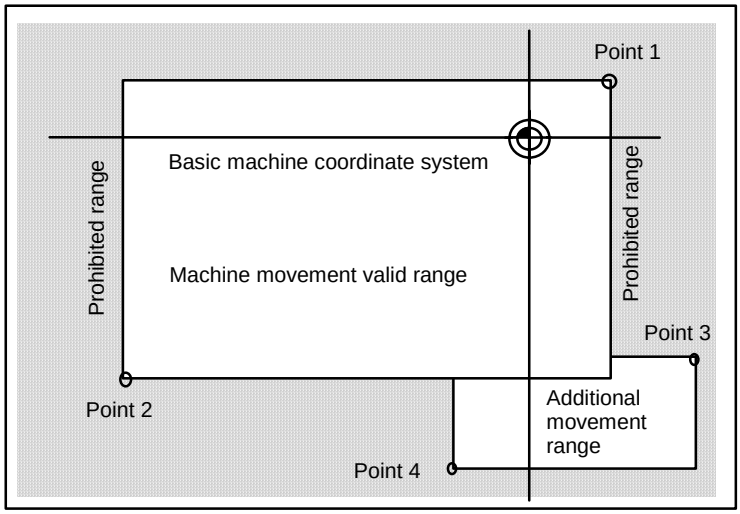


The following values are set with the coordinate values in the basic machine coordinate system.
Point 3 : #2062 OT_1B+
Point 4 : #2061 OT_1B–

Points 1 and 2 are the prohibited range set with stored stroke limit I, and points 3 and 4 are the prohibited range set with stored stroke limit II.

5.4 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 movement valid range.
This is valid when the axis parameter #2063 is set to 2, and cannot be used with soft limit IB.

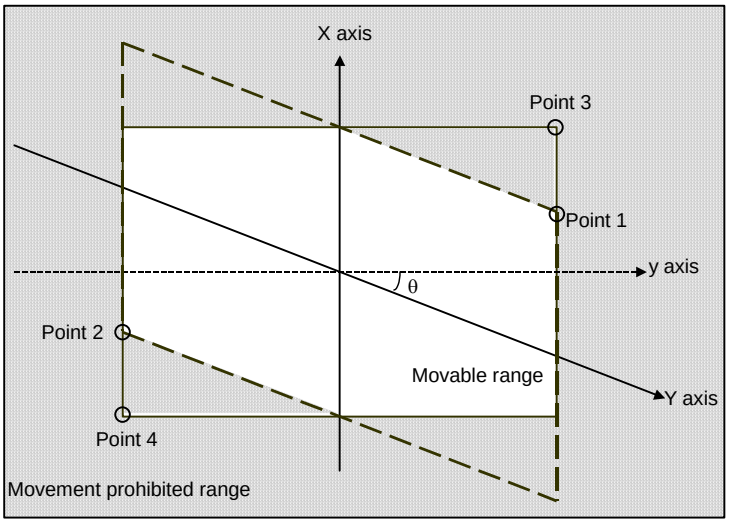


The following values are set with the coordinate values in the basic machine coordinate system.
Point 3 : #2062 OT_1B+
Point 4 : #2061 OT_1B–

Points 1 and 2 are the prohibited range set with stored stroke limit I.

5.5 Movable Range during Inclined Axis Control

By setting #2063 OT_1Btype to "3", the inclined axis control axis can be checked with the program coordinates using the stored stroke limit IB/IC range setting (#2061, #2062). The stored stroke limit IB and IC cannot be used together at this time.
By using this function with stored stroke limit I, the check can be carried out simultaneously with the actual axis and program coordinate value. In this case, the range that does not fit into either of the following two prohibited ranges will be the movable 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–

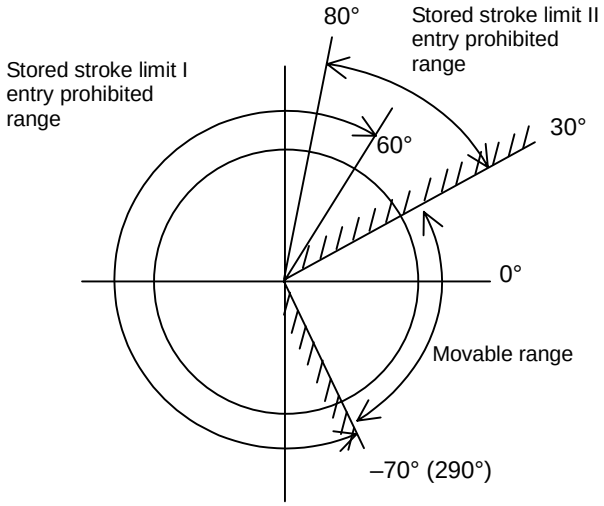
Points 1 and 2 are the prohibited range set with stored stroke limit I.

5.6 Stored Stroke Limit for Rotation Axis

Stored stroke limits I and II are used as the stored stroke limit for the rotation axis. The area between the maximum and minimum values of the prohibited range's parameters, which does not contain the 0 point of the basic machine coordinate system, is the entry prohibited range.
The prohibited range parameters for the rotation axis can be set to establish "maximum value < minimum value". This will be handled in the same manner as if "maximum value > minimum value".

(Example)

Stored stroke limit I (maximum value and minimum value of prohibited range parameter)
#2013 OT–: –70.000°
#2014 OT+: 60.000°
Stored stroke limit II (maximum value and minimum value of prohibited range parameter)
#8204 Soft limit –: 30.000°
#8205 Soft limit +: 80.000°



(Note) Do not use stored stroke limits IB, IIB or IC. Invalidate stored stroke limits IB, IIB and IC by setting the parameters as shown below.
#8210 OT INSIDE: 0 (stored stroke limit II valid, IIB invalid)
#2061, #2062 set to same value (stored stroke limits IB and IC invalid)

5.7 Precautions

(1) If the maximum value and minimum value of the stored stroke limit's prohibited range are set to the same value, 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) 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.

(3) Make sure that the lower limit value of the stored stroke limit IC setting value is "smaller than the upper limit value".

(4) The axis parameters #2063, #2061 and #2062 can be changed with DDB.

6. Daily Maintenance and Periodic Inspection and Maintenance

6.1 Maintenance Tools

(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 measures 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 HR081/HR082/HR083 DC output
Phase rotation meter		To check the connection order of the AC 3-phase input power supply.
Oscilloscope		General measurement and simple troubleshooting

(Note 1) Currently, a high-accuracy digital multi-meter is commonly used as a tester. This digital multi-meter can be used as both an AC voltmeter and an DC voltmeter.

(Note 2) A logic analyzer (sampling cycle 200MHz or more) is required for complicated troubleshooting.

(2) Tools

Screwdriver (large, medium, small)
Radio pliers

6.2 Maintenance Items

Maintenance is categorized into daily maintenance items (items to be carried at set intervals) and periodic maintenance items (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		Once/two months (Accordingly when dirty)	
Periodic maintenance	Battery (lithium battery)	Cumulative data holding time 45,000 hr	When battery voltage drop caution alarm occurs (Guideline: approx. 5 years)	Refer to Section 6.3.2 (1).
	Cooling fan (control section)	30,000 hr	Refer to left.	
Other consumable parts	Operation board	10^6 punches	Refer to left.	

6.2.1 Escutcheon

(1) Cleaning the escutcheon

1) Keep the rear side of the escutcheon as clean as possible.
2) 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.

6.2.2 LCD Panel

(1) Handling the LCD panel

(a) Precautions for use

1) The polarizing plate (display surface) of the LCD panel surface can be easily scratched, so be careful during handling.
2) 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.
3) 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.
4) Wipe off any dirt, dust, etc., on the polarizing plate using absorbent cotton or other soft cloth.
5) A CMOS LSI is used in the LCD panel, so be careful of static electricity when handling.
6) Never disassemble the LCD panel. Doing so will damage the panel.

(b) Precautions for storage

1) Do not store the LCD panel in locations having a high temperature or humidity. (Store within the storage temperature range.)
2) When storing the LCD panel as an individual unit, be sure that other objects do not touch or hit the polarizing plate (display surface).
3) When storing the LCD panel for long 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 for the 7.2-inch type and 25,000 hours for the 10.4-inch type at 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.
Deterioration of the backlight can be prevented by turning OFF the backlight with the screen saver function when the screen is not used for a long time.

(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.

(3) Replacing the backlight

Replacement should be commissioned to the Mitsubishi Service Center.

6.2.3 ATA Memory Card

(1) Handling the PCMCIA card

The general handling methods for the PCMCIA card are described below.
Refer to the instruction manual of the PCMCIA card used for details.

(a) Precautions for use

1) Insert the card in the correct direction.
2) Do not touch the connector area with the hands or metal.
3) Do not apply excessive force to the connector area.
4) Do not subject the card to bending or strong impacts.
5) Do not open the cover or disassemble the card.
6) Do not use the card in dusty locations.

(b) Precautions for storage

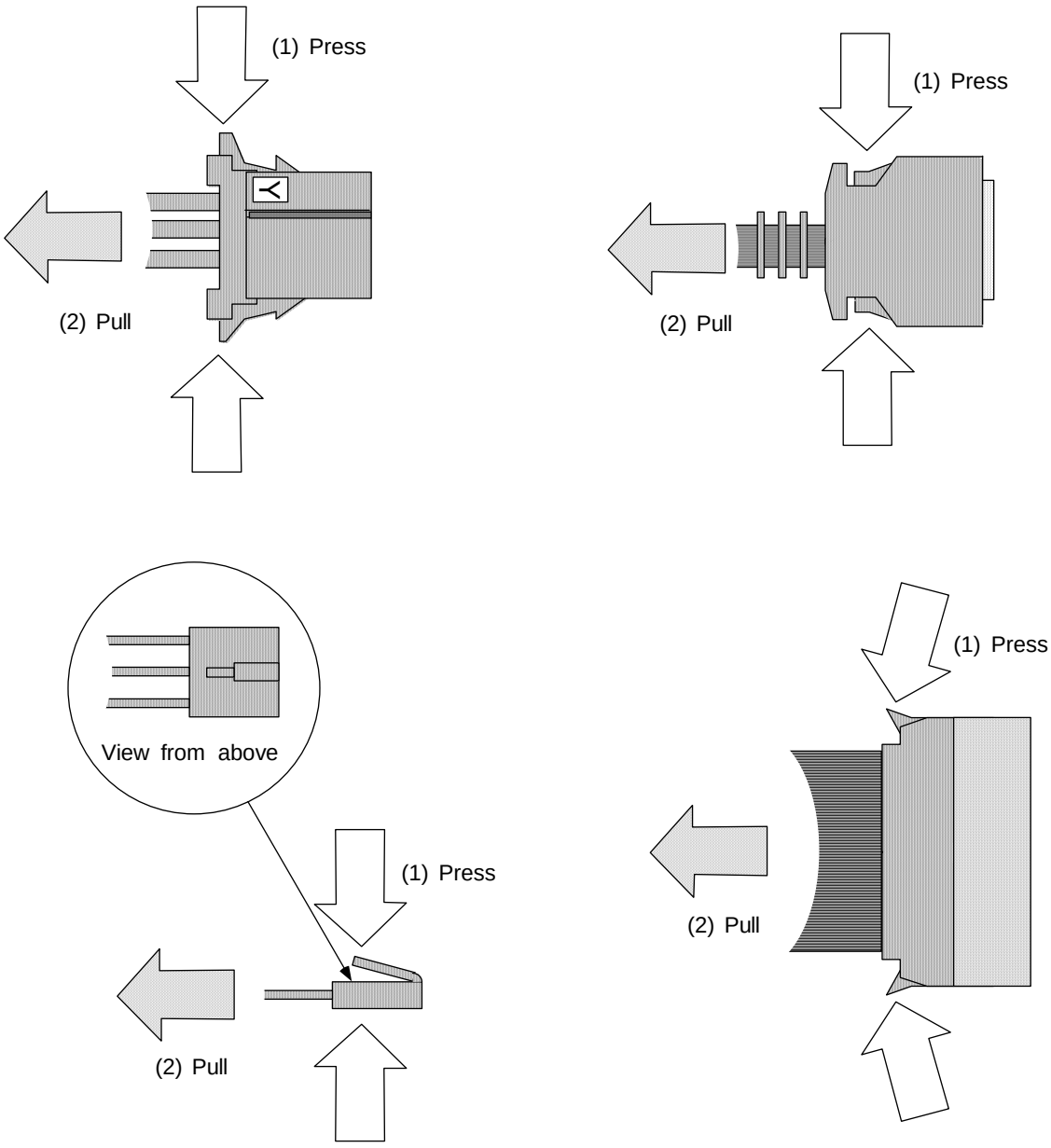
1) Do not store the card in locations having a high temperature or humidity.
2) Do not store the card in dusty locations.

6.3 Replacement Methods

6.3.1 Cable

If the cable is replaced without turning the power OFF, the normal unit or peripheral devices could be damaged, and risks could be imposed.
Disconnect each cable with the following procedures.

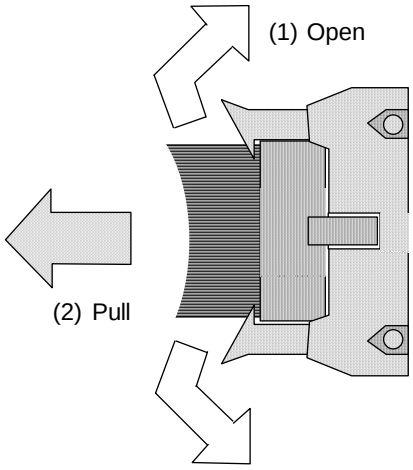
(a) For the following type of connector, press the tabs with a thumb and forefinger in the direction of the arrow, and pull the connector off.



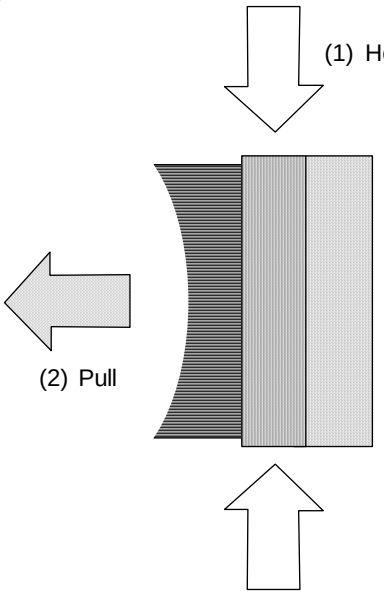
⚠ CAUTION

- **Do not connect or disconnect the connection cables between each unit while the power is ON.**
- **Do not connect the cable by pulling on the cable wire.**

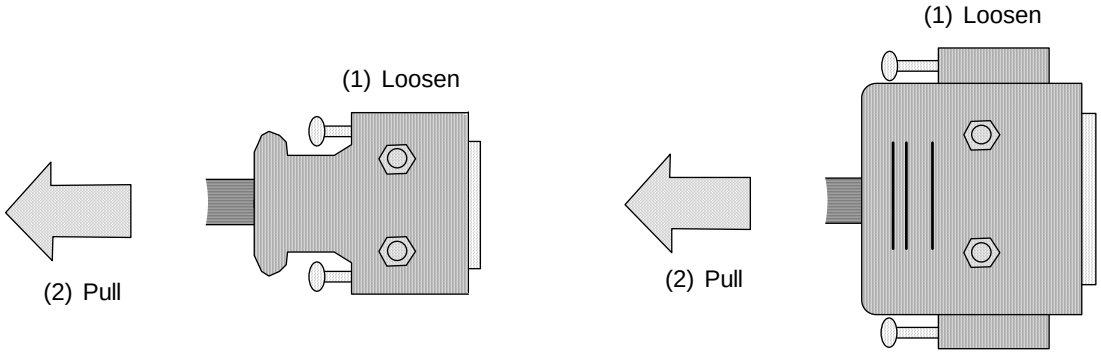
(b) For a flat cable type connector with latches, open the latches in the directions of the arrows, and pull the connector off.



(c) For a flat cable type connector without latches, hold the connector with a thumb and forefinger, and pull the connector off.



(d) For the screw fixed type connector, loosen the two fixing screws, and pull the connector off.



⚠ CAUTION

- **Do not connect or disconnect the connection cables between each unit while the power is ON.**
- **Do not connect the cable by pulling on the cable wire.**

6.3.2 Durable Parts

(1) Control unit battery

All data, such as the parameters and machining programs that need to be backed up when the power is turned OFF, are saved by a lithium battery installed in the control unit's battery holder.

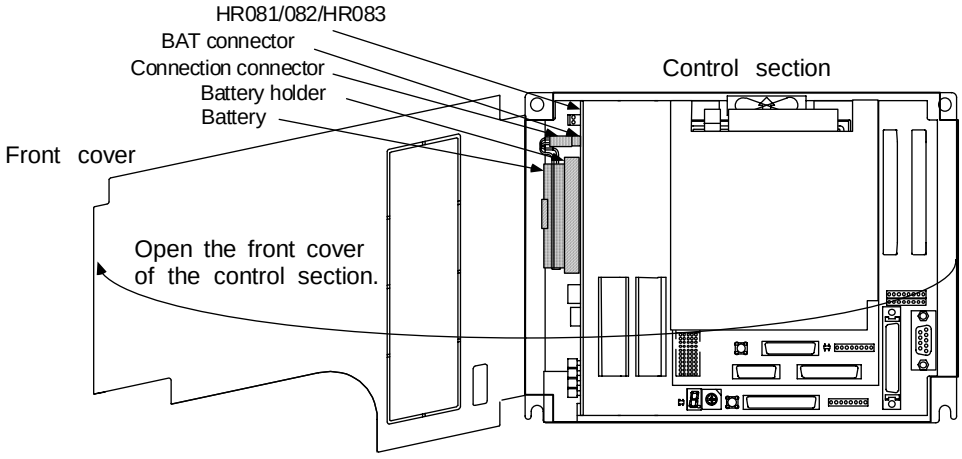
Battery	With ER6 connector (Toshiba with Mitsubishi specifications)
Initial battery voltage	3.6V
Voltage at which voltage drop is detected	2.8V (Battery voltage drop caution alarm screen display) 2.6V (Battery voltage drop warning alarm screen display + control section LED display)
Battery cumulative data holding time	45,000 hours (At room temperature. The life will be shorter if the temperature is high.)
Battery life	Approx. 5 years (from date of battery manufacture)
Discharge current	40μA or less

(Note) Replace the battery when the battery voltage drop caution alarm appears on the NC screen. The internal data could be damaged if the battery voltage drop warning alarm appears.

(a) Replacement procedures

Always replace the battery with the control section (machine) power turned OFF.
Complete the replacement within 30 minutes after turning the power OFF. (If the battery is not connected within 30 minutes, the data being backed up will be destroyed.)

1) Turn the control section (machine) power OFF.
2) Open the electric cabinet door, and confirm that the control unit LED, 7-segment display, etc., are all OFF.
3) If an extension unit is mounted, and a cable is connected to the front of the control section, disconnect the cable.
4) Open the front cover of the control section by pulling on the right side of the door (the side without fixing screws).
5) Remove the battery from the battery holder.
6) Pull the connector connected to the battery out from the BAT connector on the control section HR081/HR082/HR083 power supply PCB.
7) Replace the battery with the new one. Insert the connector connected to the new battery into the BAT connector on the control section HR081/HR082/HR083 PCB. Pay attention to the connector orientation, being careful not to insert backwards.
8) Fit the battery into the battery holder.
9) Close the front cover of the control section. At this time, confirm that the cover is closed by listening for the "click" sound when the latch catches.
10) Close the door of the electric cabinet.



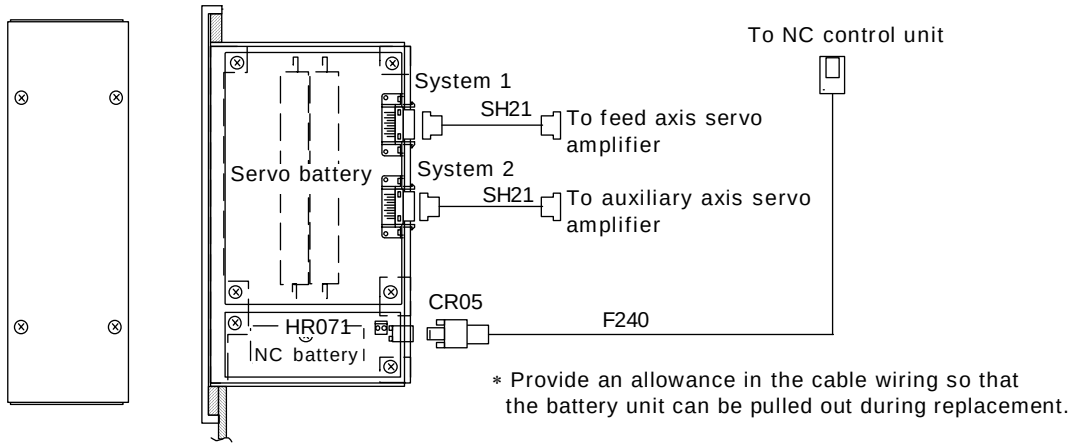
(b) Precautions for handling battery

- Always replace the battery with the same type.
- Do not disassemble the battery.
- Do not place the battery in flames or water.
- Do not pressurize and deform the battery.
- This is a primary battery so do not charge it.
- Dispose of the spent battery as industrial waste.

(2) External battery unit battery

When the external battery unit (FCU6-BT4D1) is used, the data, such as the parameters and machining programs, which needs to be backed up when the power is turned OFF, is held by the lithium battery mounted on the external battery unit. The external battery unit is provided with one AA battery (E6) for the control unit, and four AA batteries (ER6-B4D-01) for the servo amplifier.
The control unit battery is the same as the battery mounted on the control unit. The servo amplifier battery's characteristics, such as the holding time, differs according to the servo amplifier configuration, so refer to the servo amplifier manual.

Battery unit: FCU6-BT4D1



Unit type : FCU6-BT4D1
Replacement part type : For NC ER6 BKO-NC2157H01 (single battery replacement)
For servo ER6-B4D-01 BKO-NC2151H06 (PCB unit replacement)

(a) NC side battery replacement procedures

Always replace the external battery unit's battery with the control section (machine) power turned ON. (This is because there is no need to open the machine's electric cabinet door.)

1) Remove the four screws on the front cover, and pull the external battery unit out.
2) Remove the two screws on the cable relay PCB HR071, and remove the HR071A card.
3) Remove the battery (ER6) from the battery holder.
4) Pull the connector connected to the battery from the BAT connector on the cable relay PCB HR071.
5) Replace the battery with a new one. Insert the connector connected to the new battery to the BAT connector on the cable relay PCB HR071. Pay attention to the connector orientation, being careful not to insert backwards.
6) Fit the battery into the battery holder.
7) Fix the cable relay PCB HR071 onto the external battery unit, and insert and fix the external battery unit into the electric cabinet.

(b) Servo side battery replacement procedures

The absolute position data is backed up with the feed axis servo and auxiliary axis servo systems. Thus, if the battery voltage drop warning occurs in either the feed axis servo or auxiliary axis servo, the battery (ER6-B4D-01) must be replaced. The battery (ER6-B4D-01) is an assembled battery PCB type. The battery is replaced with the amplifier control power ON in the same manner as the control section replacement procedures.
If the battery voltage drop warning occurs, do not turn the amplifier control power OFF. (There is no need to turn ON the main circuit power or cancel the emergency stop.)

⚠CAUTION

- **If the battery voltage drop warning alarm occurs, the programs, tool data and parameters could be damaged. Thus, reload each data with the input/output device after replacing the battery.**
- **⚠ Do not short circuit, charge, overheat, incinerate or disassemble the battery.**
- **⚠ Dispose the spent battery according to local laws.**

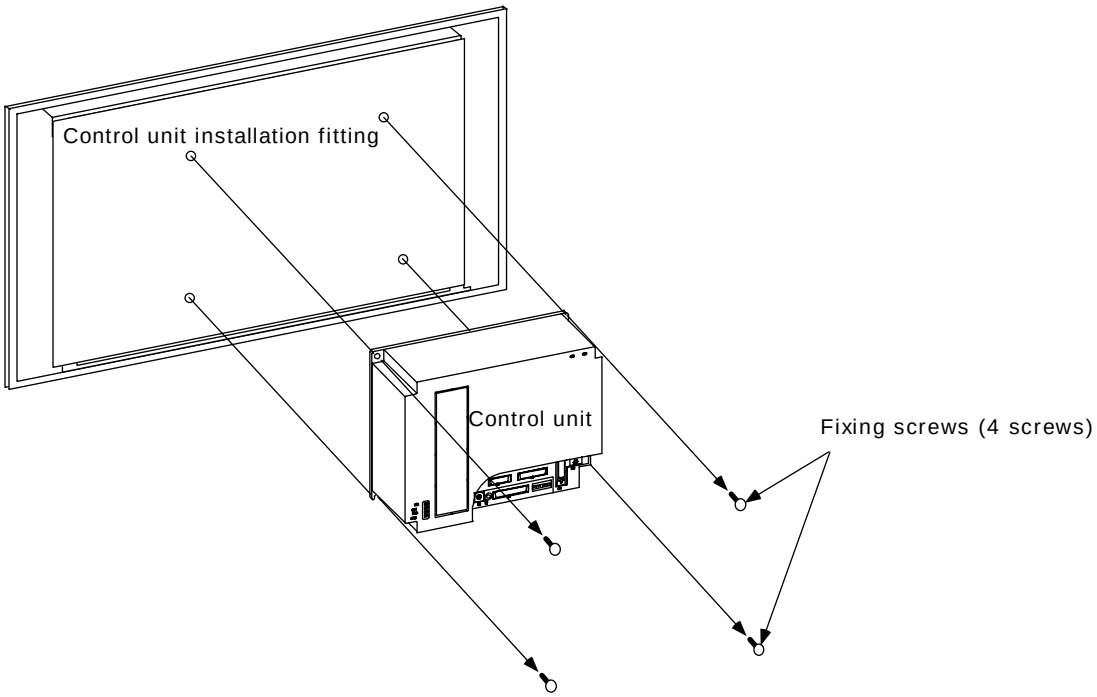
6.3.3 Unit

(1) Control unit

(a) Replacement procedures

Always replace the control unit with the machine power turned OFF.

1) Check that the machine power is turned OFF. (If the power is not OFF, turn it OFF.)
2) Open the electric cabinet door.
3) Disconnect all cables connected to the control unit.
4) Remove the screws fixing the control unit onto the control unit installation fitting, and remove the control unit from the control unit installation fitting. (Loosen the two lower fixing screws first, and then remove the two upper fixing screws while supporting the unit with a hand. Then lift the unit upward and off. The two lower fixing screws do not need to be removed.)
5) Replace with a new control unit, and fix the control unit onto control section installation fitting with the fixing screws.
6) Connect all cables connected to the control unit. (Connect the cables to the designated connectors.)
7) Check the cables and wires for connection leaks, faulty connections, etc., then close the electric cabinet door.



⚠ CAUTION

Incorrect connections may damage the devices, so connect the cables to the specified connectors.

Do not replace the control unit while the power is ON.

Do not connect or disconnect the connection cables between each unit while the power is ON.

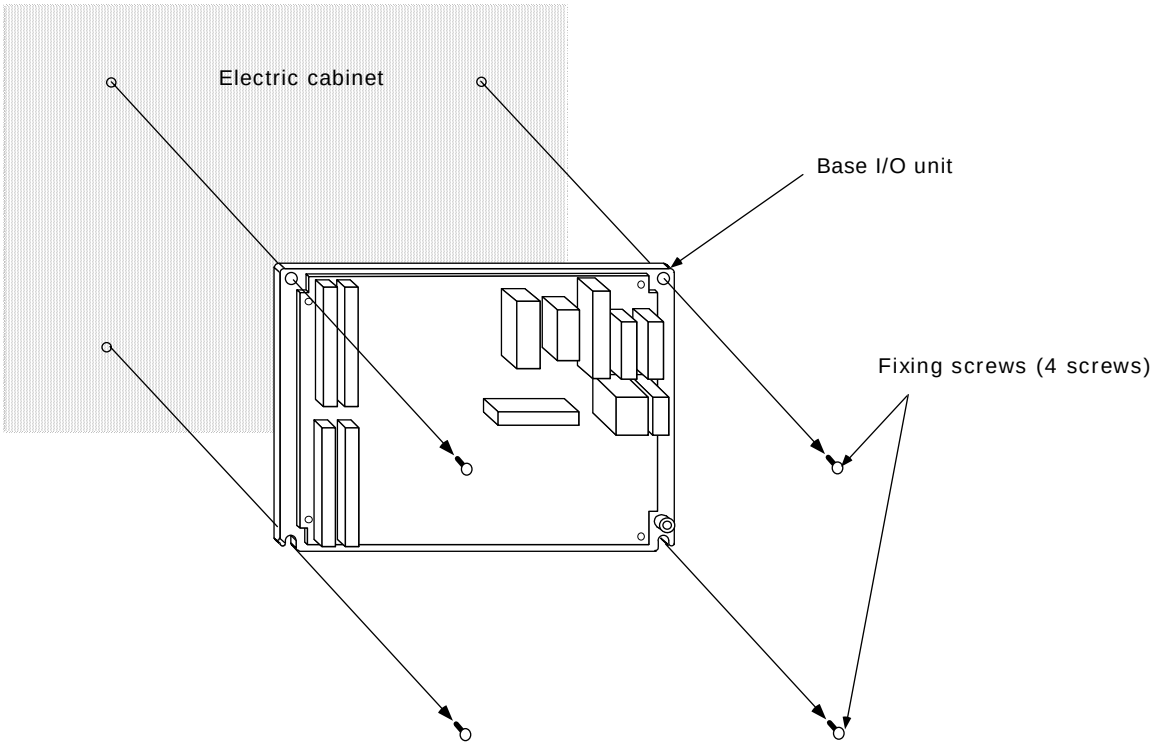
(2) Base I/O unit

The base I/O unit is generally installed on the electric cabinet side.

(a) Replacement procedures

Always replace the base I/O unit with the machine power turned OFF.

1) Check that the machine power is turned OFF. (If the power is not OFF, turn it OFF.)
2) Open the electric cabinet door.
3) Disconnect all cables connected to the base I/O unit.
4) Remove the screws fixing the base I/O unit to the electric cabinet, and remove the base I/O unit from the electric cabinet. (Loosen the two lower fixing screws first, and then remove the two upper fixing screws while supporting the unit with a hand. Then lift the unit upward and off. The two lower fixing screws do not need to be removed.)
5) Replace with a new base I/O unit, and fix the unit onto the electric cabinet with the fixing screws.
6) Connect all cables connected to the base I/O unit. (Connect the cables to the designated connectors.)
7) Check the cables and wires for connection leaks, faulty connections, etc., then close the electric cabinet door.



⚠ CAUTION

⚠ Incorrect connections may damage the devices, so connect the cables to the specified connectors.

⚠ Do not replace the base I/O unit while the power is ON.

⚠ Do not connect or disconnect the connection cables between each unit while the power is ON.

6.3.4 Control PCB

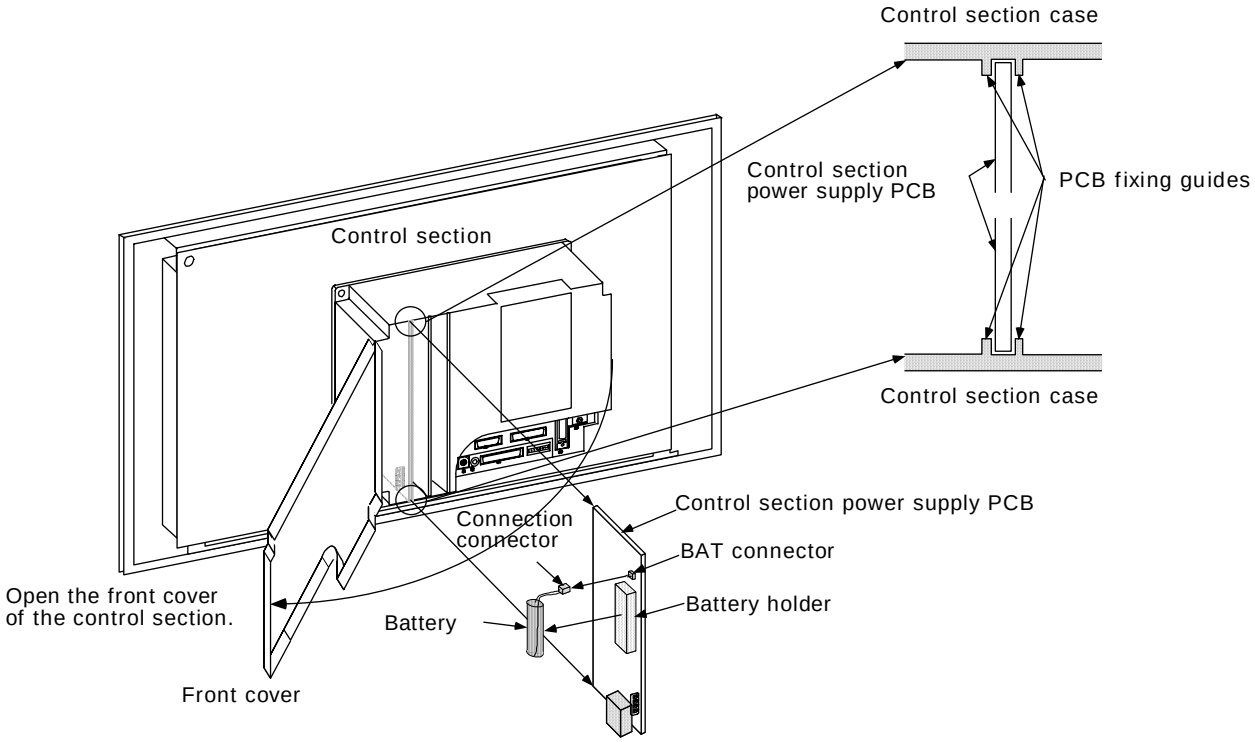
(1) Control section power supply PCB (HR081/HR082/HR083)

The control section power supply PCB is supplied with 24VDC from an external source. This PCB generates the DC voltage required for each control PCB in the control section.

(a) Replacement procedures

Always replace the control section power supply PCB with the machine power turned OFF.
A battery for backing up the memory is mounted on the control section power supply PCB, so as with the battery, replace the control section power supply PCB within 30 minutes.

1) Confirm that the machine power is OFF. (If the power is not OFF, turn it OFF.)
2) Open the electric cabinet door, and confirm that the control unit LED, 7-segment display, etc., are all OFF.
3) Disconnect all cables connected to the control section power supply PCB.
4) If an extension unit is mounted, and a cable is connected to the front of the control section, disconnect the cable.
5) Open the front cover of the control section by pulling on the right side of the door (the side without fixing screws).
6) While holding the front upper and lower sections of the control section power supply PCB with both hands, pull out the PCB from the control section.
7) Remove the battery from the control section power supply PCB that was removed from the control section.
8) Replace the battery with the new one. Insert the connector connected to the new battery into the BAT connector on the HR081/082 control section PCB.
9) Install the control section power supply PCB into the control section. (Align the control section power supply PCB with the PCB fixing guides on the inner side of the control section case, and then install.)
10) Connect all cables that were connected to the control section power supply PCB. (Connect all cables to the designated connectors.)
11) Close the front cover of the control section. At this time, confirm that the cover is closed by listening for the "click" sound when the latch catches.
12) Close the door of the electric cabinet.



⚠ CAUTION

⚠ Incorrect connections may damage the devices, so connect the cables to the specified connectors.
⚠ Do not replace the control section power supply PCB while the power is ON.
⚠ Do not connect or disconnect the connection cables between each unit while the power is ON.

(2) Expansion PCB

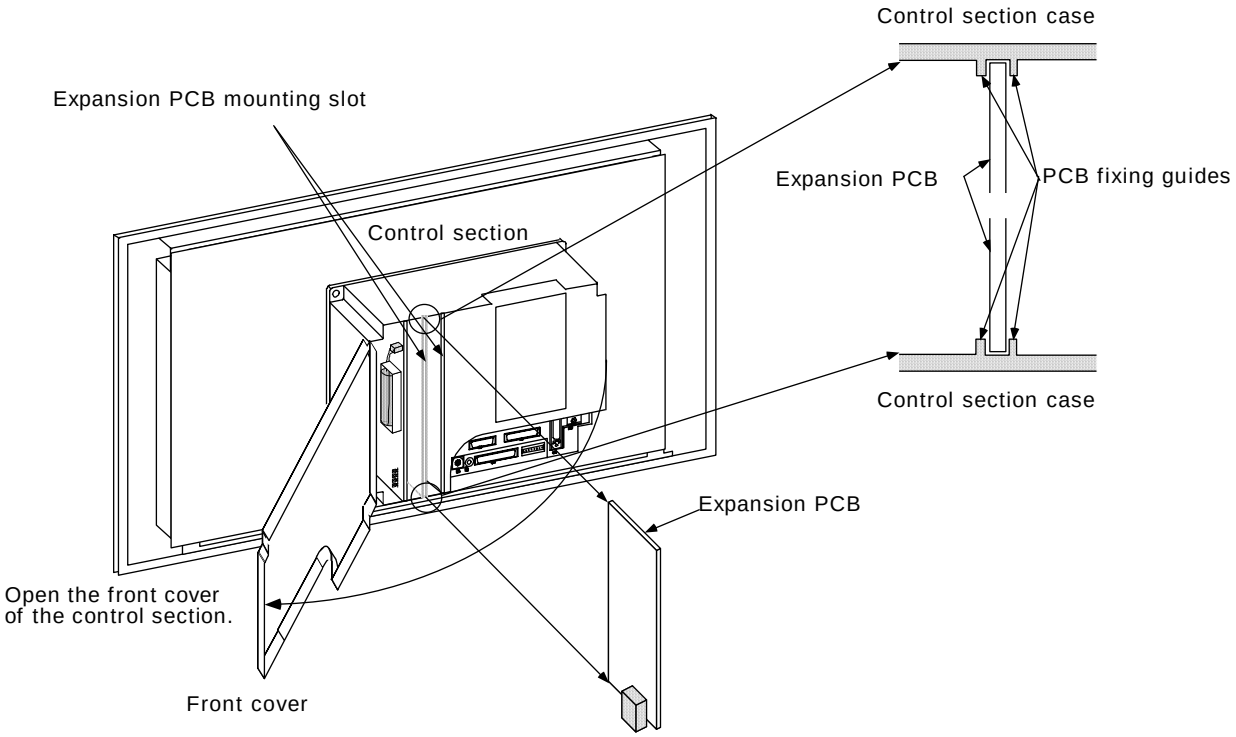
The expansion PCB is used to expand the system for adding a serial port, etc. (Refer to the "I Connection Manual, Section 2.2" in the Connection and Maintenance Manual BNP-B2183 for information related to expansion PCB types and functions.)

Up to two expansion PCBs can be added. The PCBs are mounted in the RT #1/#2 slots on the right side of the control section power supply PCB.

(a) Replacement procedures

Always replace the expansion PCB with the machine power turned OFF.

1) Confirm that the machine power is OFF. (If the power is not OFF, turn it OFF.)
2) Open the electric cabinet door, and confirm that the control unit LED, 7-segment display, etc., are all OFF.
3) Disconnect all cables connected to the control unit and expansion PCB.
4) Open the front cover of the control section as shown in the drawing below.
5) While holding the front upper and lower sections of the expansion PCB with both hands, pull out the PCB from the control section.
6) Replace with the new expansion PCB. If the PCB has setting locations, set to the same settings as the old PCB.
7) Install the expansion PCB into the slot in which the old PCB was mounted.
   (Align the expansion PCB with the PCB fixing guides on the inner side of the control section case, and then install.)
8) Close the front cover of the control section.
9) Connect all cables that were connected to the control unit and expansion PCB.
   (Connect all cables to the designated connectors.)
10) Close the door of the electric cabinet.



⚠ CAUTION

⚠ Incorrect connections may damage the devices, so connect the cables to the specified connectors.

⚠ Do not replace the expansion PCB while the power is ON.

⚠ Do not connect or disconnect the connection cables between each unit while the power is ON.

6.3.5 Memory Cassette

(1) HR4xx

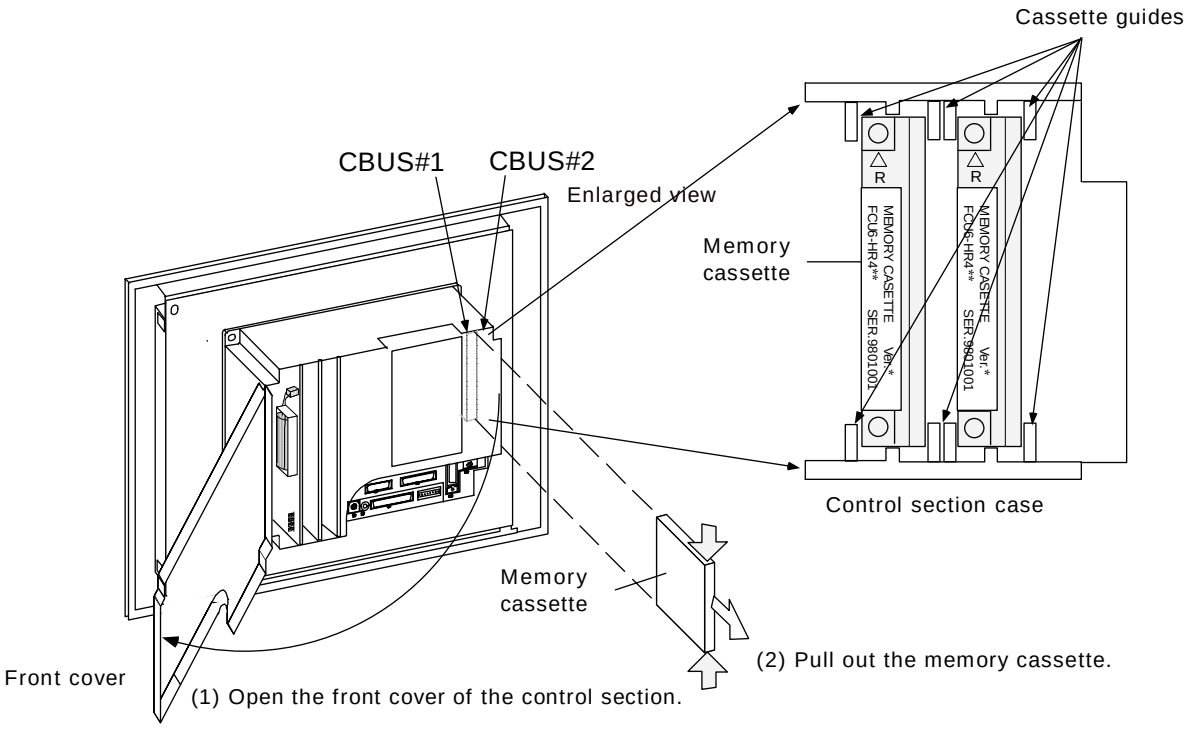
The memory cassette is a PCB used to store user PLC and machining programs, and is installed on the CBUS#2 connector of the control section.

(a) Replacement procedures

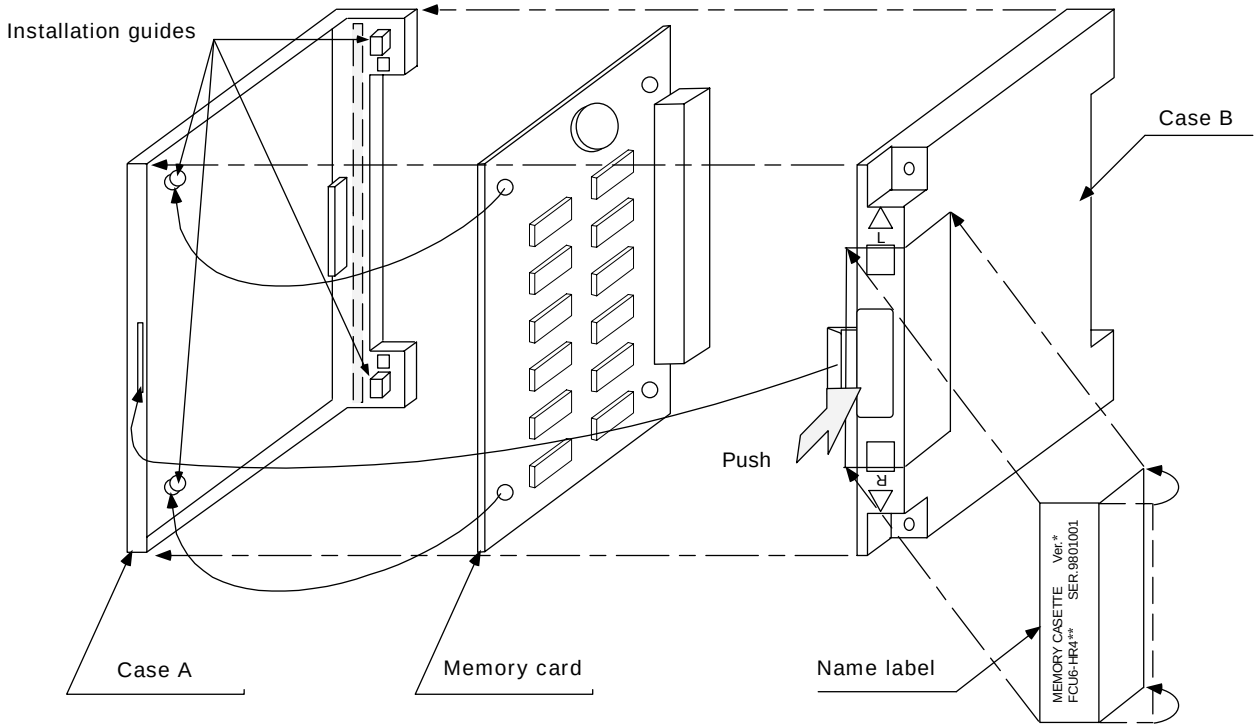
Always replace the memory cassette with the machine power turned OFF.
Before replacing RAM-type memory cassettes, always make a backup of the memory to be replaced using an external I/O device, etc. Then, re-input the details after replacing the memory cassette.

Confirm that the control unit power is OFF.

(1) Open the control unit cover from the right side.
(2) While holding the top and bottom of the memory cassette with a thumb and forefinger, pull out the memory cassette from the control section.
(3) Install the new memory cassette in the control section.
Install by carrying out steps (1) and (2) in reverse (2) → (1) order.
(Align the memory cassette with the cassette fixing guides on the NC control case.)



(2) Method for disassembling the cassette cover



1) Remove the name label holding case A and B.
2) Press the indented area on case B and separate the case into parts A and B.
3) Remove the memory card inside from the installation guides.

⚠ CAUTION

Incorrect connections may damage the devices, so connect the cables to the specified connectors.

Do not replace the memory cassette while the power is ON.

Do not connect or disconnect the connection cables between each unit while the power is ON.

Be careful that metal cutting chips, etc., do not come into contact with the connector contacts of the memory cassette.

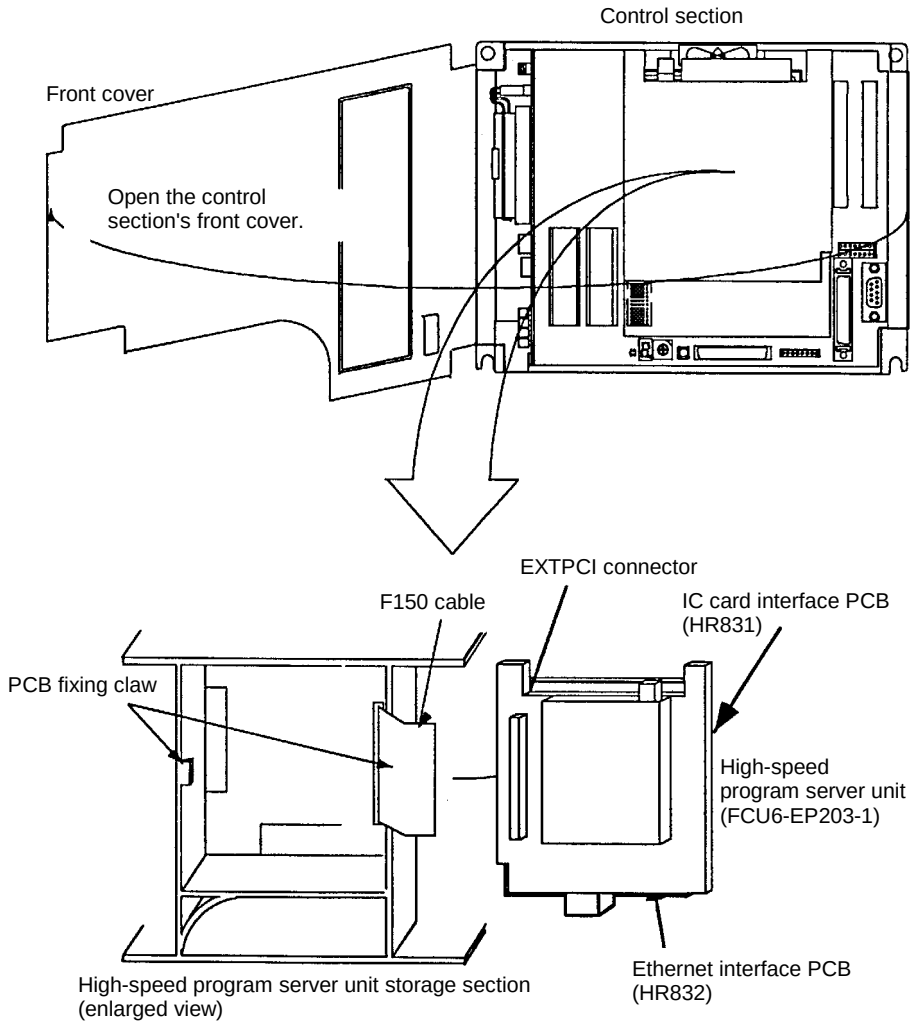
6.3.6 High-speed Program Server

(1) High-speed program server unit (FCU6-EP203-1)

The high-speed program server is configured of the HR831 and HR832 cards and an F150 cable. It is replaced as a unit.

(a) Replacement procedures

Turn the machine power OFF before replacing the high-speed program server unit.



⚠ CAUTION

⚠ Incorrect connections will cause device damage, so always connect the cable to the specified connector.

⚠ Do not replace the high-speed program server unit while the power is ON.

7. Troubleshooting

7.1 Confirmation of Trouble State

Confirm "when", "when doing what", and "what kind of" trouble occurred.

(1) When?

What time did the trouble occur?

(2) When doing what?

What was the NC operation mode?

- During automatic operation ..... Program No., sequence No. and program details when the trouble occurred.
- During manual operation ..... What was the manual operation mode?
  What was the operation procedure?
  What were the previous and next steps?
- What was the setting and display unit screen?
- Did the trouble occur during input/output operations?
- What was the machine side state?
- Did the trouble occur while replacing the tools?
- Did hunting occur in the control axis?

(3) What kind of trouble?

- What was displayed on the setting and display unit's Alarm Diagnosis screen?
  Display the Alarm Diagnosis screen, and check the alarm details.
- What was displayed for the machine sequence alarm?
- Is the CRT and LCD screen normal?

(4) How frequently?

- When did the trouble occur? What was the frequency? (Does it occur when other machines are operating?) If the trouble occurs infrequently or if it occurs during the operation of another machine, there may be an error in the power voltage or the trouble may be caused by noise, etc. Check whether the power voltage is normal (does it drop momentarily when other machines are operating?), and whether noise measures have been taken.
- Does the trouble occur during a specific mode?
- Does the trouble occur when the overhead crane is operating?
- What is the frequency in the same workpiece?
- Check whether the same trouble can be repeated during the same operation.
- Check whether the same trouble occurs when the conditions are changed.
  (Try changing the override, program details, and operation procedures, etc.)
- What is the ambient temperature?
  (Was there a sudden change in the temperature? Was the fan at the top of the control unit rotating?)
- Is there any contact defect or insulation defect in the cables?
  (Has any oil or cutting water splattered onto the cables?)

7.2 When in Trouble

If the system does not operate as planned or if there is any trouble in the operation, confirm the following points and then contact the Mitsubishi Service Center.

– Examples of trouble –

- Nothing appears in the NC screen, LED, etc., even when the power ON button is pressed.
- The power turns OFF suddenly.
- Nothing appears on the NC screen, or the screen is completely white.
- The operation keys do not function. The NC screen appears but the operation board key input does not function.
- Machining operation is not possible.

(1) Problems related to the power supply

The power does not turn ON.

Cause	Remedy
The power cord is disconnected or loose.	Check the cable between the NC unit and the external power supply, and between the external power supply and socket. Make sure that the cable is inserted securely. Check that there are no wires broken in the cable. If broken, replace the cable.
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. Repair it.
There is a problem in the power socket.	If the socket has a switch, turn the switch ON. Make sure that the specified power voltage and power frequency is being output from the socket.
The external power supply is faulty.	Check that the power can be turned ON with just the external power supply. **(Note)** Depending on the external power supply being used, the power may not turn ON in the no-load state, so install a slight load and check.
The ON/OFF cable is short-circuited.	Disconnect the ON/OFF cable and check that it is not short-circuited. If short-circuited, replace the cable.
The external power supply's input voltage is not as specified.	Check that the input voltage is within 200 to 230VAC +10 to –15%, 100 to 115VAC +10 to –15%, and 24VDC±5%.

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. Wire the cable between the NC unit and external power supply, and check that the external power supply output is normal.
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.

⚠ CAUTION

⚠ Do not apply voltages other than those indicated in this 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.

The power turns OFF.		
	Cause	**Remedy**
	There is a problem in the power socket.	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 HR081/HR082/HR083 power card PSEMG (red) LED is lit.		
	Cause	**Remedy**
	The emergency stop switch connected to the EMG connector is ON (A contact), or the EMG connector is disconnected.	Set the emergency stop switch to the release (B contact) state. Check the connection to the EMG connector.
The HR081/HR082/HR083 power card BATALM (red) LED is lit.		
	Cause	**Remedy**
	The LED lights when the voltage of the battery connected to the HR081/HR082/HR083 BAT connector has dropped to 2.6V ± 0.065V or less.	Refer to section 6.3.2 and replace the battery.

⚠ CAUTION

- **Do not apply voltages other than those indicated in this 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 CPU card's 7-segment display NCLD1.	Check that the rotary switch NCSYS is set to 0. If not set to 0, set to 0 and restart.
	E or F is displayed on the CPU card's 7-segment display NCLD1.	Contact the Mitsubishi Service Center.

(3) Problems related to remote I/O

The communication alarm LED RAL is lit.		
	Cause	**Remedy**
	The remote connection cable (SH41) of RIO System 1 or System 2 is not connected. The cable has broken wires or the contact is defective.	Check the connection of the remote connection cable F010 with the NC control section or the cable SH41 between the remote I/Os.
	The HR3×× card is defective.	Contact the Mitsubishi Service Center, and then replace the card.
The power supply system LED is not lit.		
	Cause	**Remedy**
	The 24IN LED is not lit. (The input power is not being supplied.)	Supply a +24V±5% voltage to the HR3OO card.
	The 5OUT LED is not lit. (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.

⚠ CAUTION

⚠ Do not apply voltages other than those indicated in this 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.

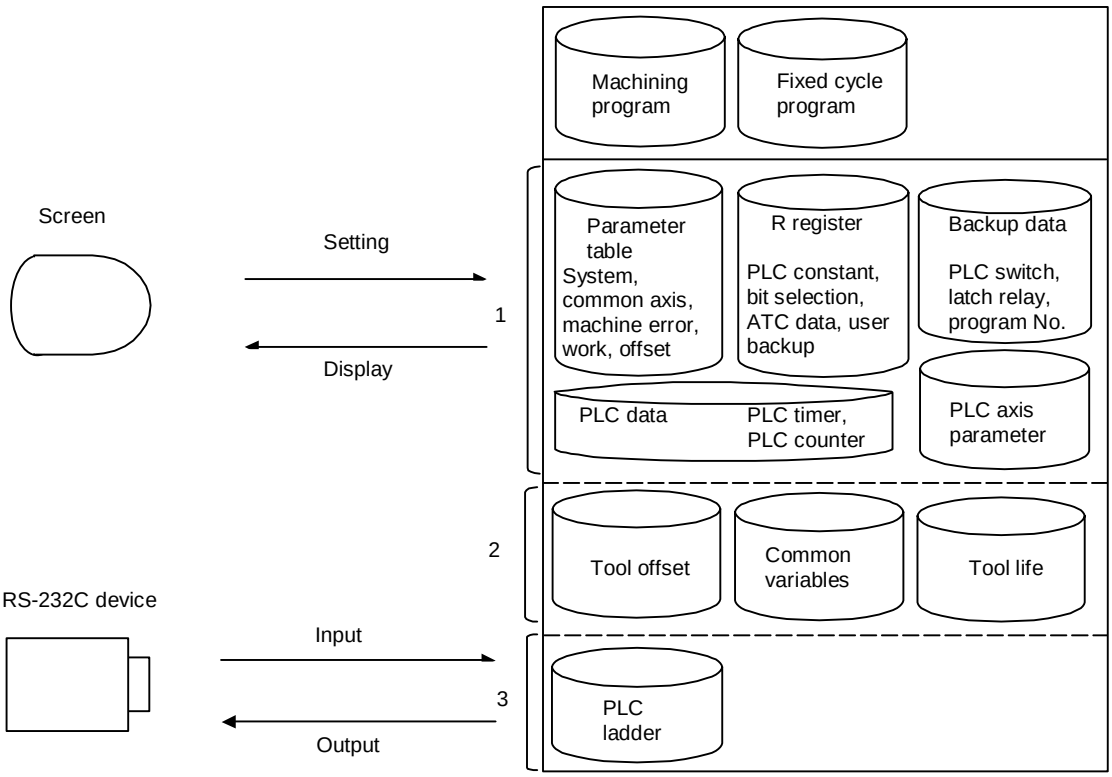
8. Maintenance Functions

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.

8.1 Data Input/Output Function

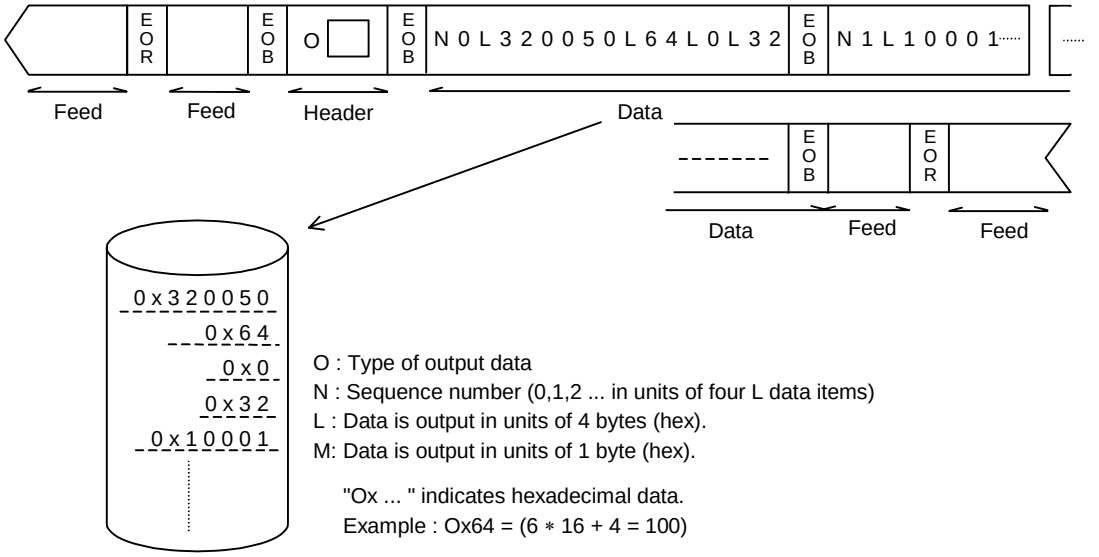
The data can be collectively input/output or collated in the data input/output screen. This function can be used to protect data and investigate failure causes at the customer premises. It can handle three types of data shown below. Hex data is converted and output in the ISO/EIA format. Individual data 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)
   Work offset data
   Backup data (PLC latch relay, PLC switch, program No.)
2. File data: The file format is required before data input.
   Tool offset data
   Common variables
   Tool life control data
3. PLC program data: PLC program area data
   Ladder (ROM: Ladder, message)

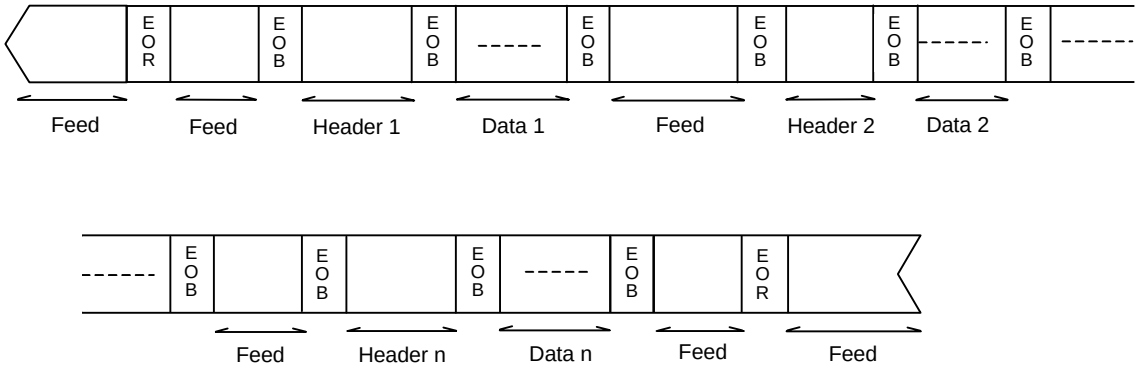


8.1.1 Data Format

The tape output format of single data is as follows:



The output format of continuous data (batch output data) is as follows:



(Note) The number of feeds is the value set with the input/output parameter #9114.

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 (R2800~R2899)	
106	PLC timer	
107	PLC counter	
108	Bit selection parameter (R2900~R2947)	
109	(Not used)	
110	Work offset	
111	R register (R2950~R3639)	
112	R register user backup (R1900~R2799)	
113	PLC latch relay L	
114	PLC switch	
115	Program No.	
116	Timer cumulative output	
117	Counter cumulative output	
118	PLC timer expansion	
119	Bit selection parameter expansion (R4400~R4449)	
130	Tool life management I (R3000~R3639)	
133	Position switch	
134	PLC timer cumulative current value	
135	PLC counter cumulative current value	
136	R register system backup (R1880~R1899)	
138	Tool life management II (R5480~R6279)	
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.200~249)

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 diameter offset (shape)	Tool offset (Z shape)	
203	Tool diameter offset (wear)	Tool offset (Z wear)	
204	—	Tool offset (Tool offset selection additional axis shape)	
205	Tool life	Tool offset (Tool offset selection additional axis wear)	
206	—	Nose R (shape)	
207	—	Nose R (wear)	
208	—	Nose point No. data	
209	Common variables for both systems (#500~)		
210	Common variable valid flags for both systems (#500~)		
230	Common variables independent for systems (#100~)		
231	Common variable valid flags independent for systems (#100~)		

PLC program (No.250~299)

No.	Data details	ALL output
250	Ladder	ALL3

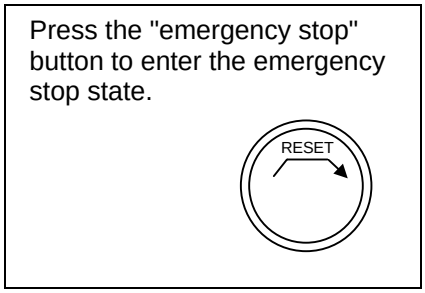
8.1.2 Data Output

(1) Explanation of function

When the user PLC is stopped, the various data on the data input/output output screen can be converted from HEX to ISO/EIA and output to the external RS-232C device.

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

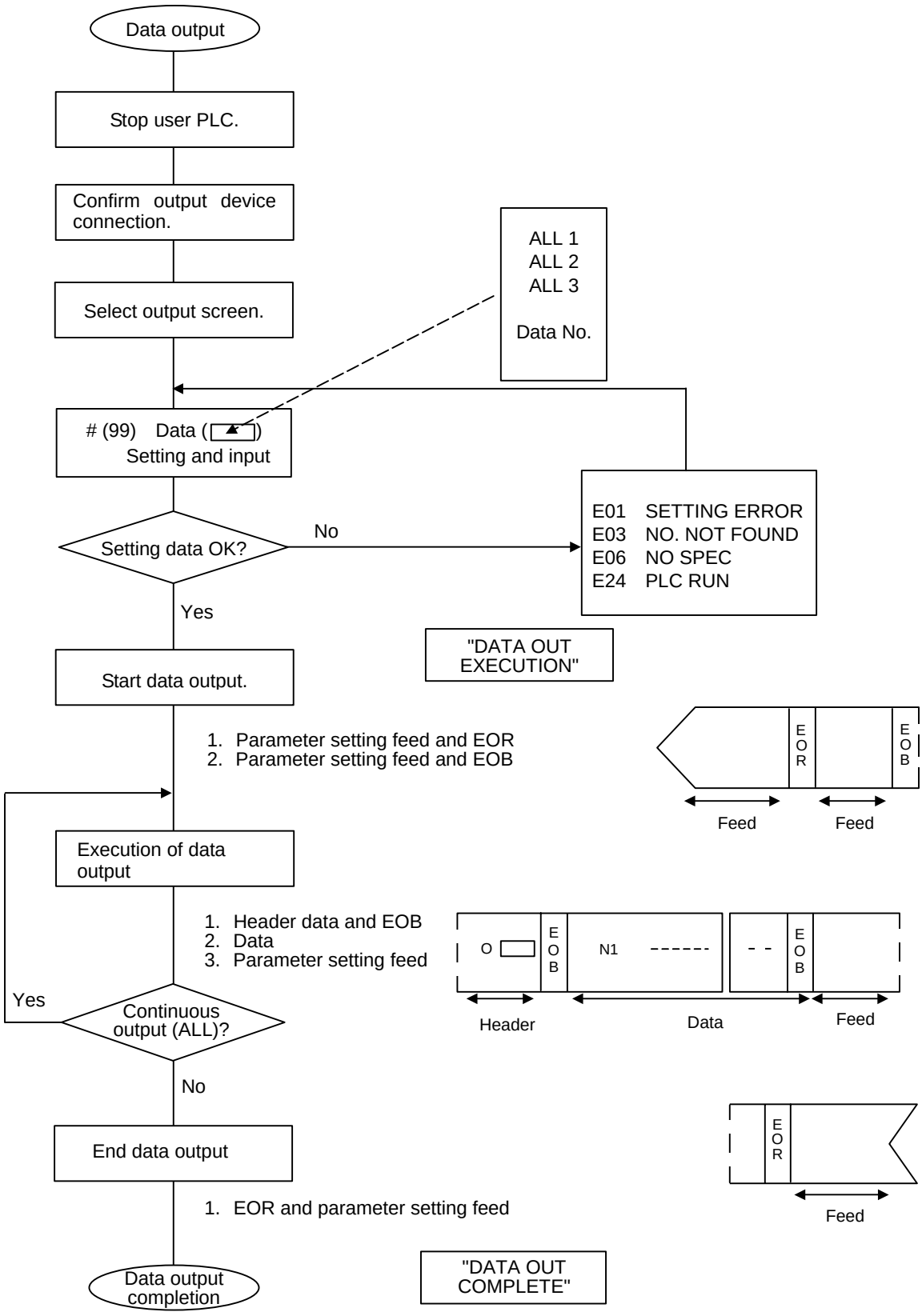
Setting area operation	Output data (No.)
# (99) DATA (ALL1)	Plain data 100~149
# (99) DATA (ALL2)	File data 200~230
# (99) DATA (ALL3)	PLC program 250~299

Single data output function

Setting area operation	Output data (No.)
# (99) DATA (☐)	Single data 100~149, 200~230, 250~299

(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.

(2) Data output operation procedure



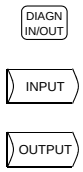
(Note) The number of feeds is the value set with the input/output parameter #9114.

(3) Example of data output operation

Confirm that the user PLC is stopped.

Connect the output device.

Call out the data output screen.



1) The data output screen will display.

Group output of data ex. ALL1: Plain data
(Parameter, R register, work 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] key.

<OUTPUT DATA>
N3L320050L64L0L32 ;

DATA OUT EXECUTION
(99) data (10 [1])

<OUTPUT DATA>
%

DATA OUT COMPLETE
(☐) DATA ( )

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 key.

<OUTPUT DATA>
N5L320000L10001LF0000BL78;

DATA OUT EXECUTION
(99) DATA (209)

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 DATA>
%

DATA OUT COMPLETE
(□) DATA ( )

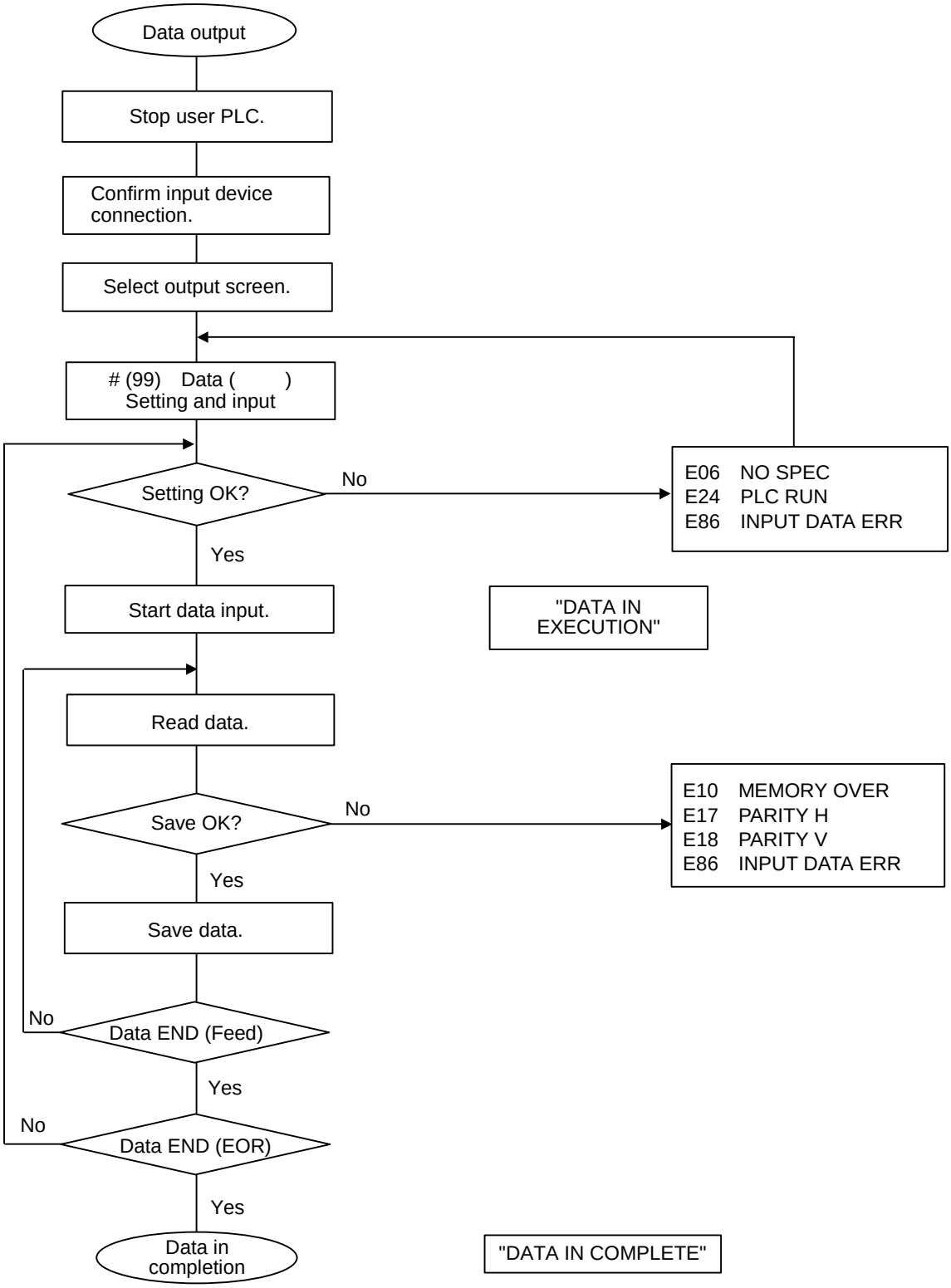
(Note 1) The setting of the data protection key will be ignored.
(Note 2) Data not found in the specifications will not be output.

8.1.3 Data Input and Compare

(1) Explanation of function
When the PLC 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 PLC, 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

IN/OUT

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>

(9[9]) DATA ( )

Press INPUT key.

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.
2) The input operation is completed when the end code % (EOR) is read in. Then the parameters will be written into the EEROM. Do not turn off the power while the message "EEROM WRITE IN PROCESS" is displayed.
   When writing to the EEROM is completed, the message "DATA IN COMPLETE" will display.

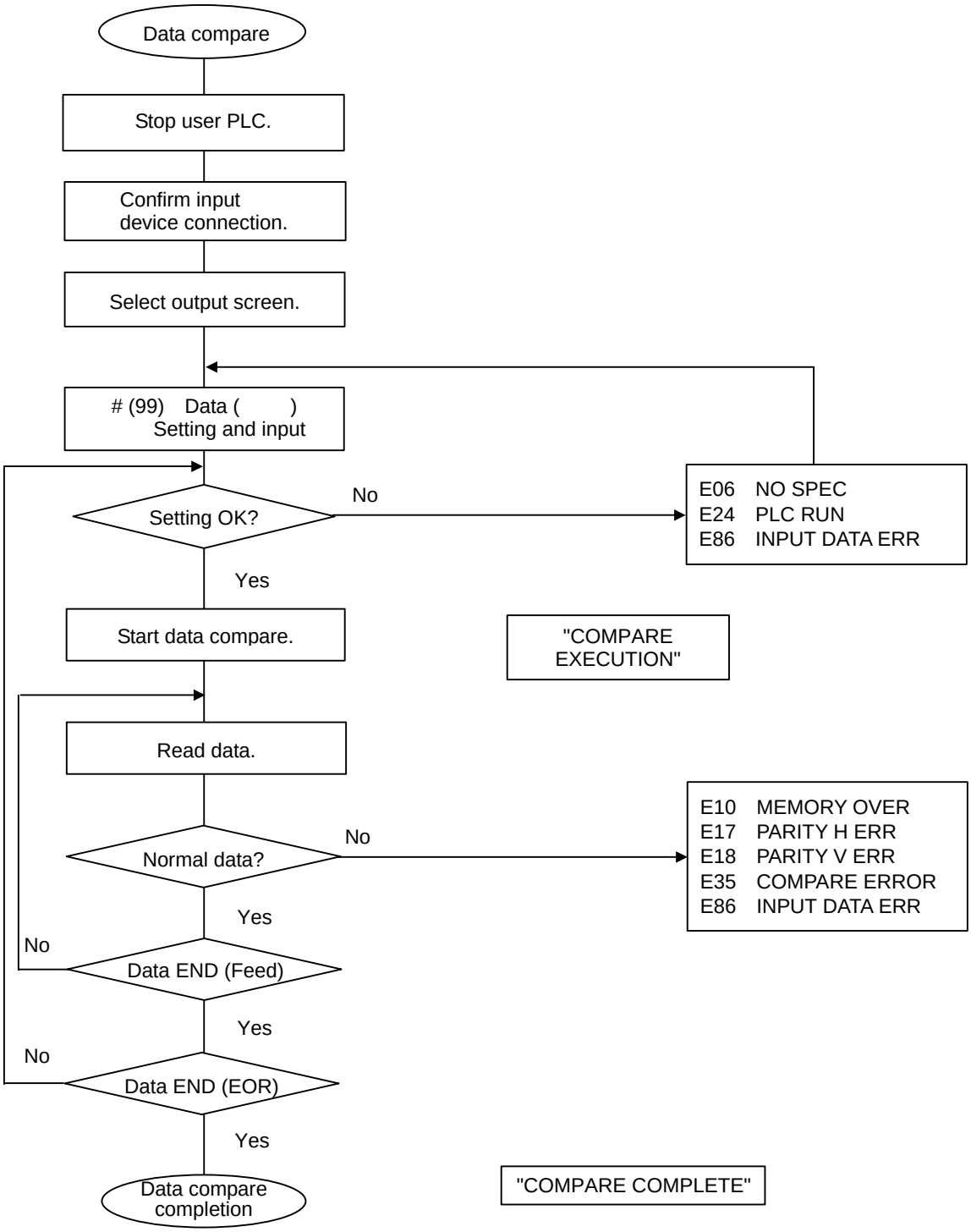
<INPUT DATA>
N2L0L0L10001L10000;
<COMP. DATA>

DATA IN EXECUTION
(9[9]) 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 ( )

<INPUT DATA>

<COMP. DATA>

(99) DATA ( )

Press INPUT key.

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 COM-PLETE" will display.
3) If a compare error occurs, the message "E35 COMPARE ERROR" will display, and the operation will be stopped.

<INPUT DATA>
N2L10001LF0000L0L0;
<COMP. DATA>
N2L10001LF0000L0L0;
COMPARE EXECUTION
(99) DATA (101)

<INPUT DATA>
%
<COMP. DATA>
%
COMPARE COMPLETE
(□) DATA ( )

8.1.4 Parameter Backup

(1) Explanation of function

The parameter backup function is used to back up and restore the NC parameters, etc. (including options).

(2) Backup targets

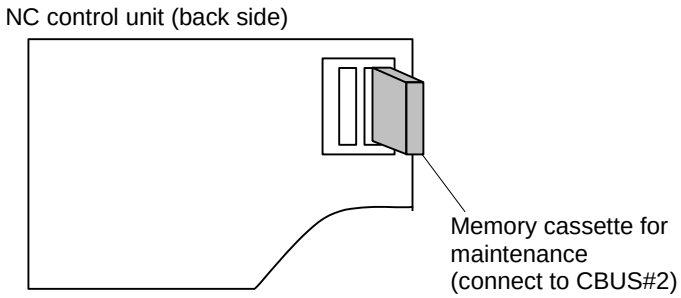
The backed up data is the maintenance data output equivalent to ALL1 (table data1) and ALL3 (PLC program). Refer to section "8.1.1 Data Format" for details.

(3) Hardware configuration

The memory cassette for maintenance is used to back up and restore the data.

Model	Compatible memory cassette for maintenance
M64AS	HR410/HR450
M64S	HR411/HR451
M65S/M66S	HR415/HR455
Others	No function

The memory cassette for maintenance is mounted in CBUS#2 on the NC control unit.



(4) Operation methods

The data is backed up and restored using the BACKUP screen. Refer to section "4.5 BACKUP Screen" in "I. OPERATION SECTION" for details on operation.

(Note 1) With the M60S, the NC system cannot start if the maintenance memory cassette is not mounted.

8.2 Data Sampling

The NC internal data (speed output from NC to amplifier, or feedback data from amplifier, etc.) can be sampled and output serially.
(Setup parameter "#1224 aux08/bit0" must be set to "1" to use this function.)

8.2.1 Specifications

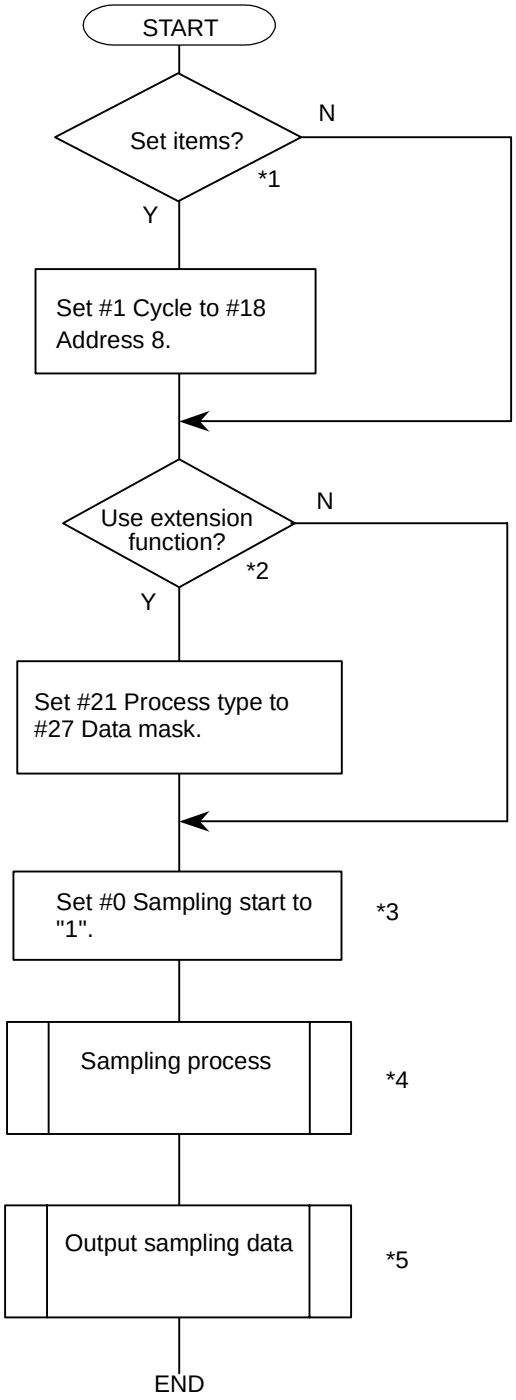
- Sampling cycle : 3.5ms ∗ scale
- Number of sampled axes : 1 to 18 axes (Servo axes ∗ 14 + spindle ∗ 4)
- Number of sampled points : 1 to 8 points
- Number of sampled data items : With no memory cassette ... Max. 3,072
  When using HR412/HR452 (512KB) ... Max. 131,072
  When using HR422/HR462 (1MB) ... Max. 262,144
  When using HR432/HR472 (2MB) ... Max. 524,288

(Note 1) The memory cassette is mounted in CBUS#1.
(Note 2) The maximum number of sampled data items is the number of all sampled points. When the data to be sampled increases, the number of sampled points will drop per data item.

[Restrictions]

(1) When using this function, do not use the input/output functions (PLC, APLC, MELDASNET or anshin-net) or the onboard.
(2) If the machining program is longer than 1280m or if APLC is added, the memory cassette cannot be designated.
(3) If a memory cassette is mounted in CBUS#1 for a purpose other than data sampling (to run APLC on the SRAM, etc.), the expansion memory cannot be used to sample the data.
(4) The data set with this function is not output to the parameters.
(5) When the power is turned ON, "#0 Sampling start" will be set to "0" (sampling stop).
(The data set in #1 to #27 will be held.)

8.2.2 Operation Procedures



*1
Go to N if previously set values are to be used.

*2
Go to Y when using ring buffer, or to set special ending conditions.

*3
During manual start
... Process is executed immediately
Other than the above
... The trigger standby state is entered, and the execution is started after the start trigger is detected.

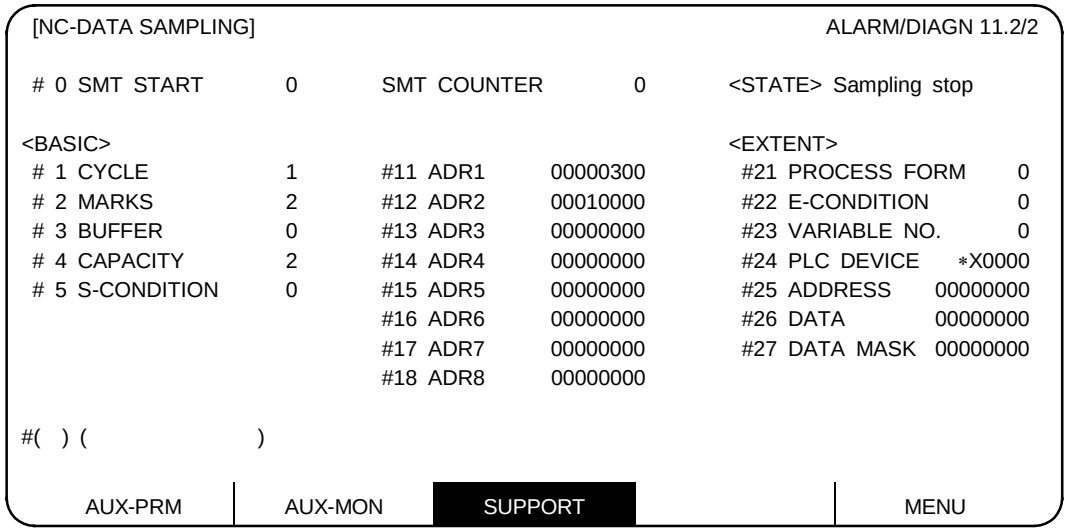
*4
Refer to section (11) in "8.2.3 Setting and Display Items" for details on the ending conditions.

*5
Refer to section "8.2.4 Data Output Procedures".

Refer to section "8.2.3 Setting and Display Items" for details on each setting item.

8.2.3 Setting and Display Items

The various data related to sampling of data is set and displayed on the NC-DATA SAMPLING screen.



(Note 1) #21 to #27 are extension functions and normally do not need to be set.
(Note 2) The data cannot be set when "#1224 aux08/bit0" is set to "0".

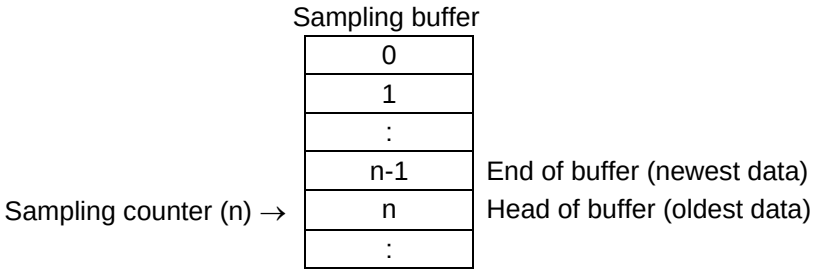
(1) STATE display

The current status is displayed.

Display	Status
"sampling"	Sampling is being executed.
"sampling stop"	Sampling is not being executed, or the sampling process has completed.
"TRIGGER WAIT"	When #5 is not set to "0" (manual start), this indicates that state from when #0 was set to "1" to when the sampling start trigger was detected and sampling was started.

(2) SMT COUNTER (sampling counter)

The position of the sampling buffer during the sampling process is displayed.
When the sampling buffer is valid, the head of the buffer can be seen with the value of the counter at the end of sampling.



(3) #0 SMT START (setting range: 0, 1, 9)

0:	Sampling stop
1:	Sampling start
2:	Sampling forced end

Sampling will start when "1" is set. This will be set to "0" when sampling is completed.
The operation when "1" is set will differ according to the #5 setting.

#5 setting	Operation
0	Sampling starts immediately.
Other than 0	System waits for trigger, and sampling starts when trigger is detected.

If "9" is set during the sampling process, the sampling will stop immediately, and the value will be set to 0.

(4) #1 CYCLE (setting range: 1 to 255)

Set the sampling cycle.
Cycle = 1.7ms ∗ 2 ∗ scale
Scale = (setting value + 1) /2 [integer value]
(Example) When setting value is 1 or 2: 3.5ms cycle
When setting value is 3 or 4: 7.1ms cycle

(5) #2 MARKS (setting range: 1 to 8)

Set the number of sampling points.
(Note) The sampling buffer will be divided by the set number of points. The number of samples per point will decrease when the number of sampling points is increased.

(6) #3 BUFFER (setting range: 0 to 1)

Set the range of the buffer to be used.

0:	Internal memory (DRAM)
1:	Memory cassette (capacity is automatically judged)

"1" cannot be selected in the following cases:
- When the memory cassette is not mounted
- When the machining program capacity is higher than 1280m

(7) #4 CAPACITY (setting range: 0 to n)

Set the capacity of the buffer used for sampling.
Buffer capacity = (setting value + 1) ∗ 1024 items
The setting value's upper limit n differs according to the buffer range being used.

Buffer range	Maximum number of sampling data items	Setting value upper limit n
Built-in memory	3,072	2
HR412/HR452	131,072	127
HR422/HR462	262,144	255
HR432/HR472	524,288	511

(8) #5 S-CONDITION (setting range: 0 to 4)

Set the conditions for starting sampling.

0:	Manual start (Sampling starts when "1" is set in #0)
1:	Variable No. start (Sampling starts when the variable set in #23 is 0 or a value other than null)
2:	PLC device start (Sampling starts at the rising edge of the signal set in #24)
3:	Address condition, true (Sampling starts when the #25 to #27 conditions are true)
4:	Address condition, false (Sampling starts when the #25 to #27 conditions are false)

(Note 1) Always set #25 to #27 when "3" or "4" is set. Sampling might not be started if these are not set. If the same value as #22 is set (if #5 is set to "3" and #22 to "3" or #5 to "4" and #22 to "4"), sampling will not take place.

(Note 2) When "1" is set, the common variable uses the floating decimal point method, so if the calculation results are used, an error will occur, and the value may not be recognized as 0. When using the multi-system, the trigger will turn ON if the conditions are satisfied in even one system.

(Note 3) The variable data settings are validated immediately only from the program. These will not be validated immediately with settings from the screen or DDB.

(9) #11 to #18 ADR1 to 8

Set the sampling address.

These items can be set with the following three methods. The applied method is automatically judged.

- Designate prepared data with index No.
- Set actual address
- Search for head address of symbol and set automatically

A different designation method can be set for each address.

(Note) If addresses exceeding the setting value are set, the setting will be ignored.

(a) Index No. method

The index No. is fixed regardless of the axis configuration.

The low-order six digits of the address are explained below.

(If the high-order two digits are not set, the setting will be interpreted as 00.)

Name	Standard setting value
Address #1	00nn00 (nn is the axis No. of the basic axis k in system 1)
Address #2	010000
Address #3	0 (no setting)
Address #4	0 (no setting)
Address #5	0 (no setting)
Address #6	0 (no setting)
Address #7	0 (no setting)
Address #8	0 (no setting)

Setting range (index No.)

Servo axis	1st axis	2nd axis	...	14th axis
Feedback position	000100	000200	...	000E00
Command position	000101	000201	...	000E01

Spindle	1st axis	2nd axis	...	4th axis
Feedback position	010000	020000	...	040000
Command position	010001	020001	...	040001

High-order 4 digits: Axis index
High-order 2 digits : Spindle index (01 to 04)
Low-order 2 digits : Servo axis index (01 to 0E)
Low-order 2 digits: Sampling target index
00: Feedback position
01: Command position

* Examples of setting for each purpose are shown below.

- Synchronized tapping setting : 3rd servo axis FB (000300) - 1st spindle axis FB (010000)
- High-accuracy setting (roundness) : 1st servo axis FB (000100) - 2nd servo axis FB (000200)
- Spindle synchronization setting : 1st spindle axis FB (010000) - 2nd spindle axis FB (020000)

(b) Actual address method
If the actual address is known beforehand, it can be set by directly inputting it.
However, if an illegal address (highest-order bit 0) is input, a setting error will occur.
If an address that does not exist is set, the system could fail.
(Note) Even if the data does not match near the 4th byte, "E02 DATA OVER" will occur.

(c) Head address automatic setting method
When the symbol is set in the setting area, the head address of that symbol will be searched for, and will be automatically set as the sampling address.
Normally, the data to be actually sampled is the actual address obtained by adding the data offset (relative address from address determined with symbol) to the address determined with this method.
When setting a symbol containing "_", substitute "=".
(Note 1) The symbols are limited to NC internal table symbols having within 16 digits.
(Note 2) The symbols are searched for as all lowercase symbols.
(Note 3) If there is no symbol, "E01 SETTING ERROR" will occur, and if the symbol is not found within the 4 bytes, "E02 DATA OVER" will occur.

(10) #21 PROCESS FORM (Setting range: 0 to 2)

Set the sampling process type.

0:	One-shot (Stops only when buffer is full.)
1:	Repeat valid (After the process ends (the buffer is full), the "TRIGGER WAIT" state will be entered again.
2:	Sampling buffer valid

(Note) If a value other than "0" is set, #22 must also be set.
If not set, the sampling process will not end until it is forcibly ended.

(11) #22 E-CONDITION (Setting range: 0 to 4)

Set the conditions for ending the sampling process.

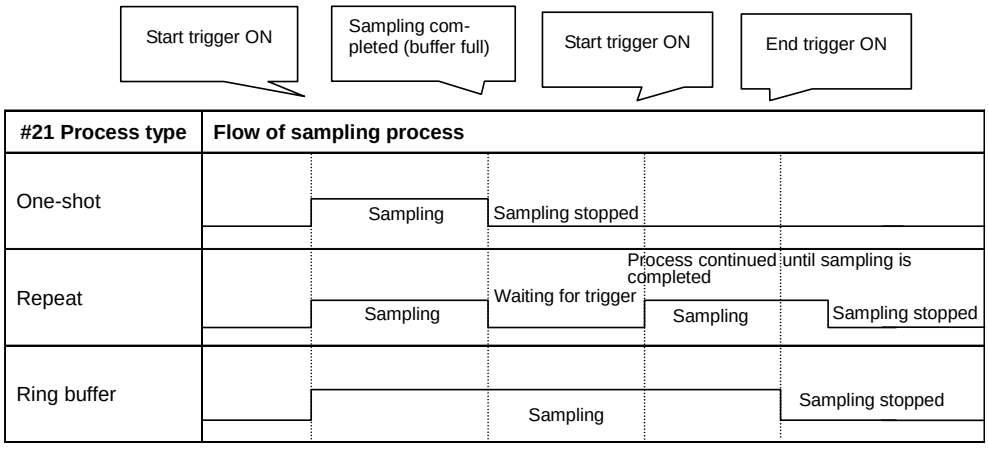
0:	Sampling completed (buffer full)
1:	Variable No. (Sampling ends when the variable set in #23 is 0 or a value other than null)
2:	PLC device (Sampling ends at the rising edge of the signal set in #24)
3:	Address condition, true (Sampling ends when the #25 to #27 conditions are true)
4:	Address condition, false (Sampling ends when the #25 to #27 conditions are false)

(Note 1) Always set #25 to #27 when "3" or "4" is set. Sampling may not end if these are not set.
If the same value as #5 is set (if #5 is set to "3" and #22 to "3" or #5 to "4" and #22 to "4"), sampling will not take place.

(Note 2) When "1" is set, the common variable uses the floating decimal point method, so if the calculation results are used, an error will occur, and the value may not be recognized as 0.
When using a multi-system, the end trigger will turn ON only when the conditions are satisfied in all systems.

(Note 3) The variable data settings are validated immediately only from the program. These will not be validated immediately with settings from the screen or DDB.

(Note 4) Even fit the ending conditions are satisfied, the subsequent flow will differ according to the value set in #21. Refer to the following diagram for details. (This also applies when ending manually.)



(12) #23 VARIABLE NO. (Setting range: 0 to 999)

Set the variable No. that causes the start/end trigger.

0:	System variable (#1299)
Other than 0:	Designated common variable (#100 ~, #500 ~)

(Note 1) If both #5 and #22 are set to a value other than "1" (variable No.), the value will be ignored even if set.

(Note 2) If a number that does not exist is set, "E02 DATA OVER" will occur.

(13) #24 PLC DEVICE

(Setting range: For PLC4B : ~ (∗) X04BF, ~ (∗) Y053F, ~ (∗) U0017F, ~ (∗) W01FF
For GPPW : ~ (∗) X063F, ~ (∗) Y073F)

Set the PLC device that functions as the start/end trigger. If a "∗" is added at the head, the device will be a b contact.

0:	Data sampling trigger signal (Y2FC)
Other than 0:	Designated device

(Note 1) If both #5 and #22 are set to a value other than "2" (PLC device), the value will be ignored even if set.

(Note 2) If a device that does not exist is input, "E01 SETTING ERROR" will occur, and if the device No. is not within the setting range, "E02 DATA OVER" will occur.

(14) #25 ADDRESS (For simple setup: 0)

Set the address targeted for applying #27 data mask.
However, if an illegal address (highest-order bit 0) is input, a setting error will occur.
If an address that does not exist is set, the system could fail.

(Note) If both #5 and #22 are set to a value other than "3" or "4" (address conditions, true/false), the value will be ignored even if set. This item can be set with the same methods as #11 to #18.

(15) #26 DATA (For simple setup: 0)

Set the data for judging the results of applying #27 data mask on #25 address.

(Note) If both #5 and #22 are set to a value other than "3" or "4" (address conditions, true/false), the value will be ignored even if set.

(16) #27 DATA MASK (For simple setup: 0)

Set the data mask to be applied on #25 address.

(Note) If both #5 and #22 are set to a value other than "3" or "4" (address conditions, true/false), the value will be ignored even if set.

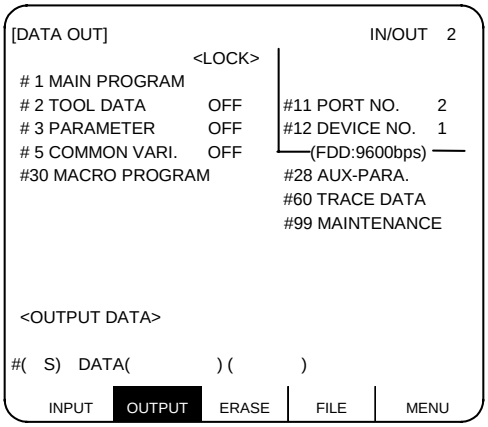
(17) Example of conditions for ending with address

[Setting details]
#22 E-CONDITION ... "3"
#25 ADDRESS ... "12345678"
#26 DATA ... "00000100"
#27 DATA MASK ... "0000FFFF"

[Meaning of setting details]
The point where results of applying a mask with 0000FFFF onto the data in address 12345678 are 00000100 will function as the sampling end trigger.

8.2.4 Data Output Procedures

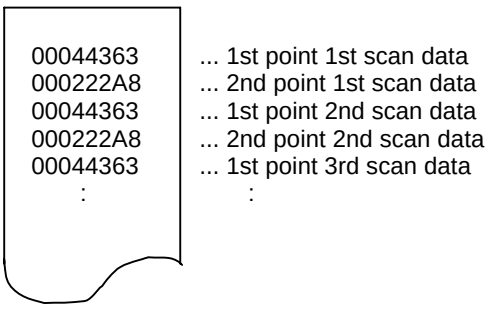
The sampled data can be output by setting "S" in "# ( )" on the DATA OUT screen.



(Note 1) The data will not be output if sampling has not been executed.
(Note 2) The data will not be output if the sampling output valid parameter is not set to valid. (Sampling output is validated by setting setup parameter "#1224 aux08/bit0" to 1.)

The output data is the sampling buffer dumped with a long data length (4 bytes).
The header, etc., are not added, and the sampling data is output immediately.

(Example) When number of sampling points is "2" ("2" MARKS" = 2)



IV. APPENDIXES

Appendix 1 List of Function Codes

Function code ISO	Control unit recognition	Subject to parity V count	Screen display	Setting and display unit key-in	Stored in memory	Punch-out output ISO	Internal NC system function
0~9	Yes	Counted	Displayed	Key-in	Stored	0~9	Numerical data
A~Z	Yes	Counted	Displayed	Key-in	Stored	A~Z	Addresses
+	Yes	Counted	Displayed	Key-in	Stored	+	Sign, variable operator (+)
–	Yes	Counted	Displayed	Key-in	Stored	–	Sign, variable operator (–)
.	Yes	Counted	Displayed	Key-in	Stored	.	Decimal point
,	Yes	Counted	Displayed	Key-in	Stored	,	
/	Yes	Counted	Displayed	Key-in	Stored	/	Block delete (optional block skip) Variable operator (÷)
%	Yes	Counted	Displayed (%)	No key-in (automatical-ly inserted)	Stored	%	End of record (tape storage end) Rewind start & stop during tape search
LF/NL	Yes	Counted	Displayed (;)	Key-in ;/EOB	Stored	LF	End of block
(	Yes	Counted	Displayed	Key-in	Stored	(	Control out (comment start)
)	Yes	Counted	Displayed	Key-in	Stored	)	Control in (comment end)
:	Yes	Counted	Displayed	No key-in	Stored	:	Program number address (instead of O.)
#	Yes	Counted	Displayed	Key-in	Stored	#	Variable number
*	Yes	Counted	Displayed	Key-in	Stored	*	Variable operator (´)
=	Yes	Counted	Displayed	Key-in	Stored	=	Variable definition
[	Yes	Counted	Displayed	Key-in	Stored	[	Variable operator
]	Yes	Counted	Displayed	Key-in	Stored	]	Variable operator
BS	No	Counted	Not displayed	No key-in	Stored		
HT	No	Counted	Not displayed	No key-in	Stored		
SP	No		Not displayed	Key-in	Stored	SP (T-V automatic adjustment)	
CR	No		Not displayed	No key-in	Stored		
DEL	No	Not counted	Not displayed	No key-in	Not stored		
NULL	No	Not counted	Not displayed	No key-in	Not stored		
Any other	No	Counted	**(Note 2)**	No key-in	Stored		

(Note 1) Codes not listed in the above table are stored on tape, but an error will result during operation if they are not comments.

(Note 2) This denotes characters (including blanks) which are stored inside and which correspond to the command codes. "@" is not displayed.

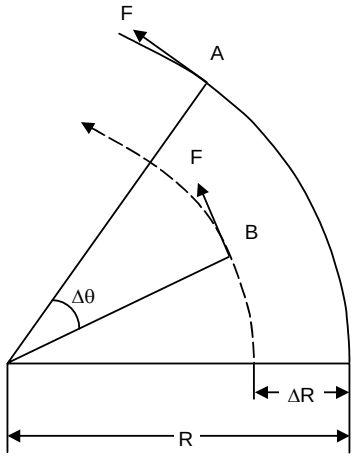
Appendix 2 Table of Command Value Ranges

	Linear axis		Rotary axis
	Input unit (mm)	**Input unit (inch)**	**Input unit (°)**
Minimum input setting unit	0.001/0.0001/0.00001	0.0001/0.00001	0.001/0.0001/0.00001
Maximum stroke (value for machine coordinate system)	±99999.999mm ±9999.9999mm ±999.99999mm	±9999.9999inch ±999.99999inch	±99999.999° ±9999.9999° ±999.99999°
Maximum programmable dimension	±99999.999mm ±9999.9999mm ±999.99999mm	±9999.9999inch ±999.99999inch	±99999.999° ±9999.9999° ±999.99999°
Rapid traverse rate (including dry run)	1 to 1000000mm/min 1 to 100000mm/min 1 to 10000mm/min	1 to 39370inch/min 1 to 3937inch/min	1 to 1000000°/min 1 to 100000°/min 1 to 10000°/min
M system Cutting feed rate (including dry run)	0.01 to 1000000mm/min 0.001 to 100000mm/min –	0.001 to 100000inch/min 0.0001 to 10000inch/min	0.01 to 1000000°/min 0.001 to 100000°/min –
L system Cutting feed rate (including dry run)	0.001 to 1000000mm/min 0.0001 to 100000mm/min 0.00001 to 10000mm/min	0.0001 to 39370.0787inch/min 0.00001 to 3937.00787inch/min	0.001 to 1000000°/min 0.0001 to 100000°/min 0.00001 to 10000°/min
M system Synchronous feed	0.001 to 999.999mm/rev 0.0001 to 99.9999mm/rev –	0.0001 to 999.9999inch/rev 0.00001 to 99.99999inch/rev	0.01 to 999.99°/rev 0.001 to 99.999°/rev –
L system Synchronous feed	0.0001 to 999.9999mm/rev 0.00001 to 99.99999mm/rev 0.000001 to 9.999999mm/rev	0.000001 to 99.999999inch/rev 0.0000001 to 9.9999999inch/rev	0.0001 to 999.9999°/rev 0.00001 to 99.99999°/rev 0.000001 to 9.999999°/rev
2nd to 4th reference point offset (value with machine coordinate system)	±99999.999mm ±9999.9999mm ±999.99999mm	±9999.9999inch ±999.99999inch	±99999.999° ±9999.9999° ±999.99999°
Tool offset amount (tool length)	±999.999mm ±99.9999mm ±9.99999mm	±99.9999inch ±9.99999inch	
Tool offset amount (wear)	±9999.999mm ±999.9999mm ±99.99999mm	±9.9999inch ±0.99999inch	
Incremental feed amount	0.001mm/pulse 0.0001mm/pulse 0.00001mm/pulse	0.0001inch/pulse 0.00001inch/pulse	0.001°/pulse 0.0001°/pulse 0.00001°/pulse
Handle feed amount	0.001mm/pulse 0.0001mm/pulse 0.00001mm/pulse	0.0001inch/pulse 0.00001inch/pulse	0.001°/pulse 0.0001°/pulse 0.00001°/pulse
Soft limit range (value with machine coordinate system)	–99999.999mm to +99999.999mm –9999.9999mm to +9999.9999mm –999.99999mm to +999.99999mm	–9999.9999inch to +9999.9999inch –999.99999inch to +999.99999inch	1 to 359.999° 1 to 359.9999° 1 to 359.99999°
Dwell time	0 to 99999.999s	0 to 99999.999s	
Backlash compensation amount	0 to ±9999pulse	0 to ±9999pulse	0 to ±9999pulse
Pitch error compensation amount	0 to ±9999pulse	0 to ±9999pulse	0 to ±9999pulse
M system Thread lead (F)	0.001 to 999.999mm/rev 0.0001 to 99.9999mm/rev –	0.0001 to 99.9999inch/rev 0.00001 to 9.99999inch/rev	
M system Thread lead (precision E)	0.00001 to 999.99999mm/rev 0.000001 to 99.999999mm/rev –	0.000001 to 39.370078inch/rev 0.000001 to 3.937007inch/rev	
L system Thread lead (F)	0.0001 to 999.9999mm/rev 0.00001 to 99.99999mm/rev 0.000001 to 9.999999mm/rev	0.000001 to 99.999999inch/rev 0.0000001 to 9.9999999inch/rev	
L system Thread lead (precision E)	0.00001 to 999.99999mm/rev 0.000001 to 99.999999mm/rev 0.0000001 to 99.9999999mm/rev	0.000010 to 9.9999999inch/rev 0.0000010 to 0.99999999inch/rev	

(Note 1) The second line in the table applies when the least setting increment is 0.001, 0.0001 from the first line. The third line applies when the least setting increment is 0.001, 0.0001, 0.00001 from the first line.

Appendix 3 Circular Cutting Radius Error

When circular cutting is performed, an error is caused between the command coordinate and the tracking coordinate due to the tracking delay in the smoothing circuit and servo system, and the workpiece ends up with a radius smaller than the commanded value. The method for obtaining this error (radius error) is shown below.



A : Command coordinate
B : Tracking coordinate
R : Command radius (mm)
ΔR : Radius error (mm)
Δθ : Angle error (rad)
F : Cutting feed rate (m/min)

The radius error ΔR and angle error Δθ are calculated from the following formula.

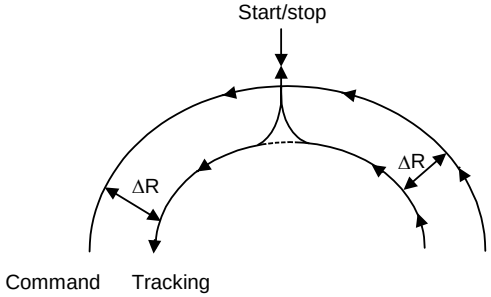
Exponential acceleration/ deceleration	$\Delta R = \frac{1}{R} \cdot (\frac{1}{2} Ts^2 + \frac{1}{2} Tp^2) \cdot (\frac{F \times 10^3}{60})^2$ (mm)
Linear acceleration/ deceleration	$\Delta R = \frac{1}{R} \cdot (\frac{1}{24} Ts^2 + \frac{1}{2} Tp^2) \cdot (\frac{F \times 10^3}{60})^2$ (mm)

$$\Delta\theta = \tan^{-1}(Ts \cdot \frac{F}{R}) + \tan^{-1}(Tp \cdot \frac{F}{R}) \text{ (rad)}$$

TS: Time constant (s) of specified smoothing circuit
TP: Position loop time constant

(Note 1) When the ΔR radius error applying with circular cutting does not come within the allowable value, proceed to reduce the cutting feed rate F, set Ts to a lower value or review the program.

(Note 2) In the steady state, ΔR is constant. However, it is not constant with command start and stop transitions. Under command start and stop conditions, therefore, the tracking coordinate should be as shown in the figure below.



Appendix 4 Registering/Editing the Fixed Cycle Program

The subprogram for the fixed cycle can be input, output and edited.

⚠ CAUTION

⃠ Do not change the fixed cycle program without prior approval from the machine maker.

4.1 Fixed Cycle Operation Parameters

To input/output or edit the data of each fixed cycle subprogram, use the IN/OUT and EDIT screens in the same way as when creating usual user-created work programs. In this case, the parameters must have been set. Set "1" in parameter "#1166 fixpro" on the BASE SPEC. PARAM screen. If this parameter is valid, the IN/OUT and EDIT screens are usable only for operating a fixed cycle control subprogram. During this period, PROGRAM FILE displays only fixed cycle programs. Thus, after fixed cycle program operation, return parameter to "0".

(Note) Parameter fixpro will be set to 0 when the power is turned OFF.

4.2 Inputting the Fixed Cycle Program

Input the fixed cycle program from the DATA INPUT screen. Before input, check that fixed cycle operation parameter "#1166 fixpro" is valid.
The operating procedure is the same as a user machining program. It is desirable that the data can be input consecutively.
After the data is registered, check proper registration on the PROGRAM FILE and EDIT screen.

4.3 Outputting the Fixed Cycle Program

Output the fixed cycle program on the DATA OUTPUT screen. Before output, check that fixed cycle operation parameter "#1166 fixpro" is valid.
The operating procedure is the same as a user machining program. The registered fixed cycle programs can be output one by one or at a time.
If the programs have been output at a time, they can be input consecutively by one operation.
After the data is output, be sure to collate it with the data on tape. At this time, make sure that there are no data output mistakes (punch mistakes, etc.).

4.4 Erasing the Fixed Cycle Program

Erase the fixed cycle program on the PROGRAM ERASE screen. Before erasing, check that fixed cycle operation parameter "#1166 fixpro" is valid.
The operating procedure is the same as a user machining program, except set 4 in # ( ) to designate the fixed C program.

4.5 Standard Fixed Cycle Subprogram (For L system)

G37 (O370)	Automatic tool length measurement

```
G31 Z #5 F #3 ;
1F [ROUND [ ABS [#2 – [ ##10 * #11 – #12 ] ] ] GT #8 ] GOTO 1 ;
1F [ROUND [ ##10 * #11 – #12 ] EQ #4 ] GOTO 1 ;
##9 = #10 – #12/#11 – #2/#11 + ##9 ;
#3003 = #1 ;
N2 ;
M99 ;
N1 # 3901 = 126 ;
```

G74 (O740)	End face cutoff cycle

```
G. 1 ;
1F [ ABS [ #2 ] GT 0 ] GOTO 10 ;
#14 = 1 ;
N10 #13 = #3 ;
IF [ #15 NE 0 ] GOTO 11 ;
#13 = #3 – #5 ;
N11 #16 = 0 ;
D0 1 ;
#10 = 0 ;
#11 = #4 ;
D0 2 ;
#10 = #10 + #4 ;
IF [ ABS [ #10 ] GE [ABS [ #1 ] ] ] GOTO 1 ;
G01 X #11 ;
G00 X #6 ;
#11 = #4 – #6 ;
END 2 ;
N1 G01 X#1 – #10 + #11 ;
IF [ #15 EQ 0 ] GOTO 20 ;
IF [ #16 EQ 0 ] GOTO 21 ;
N20 G00 Y#5 ;
N21 #16 = 1 ;
G00X – #1 ;
IF [ #14 ] GOTO 3 ;
#12 = #12 + #3 ;
IF [ ABS [ #12 ] LT [ABS [ #2 ] ] ] GOTO 2 ;
#14 = 1 ;
#13 = #2 – #12 + #13 ;
N2 G00 Y #13 ;
#13 = #3 – #5 ;
END 1 ;
N3 G00 Y – #2 – #5 ;
M99 ;
```

G75 (O750)	Longitudinal cutting cycle

```
G. 1 ;
1F [ ABS [ #1 ] GT 0 ] GOTO 10 ;
#14 = 1 ;
N10 #13 = #4 ;
IF [ #15 NE 0 ] GOTO 11 ;
#13 = #4 – #5 ;
N11 #16 = 0 ;
D0 1 ;
#10 = 0 ;
#11 = #3 ;
D0 2 ;
#10 = #10 + #3 ;
IF [ ABS [ #10 ] GE [ABS [ #2 ] ] ] GOTO 1 ;
G01 Y #11 ;
G00 Y #6 ;
#11 = #3 – #6 ;
END 2 ;
N1 G01 Y#2 – #10 + #11 ;
IF [ #15 EQ 0 ] GOTO 20 ;
IF [ #16 EQ 0 ] GOTO 21 ;
N20 G00 X#5 ;
N21 #16 = 1 ;
G00Y – #2 ;
IF [ #14 ] GOTO 3 ;
#12 = #12 + #4 ;
IF [ ABS [ #12 ] LT [ABS [ #1 ] ] ] GOTO 2 ;
#14 = 1 ;
#13 = #1 – #12 + #13 ;
N2 G00 X #13 ;
#13 = #4 – #5 ;
END 1 ;
N3 G00 X – #1 – #5 ;
M99 ;
```

G75.1 (O751)	Groove cutting cycle

```
G. 1 ;
#3003 = #8 OR 1 ;
G0 X #1 ;
G1 Y #2 ;
G0 Y – #2 ;
X #5 ;
1F [ #3 EQ 0 ] GOTO 1 ;
G1 X – #3 Y #4 ;
N1 G1 Y#6 ;
X – #7 ;
G0Y – #2 ;
X – #5 ;
1F [ #3 EQ 0 ] GOTO 2 ;
G1 X #3 Y #4 ;
N2 G1 Y#6 ;
X #7 ;
#3003 = #8 ;
G0Y – #2 ;
M99 ;
```

G76 (O760)	Compound thread cutting cycle

```
G. 1 ;
#12 = 1 ;
#13 = #9 ;
1F [ ABS [ #13 ] GE [ ABS [ #8 ] ] ] GOTO 1 ;
#16 = 1 ;
#13 = #8 ;
N1 #11 = #13 ;
1F [ ABS [ #11 ] LT [ ABS [ #4 – #5 ] ] ] GOTO 2 ;
#11 = #4 – #5 ;
#14 = 1 ;
N2 #17 = #11 ;
#18 = ROUND [ [ #4 – #11 – #5 ] * #7 ] ;
IF [ [ #18 XOR #1 ] GE 0 ] GOTO 10 ;
#18 = – #18 ;
N10 #19 = #18 ;
#10 = ROUND [ [ #11 + #5 ] * #7 ] ;
IF [ [ #10 XOR #1 ] GE 0 ] GOTO 20 ;
#10 = – #10 ;
N20 G00 X#10 ;
#20 = #10
D0 1 ;
#15 = ROUND [ #10 * #3/#1 ] ;
G00 Y #2 + #3 – #4 – #15 + #11 ;
G33 X#1 – #10 – #18 Y –#3 + #15 ;
G00 Y – #2 + #4 – #11 ;
IF [ #14 GT 0 ] GOTO 3 ;
IF [ #16 GT 0 ] GOTO 7 ;
#12 = #12 + 1 ;
#13 = ROUND [ #9 * SQRT [ #12 ] ] ;
```

```
IF [ ABS [ #13 – #11 ] GE [ ABS [ #8 ] ] ] GOTO 8 ;
#16 = 1 ;
N7 #13 = #11 + #8 ;
N8 #11 = #13 ;
IF [ ABS [ #11 ] LT [ ABS [ #4 – #5 ] ] ] GOTO 9 ;
#11 = #4 – #5 ;
#14 = 1 ;
N9 #10 = ROUND [ [ #17 – #11 ] * #7 ] ;
IF [ [ #10XOR#1] GE 0 ] GOTO 6 ;
#10 = –#10 ;
N6 #10 = #10 + #20 ;
G00 X – #1 + #10 + #18 ;
IF [ #14 LT 0 ] GOTO 11 ;
#18 = 0 ;
GOTO 12 ;
N11 #18 = #19 – #10 + #20 ;
N12 END 1 ;
N3 IF [ ABS [ #6 ] LT 1 ] GOTO 5 ;
#14 = 0 ;
#13 = 0 ;
D0 2 ;
IF [ #14 GT 0 ] GOTO 5 ;
#13 = #13 + #6 ;
IF [ ABS [ #13 ] LT [ ABS [ #5 ] ] ] GOTO 4 ;
#13 = #5 ;
#14 = 1 ;
N4 G00 X #10 – #1 ;
G00 Y #2 + #3 – #4 + #13 – #15 + #11 ;
G33 X #1 – #10 Y – #3 + #15 ;
G00 Y – #2 + #4 – #13 – #11 ;
END 2 ;
N5 G00 X – #1 ;
M99 ;
```

G76.1 (O761)	2-system simultaneous compound thread cutting cycle

```
G. 1 ;
N761 !L10
#12 = 1 ;
#13 = #9 ;
1F [ ABS [ #13 ] GE [ ABS [ #8 ] ] ] GOTO 1 ;
#16 = 1 ;
#13 = #8 ;
N1 #11 = #13 ;
1F [ ABS [ #11 ] LT [ ABS [ #4 – #5 ] ] ] GOTO 2 ;
#11 = #4 – #5 ;
#14 = 1 ;
N2 #17 = #11 ;
#18 = ROUND [ [ #4 – #11 – #5 ] * #7 ] ;
IF [ [ #18 XOR #1 ] GE 0 ] GOTO 10 ;
#18 = – #18 ;
N10 #19 = #18 ;
#10 = ROUND [ [ #11 + #5 ] * #7 ] ;
IF [ [ #10 XOR #1 ] GE 0 ] GOTO 20 ;
#10 = – #10 ;
N20 G00 X#10 ;
```

```
#20 = #10
D0 1 ;
#15 = ROUND [ #10 * #3/#1 ] ;
G00 Y #2 + #3 – #4 – #15 + #11 ;
!L11 ;
G33 X#1 – #10 – #18 Y –#3 + #15 ;
G00 Y – #2 + #4 – #11 ;
!L12 ;
IF [ #14 GT 0 ] GOTO 3 ;
IF [ #16 GT 0 ] GOTO 7 ;
#12 = #12 + 1 ;
#13 = ROUND [ #9 * SQRT [ #12 ] ] ;
IF [ ABS [ #13 – #11 ] GE [ ABS [ #8 ] ] ] GOTO 8 ;
#16 = 1 ;
N7 #13 = #11 + #8 ;
N8 #11 = #13 ;
IF [ ABS [ #11 ] LT [ ABS [ #4 – #5 ] ] ] GOTO 9 ;
#11 = #4 – #5 ;
#14 = 1 ;
N9 #10 = ROUND [ [ #17 – #11 ] * #7 ] ;
IF [ [ #10XOR#1] GE 0 ] GOTO 6 ;
#10 = –#10 ;
N6 #10 = #10 + #20 ;
G00 X – #1 + #10 + #18 ;
IF [ #14 LT 0 ] GOTO 11 ;
#18 = 0 ;
GOTO 12 ;
N11 #18 = #19 – #10 + #20 ;
N12 END 1 ;
N3 IF [ ABS [ #6 ] LT 1 ] GOTO 5 ;
#14 = 0 ;
#13 = 0 ;
D0 2 ;
IF [ #14 GOTO ] GOTO 5 ;
#13 = #13 + #6 ;
IF [ ABS [ #13 ] LT [ ABS [ #5 ] ] ] GOTO 4 ;
#13 = #5 ;
#14 = 1 ;
N4 G00 X #10 – #1 ;
G00 Y #2 + #3 – #4 + #13 – #15 + #11 ;
!L11 ;
G33 X #1 – #10 Y – #3 + #15 ;
G00 Y – #2 + #4 – #13 – #11 ;
!L12 ;
END 2 ;
N5 G00 X – #1 ;
M99 ;
```

G76.2 (O762)	2-system simultaneous compound thread cutting cycle

```
G. 1 ;
N762 !L10 ;
#12 = 1 ;
#13 = #9 ;
1F [ ABS [ #13 ] GE [ ABS [ #8 ] ] ] GOTO 1 ;
#16 = 1 ;
#13 = #8 ;
N1 #11 = #13 ;
1F [ ABS [ #11 ] LT [ ABS [ #4 – #5 ] ] ] GOTO 2 ;
#11 = #4 – #5 ;
#14 = 1 ;
N2 #17 = #11 ;
#18 = ROUND [ [ #4 – #11 – #5 ] ∗ #7 ] ;
IF [ [ #18 XOR #1 ] GE 0 ] GOTO 10 ;
#18 = – #18 ;
N10 #19 = #18 ;
#10 = ROUND [ [ #11 + #5 ] ∗ #7 ] ;
IF [ [ #10 XOR #1 ] GE 0 ] GOTO 20 ;
#10 = – #10 ;
N20 IF [# 27 NE 1 ] GOTO 21 ;
G00 X#10 ;
N21 #20 = #10 ;
#28 = 1 ;
D01 ;
#15 = ROUND [ #10 ∗ #3/#1 ] ;
#29 = #28 MOD 2 ;
IF [ [ #27 EQ 1 ] AND [#29 EQ 0 ] GOTO 22 ;
IF [ [ #27 EQ 2 ] AND [#29 EQ 1 ] GOTO 22 ;
G00 Y #2 + #3 – #4 – #15 + #11 ;
!L11 ;
G33 X#1 – #10 – #18 Y –#3 + #15 ;
G00 Y – #2 + #4 – #11 ;
#21 = #18 ;
!L12 ;
N22 IF [ #14 GT 0 ] GOTO 3 ;
IF [ #16 GT 0 ] GOTO 7 ;
#12 = #12 + 1 ;
#13 = ROUND [ #9 ∗ SQRT [ #12 ] ] ;
IF [ ABS [ #13 – #11 ] GE [ ABS [ #8 ] ] ] GOTO 8 ;
#16 = 1 ;
N7 #13 = #11 + #8 ;
N8 #11 = #13 ;
IF [ ABS [ #11 ] LT [ ABS [ #4 – #5 ] ] ] GOTO 9 ;
#11 = #4 – #5 ;
#14 = 1 ;
N9 #10 = ROUND [ [ #17 – #11 ] ∗ #7 ] ;
IF [ [ #10XOR#1] GE 0 ] GOTO 6 ;
#10 = –#10 ;
N6 #10 = #10 + #20 ;
IF [ [ #27 EQ 1 ] AND [#29 EQ 1 ] GOTO 24 ;
IF [ [ #27 EQ 2 ] AND [#29 EQ 0 ] GOTO 24 ;
IF [ [ #27 EQ 2 ] AND [#28 EQ 1 ] GOTO 23 ;
G00 X – #1 + #10 + #21 ;
```

```
GOTO 24 ;
N23 G00 X#10 ;
N24 IF [ #14 LT 0 ] GOTO 11 ;
#18 = 0 ;
GOTO 12 ;
N11 #18 = #19 – #10 + #20 ;
N12 #28 = #28 + 1 ;
END 1 ;
N3 IF [ ABS [ #6 ] LT 1 ] GOTO 5 ;
#14 = 0 ;
#13 = 0 ;
D0 2 ;
IF [ #14 GT 0 ] GOTO 5 ;
#13 = #13 + #6 ;
IF [ ABS [ #13 ] LT [ ABS [ #5 ] ] ] GOTO 4 ;
#13 = #5 ;
#14 = 1 ;
N4 #29 = #28 MOD 2 ;
IF [ [ #27 EQ 1 ] AND [#29 EQ 1 ] GOTO 25 ;
IF [ [ #27 EQ 2 ] AND [#29 EQ 0 ] GOTO 25 ;
G00 X #10 – #1 + #21 ;
#21 = 0 ;
G00 Y #2 + #3 – #4 + #13 – #15 + #11 ;
!L11 ;
G33 X #1 – #10 Y – #3 + #15 ;
G00 Y – #2 + #4 – #13 – #11 ;
!L12 ;
N25 #28 = #28 + 1 ;
END 2 ;
N5 G00 X – #1 ;
M99 ;
%
```

G77 (O770)	Longitudinal cutting cycle

```
G. 1 ;
1F [ [ #1 EQ 0 ] OR [ #2 EQ 0 ] ] GOTO 1 ;
Y #2 + #7 ;
G1 X #1 Y – #7 ;
Y – #2 ;
G0 X – #1 ;
N1 M99 ;
```

G78 (O780)	Thread cutting cycle

```
G. 1 ;
1F [ [ #1 EQ 0 ] OR [ #2 EQ 0 ] ] GOTO 1 ;
Y #2 + #7 ;
G33 X #1 Y – #7 F #9 E #10 ;
G0 Y – #2 ;
X – #1 ;
N1 M99 ;
```

G79 (O790)	End face cutoff cycle

```
G. 1 ;
1F [ [ #1 EQ 0 ] OR [ #2 EQ 0 ] ] GOTO 1 ;
X #1 + #7 ;
G1 X – #7 Y #2 ;
X – #1 ;
G0 Y – #2 ;
N1 M99 ;
```

G83 G87 (O830)	Deep hole drilling cycle B

```
G. 1 ;
1F [ #30 ] GOTO 2 ;
M #24 ;
#29 = #11 #28 = 0 ;
Z #2 ;
#2 = ##5 #3003 = #8 OR 1 ;
D0 1 ;
#28 = #28 – #11 #26 = – #28 – #29 ;
Z #26 ;
IF [ ABS [ #28 ] GE [ ABS [ #3 ] ] ] GOTO 1 ;
G1 Z #29 ;
G0 Z #28 ;
G29 = #11 + #14 ;
END 1 ;
N1 G1 Z #3 – #26 ;
G4 P #4 ;
#3003 = #8 ;
G0Z – #3 – #2 ;
IF [#24 EQ #0 ] GOTO 2 ;
M #24 + 1 ;
G4 P #21 ;
N2 M99 ;
```

G83 G87 (O831)	Deep hole drilling cycle A

```
G. 1 ;
1F [ #30 ] GOTO 2 ;
M #24 ;
#29 = #0 #28 = #11 ;
Z #2 ;

#2 = ##5 #3003 = #8 OR 1 ;
D0 1 ;
#29 = #29 + #11 ;
IF [ ABS [ #29 ] GE [ ABS [ #3 ] ] ] GOTO 1 ;
G1 Z #28 ;
G0 Z – #14 ;
#28 = #11 + #14 ;
END 1 ;
N1 G1 Z #3 – #29 + #28 ;
G4 P #4 ;
#3003 = #8 ;
G0Z – #3 – #2 ;
IF [#24 EQ #0 ] GOTO 2 ;
M #24 + 1 ;
G4 P #21 ;
N2 M99 ;
```

G83.2 (O832)	Deep hole drilling cycle 2

```
G. 1 ;
1F [ #30 ] GOTO 3 ;
#3003 = #8 OR 1 ;
#29 = #12 #28 = 0 ;
G0 Z #2 ;
IF [ #12 NE #0 ] GOTO 1 ;
IF [ #11 EQ #0 ] GOTO 2 ;
N1 #28 = #28 – #12 #26 = – #28 – #29 ;
IF [ ABS [ #28 ] GE [ ABS [ #3 ] ] ] GOTO 2 ;
G1 Z #12 ;
G4 P #4 ;
G0 Z #28 – #2 ;
G4P # 13 ;
G29 = #11 + #15 ;
D0 1 ;
#28 = #28 – #11 #26 = – #28 – #29 ;
G0 Z #26 + #2 ;
IF [ ABS [ #28 ] GE [ ABS [ #3 ] ] ] GOTO 2 ;
G1 Z #29 ;
G4 P #4 ;
G0 Z #28 – #2 ;
G4 P # 13 ;
END 1 ;
N2 G1 Z #3 – #26 ;
G4 P #4 ;
```

```
#3003 = #8 ;
G0Z – #3 – #2 ;
N3 M99 ;
```

G84 G88 (O840)	Tap cycle

```
G. 1 ;
1F [ #30 ] GOTO 2 ;
M #24 ;
Z #2 ;
#2 = ##5 #3003 = #8 OR 1 #3004 = #9 OR 3 ;
G1 Z #3 ;
G4 P #4 ;
M4 ;
#3900 = 1 ;
G1 Z – #3 ;
#3004 = #9 ;
M3 ;
#3003 = #8 ;
IF [#24 EQ #0 ] GOTO 1 ;
M #24 + 1 ;
G4 P #21 ;
N1 G0 Z – #2 ;
N2 M99 ;
```

G85 G89 (O850)	Boring cycle

```
G. 1 ;
1F [ #30 ] GOTO 2 ;
M #24 ;
Z #2 ;
#2 = ##5 #3003 = #8 OR 1 ;
G1 Z #3 ;
G4 P #4 ;
#3003 = #8 ;
Z – #3 F #23 ;
F #22 ;
IF [#24 EQ #0 ] GOTO 1 ;
M #24 + 1 ;
G4 P #21 ;
N1 G0 Z – #2 ;
N2 M99 ;
```

4.6 Standard Fixed Cycle Subprogram (For M system)

G81 (O810)	Drill, spot drill

```
      G. 1 ;                          Fixed cycle block 1 movement command
      1F [#30] GOTO1 ;                Check for fixed cycle invalidity.
      Z#2 G#6 H#7 ;
      #2 = ##5
      #3003 = #8 OR 1 ;               Inhibit single block stop.
      G1 Z#3 ;
      #3003 = #8 ;
      G0 Z – #3 – #2 ;                Return.
N1    M99%
```

G82 (O820)	Drill, counter boring

```
      G. 1 ;                          Fixed cycle block 1 movement command
      1F [#30] GOTO1 ;                Check for fixed cycle invalidity.
      Z#2 G#6 H#7 ;
      #2 = ##5
      #3003 = #8 OR 1 ;               Inhibit single block stop.
      G1 Z#3 ;
      G4 P#4 ;                        Dwell.
      #3003 = #8 ;
      G0 Z – #3 – #2 ;
N1    M99%                            Return.
```

G83 (O830)	Deep hole drill cycle

```
      G. 1 ;                          Fixed cycle block 1 movement command
      1F [#30] GOTO2 ;                Check for fixed cycle invalidity.
      #29 = #11                       Define the cutting amount.
      #28 = 0                         Initialize the return amount (total cutting amount).
      Z#2 G#6 H#7 ;
      #2 = ##5
      #3003 = #8 OR 1 ;               Inhibit single block stop.
      DO  1 ;
      #28 = #28 – #11                 Define the return amount of the next block.
      #26 = – #28 – #29               Calculate the feed amount.
      Z#26 ;                          Feed.
      IF [ABS [#28] GE [ABS [#3] ] ] GOTO 1 ;   Does the total cutting amount (return amount)
                                                exceed the cut amount?
      G1 Z#29 ;                       Cutting feed
```

```
        G0 Z#28 ;                    Return.
        #29 = #11 + #14 ;            Define the cutting amount for block 2 and after.
        END1 ;
N1      G1 Z#3 – #26 ;               Cutting feed
        #3003 = #8 ;
        G0 Z – #3 – #2 ;             Return.
N2      M99%
```

G84 (O840)	Tap cycle

```
        G. 1 ;                       Fixed cycle block 1 movement command
        1F [#30] GOTO1 ;             Check for fixed cycle invalidity.
        Z#2 G#6 H#7 ;
        #2 = ##5
        #3003 = #8 OR 1 ;            Inhibit single block stop.
        #3004 = #9 OR 3 ;            Invalidate feed hold/override.
        G1 Z#3 ;
        G4 P#4 ;                     Dwell.
        M4 ;                         Rotate the spindle reversely.
        #3900 = 1 ;
        G1 Z – #3 ;
        #3004 = #9 ;
        G4 P#4 ;                     Dwell.
        M3 ;                         Rotate the spindle forward.
        #3003 = #8 ;
        G0 Z = #2 ;                  Return.
N1      M99%
```

G85 (O850)	Boring 1

```
        G. 1 ;                       Fixed cycle block 1 movement command
        1F [#30] GOTO1 ;             Check for fixed cycle invalidity.
        Z#2 G#6 H#7 ;
        #2 = ##5
        #3003 = #8 OR 1 ;            Inhibit single block stop.
        G1 Z#3 ;
        #3003 = #8 ;
        Z – #3 ;
        G0 Z – #2 ;                  Return.
N1      M99%
```

G86 (O860)	Boring 2

```
      G. 1 ;                          Fixed cycle block 1 movement command
      1F [#30] GOTO1 ;                Check for fixed cycle invalidity.
      Z#2 G#6 H#7 ;
      #2 = ##5
      #3003 = #8 OR 1 ;               Inhibit single block stop.
      G1 Z#3 ;
      G4 P#4 ;                        Dwell.
      M5 ;                            Stop the spindle.
      G0 Z – #3 – #2 ;                Return.
      #3003 = #8 ;
      M3 ;                            Rotate the spindle forward.
N1    M99%
```

G87 (O870)	Back boring

```
      G. 1 ;                          Fixed cycle block 1 movement command
      1F [#30] GOTO1 ;                Check for fixed cycle invalidity.
      #3003 = #8 OR 1 ;               Inhibit single block stop.
      M19 ;                           Orient the spindle.
      X#12 Y#13 ;
      #3003 = #8 ;                    Cancel single block stop inhibition.
      Z#2 G#6 H#7 ;
      #3003 = #8 OR 1 ;               Inhibit single block stop.
      G1 X – #12 Y – #13 ;
      #3003 = #8 ;                    Cancel single block stop inhibition.
      M3 ;                            Rotate the spindle forward.
      #3003 = #8 OR 1 ;               Inhibit single block stop.
      Z#3 ;
      M19 ;                           Orient the spindle.
      G0 X#12 Y#13 ;                  Shift.
      Z – #2 – #3 ;                   G87 is not affected by the G98 or G99 modal.
      #3003 = #8 ;                    Cancel single block stop inhibition.
      X – #12 Y – #13 ;               Shift.
      M3 ;                            Rotate the spindle forward.
N1    M99%
```

G88 (O880)	Boring 3

```
        G. 1 ;                                  Fixed cycle block 1 movement command
        1F [#30] GOTO1 ;                        Check for fixed cycle invalidity.
        Z#2 G#6 H#7 ;
        #2 = ##5                                Inhibit single block stop.
        #3003 = #8 OR 1 ;
        G1 Z#3 ;                                Dwell.
        G4 P#4 ;                                Cancel single block stop inhibition.
        #3003 = #8 ;                            Rotate the spindle forward
        M5 ;                                    Inhibit single block stop.
        #3003 = #8 OR 1 ;                       Return.
        G0 Z – #3 – #2 ;                        Cancel single block stop inhibition.
        #3003 = #8 ;                            Rotate the spindle forward.
        M3 ;
N1      M99 ;
```

G89 (O890)	Boring 4

```
        G. 1 ;                                  Fixed cycle block 1 movement command
        1F [#30] GOTO1 ;                        Check for fixed cycle invalidity.
        Z#2 G#6 H#7 ;
        #2 = ##5
        #3003 = #8 OR 1 ;                       Inhibit single block stop.
        G1 Z#3 ;
        G4 P#4 ;                                Dwell.
        #3003 = #8
        Z – #3 ;
        G0 Z – #2 ;                             Return.
N1      M99%
```

G73 (O831)	Step cycle

```
        G. 1 ;                                  Fixed cycle block 1 movement command
        1F [#30] GOTO2 ;                        Check for fixed cycle invalidity.
        #29 = 0                                 Initialize the total cutting amount.
        #28 = #11 ;                             Define the cutting amount.
        Z#2 G#6 H#7 ;
        #2 = ##5
        #3003 = #8 OR 1 ;                       Single block stop command
        DO  1 ;
        #29 = #29 + #11 ;                       Increment the total cutting amount counter.
        IF [ABS [#29] GE [ABS [#3] ] ] GOTO 1 ; Does the total cutting amount exceed the cut
                                                amount Z?
```

```
        G1 Z#28 ;                    Cutting feed
        G4 P#4 ;                     Dwell.
        G0 Z – #14 ;                 Return.
        #28 = #11 + #14 ;            Define the cutting amount for block 2 and after.
        END1 ;
N1      G1 Z#3 – #29 + #28 ;         Cutting feed
        G4 P#4 ;                     Dwell.
        #3003 = #8 ;
        G0Z – #3 – #2 ;              Return.
N2      M99%
```

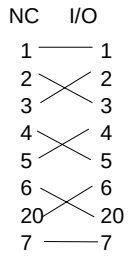
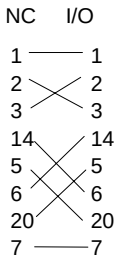
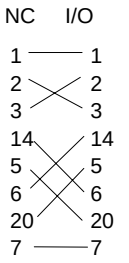
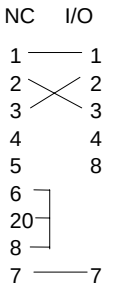
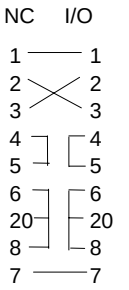
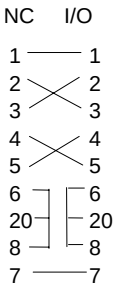
G74 (O841)	Reverse tap cycle

```
        G. 1 ;                       Fixed cycle block 1 movement command
        1F [#30] GOTO1 ;             Check for fixed cycle invalidity.
        Z#2 G#6 H#7 ;
        #2 = ##5
        #3003 = #8 OR 1 ;            Inhibit single block stop.
        #3004 = #9 OR 3 ;            Invalidate feed hold/override.
        G1 Z#3 ;
        G4 P#4 ;                     Dwell.
        M3 ;                         Rotate the spindle forward.
        #3900 = 1 ;
        Z – #3 ;
        #3004 = #9 ;
        G4 P#4 ;                     Dwell.
        M4 ;                         Rotate the spindle reversely.
        #3003 = #8 ;
        G0 Z – #2 ;
N1      M99%                         Return.
```

G76 (O861)	Fine boring

```
        G. 1 ;                       Fixed cycle block 1 movement command
        1F [#30] GOTO1 ;             Check for fixed cycle invalidity.
        Z#2 G#6 H#7 ;
        #2 = ##5
        #3003 = #8 OR 1 ;            Single block stop command
        G1 Z#3 ;
        M19 ;                        Orient the spindle.
        X#12 Y#13 ;                  Shift.
        G0 Z – #3 – #2 ;             Return.
        #3003 = #8 ;
        X – #12 Y – #13 ;            Shift.
        M3 ;                         Rotate the spindle forward.
N1      M99%
```

Appendix 5 RS-232C I/O Device Parameter Setting Examples and Cable Connection

I/O device / Parameter	Personal computer	Tape reader (Mitsubishi)	Tape puncher (Mitsubishi)	Printer (Mitsubishi)	Floppy disk drive unit (Kyohritsu-sha)	Floppy disk drive unit (Mitsubishi)	Floppy disk drive unit (Tanaka Business)
		PTR-02A	PTP-02A	PRT-02A	D-30	FD-3.5	TBM-F1
DEVICE NAME	Follows communi-cation software.						
BAUD RATE		2	2	2	2	2	2
STOP BIT		3	3	3	3	3	3
PARITY EFFECTIVE		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	Number of characters	0	0	0	0
PARITY V		0	0/1	0	0	0/1	0/1
TIME-OUT SET		100	100	100	100	100	100
Cable connection (Cable accessories)	Follows communi-cation protocol. (NC side) 2 : SD 3 : RD 4 : RS 5 : CS 6 : DR 20: ER 7 : GND						

Appendix 6 Data Input/Output Data List

The NC data that can be input/output with each connection method (RS-232C connection, Ethernet connection, IC card connection) is listed below.

Data type		Screen (connection method)				Remarks
		"INPUT" "OUTPUT" screen (RS232C)	"HOST" screen (Ethernet)	"HOST" screen (IC card)	"IC card" screen (IC card)	
Machining program		○	○	○	○	<"INPUT"/"OUTPUT" screen> All files can be input/output by setting the keyword ALL <"HOST" screen> Multiple files can be input/output by using "*" <"IC Card" screen> Multiple files cannot be input/output
Parameter		○	○ (Note 1)	○ (Note 1)	×	
Common variable (Note 3)		○	○ (Note 1)	○ (Note 1)	×	
Tool compensation data (Note 4)		○	○ (Note 1)	○ (Note 1)	×	
Tool life control data (Note 5)		○	○ (Note 1)	○ (Note 1)	×	
User macro		○	×	×	×	
Machine maker macro		○	×	×	×	
Auxiliary axis parameter		○	×	×	×	
Operation history		○	×	×	×	Cannot be input to the NC side.
Maintenance data	ALL1	○	×	×	×	
	ALL2	○	×	×	×	
	ALL3	○	○	○	×	
	SRAM data (Note 2)	×	○ (Note 1)	○ (Note 1)	×	
	Extended SRAM data (Note 2)	×	○ (Note 1)	○ (Note 1)	×	
Waveform data		○	×	×	×	

(Note 1) The following file names will be assigned if the file name is not assigned when outputting from the NC.

Data type	File name
Parameter	ALL.PRM
Common variable	COMMON.VAR
Tool compensation data	TOOL.OFS
Tool life control data	TLIFE.TLF
SRAM data	SRAM.BIN
Extended SRAM data	EXTSRAM.BIN

(Note 2) This data is for maintenance by Mitsubishi and must not be used by the user.

(Note 3) When inputting the common variables, set the following items to the conditions as when data is output.

- Number of systems
- Number of variable commands

If conditions do not match when inputting data to the NC, the error "E315 Write error" (during host transmission) and "E86 Input data illegal" (during data input) will occur.

(Note 4) When inputting the tool compensation data, set the following items to the same conditions as when data is output.

- Model class
- Number of systems
- Number of axes, axis name
- Compensation type
- Number of compensation sets

If conditions do not match when inputting data to the NC, the error "E315 Write error" (during host transmission) and "E86 Input data illegal" (during data input) will occur.

(Note 5) When inputting the tool life management data, set the following items to the same conditions as when data is output.

- Model class
- Number of systems
- Number of compensation sets

If conditions do not match when inputting data to the NC, the error "E315 Write error" (during host transmission) and "E86 Input data illegal" (during data input) will occur.

Appendix 7 Operation Messages on Setting and Display Unit

If a setting operation error occurs on any setting and display unit screen, the error No. E○○ and a message describing the details of the error will display in the line above the data setting area or the menu display area.

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 absolute position setting) 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 (0253 and 0254) of the language data to be output.
			• When the machine manufacturer macro program memory area is the SRAM area, the setup parameter #1060 SETUP was set to "20". • When the machine manufacturer macro program memory area is the SRAM area, writing of the machine manufacturer macro program was attempted on PROGRAM COPY screen.
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 workpiece 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 wear data of cutting edges used for the calculation.
			• #1003 iunit was set to D when the least increment command 0.01μm option was not available.
			• When there was no option, 2 or more was specified for #1043 lang. Otherwise, an option was added and 23 or more was specified for #1043 lang.

Error No.	Error message		Details
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.
E04	**DEV. NOT READY**	×	• The input/output unit 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 total of axes set in #1001 SYS_ON, #1002 axisno is illegal. Set so that the total No. of axes is within the specifications range for the target model. • #1037 cmdtyp is not within the setting range.
			• 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 first monitor screen was manipulated by manual command operation (M, S, and T keys).

Error No.	Error message		Details
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) • Set up parameter #1049 mmac_R was set to “1” when the machine manufacturer macro option was not valid. • Set up parameter #1060 SETUP was set to “20” when the machine manufacturer macro option was not valid. • Writing of the machine manufacturer macro program with the PROGRAM COPY screen was attempted when the machine manufacturer macro option was not valid.
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 “#9116 TIME-OUT SET” setting was too short. • There is no EOB code in the machining program.
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 MDI, 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. • A program with the same No. as the machining program to be copied from the memory was found in the IC card. • 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.

Error No.	Error message		Details
E15	**EDIT LOCK B**	×	• An operation (edit, input/output, buffer correction, etc.) inhibited for machining program B was attempted.
E16	**EDIT LOCK C**	×	• An operation (edit, input/output, buffer correction, etc.) inhibited for machining program C 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. • When using the two systems, the program being buffer corrected was running with the other system. • Erasing or inputting (IC → NC) of a program in the IC card being used was attempted. • Formatting of the IC card was attempted during automatic start. • Accessing to the host computer was attempted during automatic 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. • When the machine manufacturer macro program memory area was set to the FROM area, formatting of the FROM area (#1060 SETUP “20”) was attempted when the PLC was not stopped, writing of the machine manufacturer macro program was attempted on the PROGRAM COPY screen, or input of the macro program was attempted. (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.
E40	**OP MODE ERROR**	×	• Continuous or step graphic check was not possible because the operation mode was illegal.
E50 **E51** **E52** **E53** **E54** **E55** **E56**	**FILE ERR** **FILE OPEN ERR** **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.

Error No.	Error message		Details
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 – 2 (Port already being used) 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 ERR) E60 lOP ERROR – 18 (E22 CODE CHANGE ERR) E60 IOP ERROR – 20 (framing and H/W errors) • 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). • Data was input/output or the tape search was executed during the host link. Set 0 in #8109 HOST LINK, and then set 1 again before performing the cycle start. (IOP error -2) • The host link parameter was turned ON during connecting to the Anshin-net. Turn the Anshin-net valid OFF. (IOP error -2) • When #10812 Anshin-net/ Machine builders network system valid is set to 1, the modem connection port of the Anshin-net or machine builders network system is occupied. Perform inputting or outputting using a port other than the modem connection port. (IOP error -2) • When connecting the GX Developer (when the bit selection parameter #6451 bit5 is set to 1), the port 2 of the RS232C communication port is always used. Use a port other than the port 2 of the RS232C communication port. (IOP error -2)

Error No.	Error message		Details
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.
E73	**CAN’T CALCULATE**	×	• The coordinate value of the hole center cannot be obtained. • Reset the measurement point, which must not applied to the following conditions. The measurement A point is the same as the Y coordinate of the C point. The measurement B is the same as the Y coordinate of the C point. The slope of the line through A and C point is the same as the slope of the line through B and C point.
E74	**MENU IMPOSSIBLE**	×	• Press the operation menu “= Input” or “+ input” during the tool measurement. • Press the operation menu “= Input” or “+ input” during the manual value command mode. • Press the screen selection menu on which “↓” is displayed during the tool measurement. • Press the screen selection menu on which “↓” is displayed during the manual value command mode.

Error No.	Error message		Details
E75	**TLM ILL. SIGNAL**		• The sensor signal was already ON when the tool measurement mode (TLM) signal was validated. • After the tool measurement mode (TLM) signal was validated, the sensor signal turned ON when there was no axis movement. • The sensor signal turned ON at a position within 100μm from the final entry start position. • Turn the tool measurement mode signal input OFF, or turn the sensor signal OFF and move the axis in a safe direction. Note) This display will be erased when another screen is opened. The display will not be erased even if the tool measurement mode signal input is turned OFF, or if the axis is moved in a direction away from the sensor.
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 the zero point.
E80	**TOP SEARCH ERR**	×	• The program head search (unmodal type search) was not executed before type 2 (standard specification) restart search was executed for program restart. Set the type to unmodal, search for the head of the program, and then search for the restart block with type 2.
E82	**ALREADY RESEARCH**	×	• After completing the type 1 or type 2 search for program restart, the unmodal type, type 1 or type 2 search was attempted again. If program restart is continued (if the axis is return to the restart position with automatic or manual operations), the program will restart from the block searched for first. To search again, cancel the previous search by resetting, and then search again.
E84	**CAN'T IN/OUT**	×	• An attempt was made to input a parameter in the setup parameter locked state. Refer to the manual issued by the machine manufacturer. • The parameter "#1925 EtherNet" of the high-speed program server function is set to 0. • When writing data to the IC card, the file name is illegal. (Exceeding 8 characters of file name + 3 characters of extension.) • Input of maintenance data from the host or IC card was attempted.

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. • The format of file written to the NC memory is illegal.
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 "PB_G90" was OFF. • G91 was set when "PB_G90" was ON.
E98	**CAN'T RESEARCH**	×	• When restarting the program, the type 3 restart search was attempted with a program containing no T command. Check the program. • When restarting the program, the T command corresponding to the type 3 restart search was not found in the program. Check the program. • When restarting a program for 2-systems, restart search was performed simultaneously for the 1st and 2nd systems, and then the 2nd system was searched again. If program restart is continued (if the axis is return to the restart position with automatic or manual operations), the program will restart from the block searched for first. To search again, cancel the previous search by resetting, and then search again. • When carrying our program restart, type 3 restart search was attempted while the machine was at the negative (–) side of the restart limit parameters. Manually move the machine to the positive (+) side of the restart limit parameters, and search again.
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 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 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**	×	• A socket connection attempt failed during Ethernet communication. • Check the host address, the setting of the port No. and that the host computer is turned ON.
E302	**LOGIN ERR**	×	• A login attempt failed during Ethernet communication. Check the user name and password. • Check the account settings, such as the home directory.
E303	**TIME OUT**	×	• Transmission of a file with Ethernet communication ended because of timeout.
E311	**DOWNLOAD ERR**	×	• An attempt to read a host file failed during Ethernet communication.
E312	**UPLOAD ERR**	×	• An attempt to write to a host file failed during Ethernet communication.
E313	**NO FILE**	×	• The file specified by host receive (host → IC) operation during Ethernet communication is not found in the host. • The file specified by host send (IC → host) operation during Ethernet communication is not found in the IC card.
E314	**FILE DUPLICATE**	×	• The file name specified to be stored by host receive (host → IC) operation during Ethernet communication already exists in the IC card. • The file name specified to be loaded by host send (IC → host) operation during Ethernet communication already exists in the host.
E315	**FILE WRITE ERR**	×	• An attempt to write to the IC card failed during Ethernet communication.
E316	**FILE READ ERR**	×	• An attempt to read a file from the IC card failed during Ethernet communication.
E317	**MEMORY OVER**	×	• IC card memory is full. • NC memory is full.
E318	**OVER FLOW ERR**	×	• A host directory contains too many files.
E319	**DIRECTORY ERR**	×	• An attempt to move a directory failed. • In the IC card device, accessing a directory in the nineteenth layer or more was attempted.
E320	**HR437 UNEQU**	×	• When backup or writing of the expansion cassette into/from the FROM, the expansion cassette (HR437) was not mounted in CBUS#1 or was incorrectly mounted. • When backup or writing of the expansion cassette into/from the FROM, a card other than the expansion cassette (HR437) was mounted in CBUS#1. • When the program memory was formatted to 1280m or more, the expansion cassette (HR437) was not mounted in CBUS#1.

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 COMPLET	• Search was completed normally.
BUFFER EDIT	The buffer is being corrected. This appears when the cursor or a tab key is pressed and the buffer correction mode is entered. This is erased when INPUT is pressed.
CAN'T BUF. EDIT	When using 2 systems, the program being buffer corrected is being used by the other system. Buffer correcting of a machine manufacturer macro program was attempted.
DATA PROTECTING	Buffer correcting is prohibited since the data protection key 3 is valid.

(2) MDI/editing related

Message	Message details
MDI NO SETTING	• Only display of MDI data (no execution)
MDI SETTING COMPLETE	• 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 SERCH/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.
PROGRAM RUNNING	• A machining program to be edited is currently being run with memory operation, and cannot be edited.
DELETE? (Y/N)	• 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.

Message	Message details
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 paper tape.
DATA WRITING IN PROGRESS	• 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 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	• The machining program is being copied without error.
COPY COMPLETE	• The machining program 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.

(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

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) Parameter backup related

Message	Message details
BACKUP EXEC. Y/N	Type “Y” or “N” to specify whether to perform the operation.
BACKUP EXECUTION	The parameters are being backed up.
BACKUP COMPLTE	Backup of the parameters has been completed.
RESTORE EXEC. Y/N	Type “Y” or “N” to specify whether to perform the operation.
RESTORE EXECUTION	The parameters are being restored.
RESTORE COMPLETE	Restoration of the parameters has been completed.

(7) 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.
EXECUTE 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. Setup with #1060 SETUP "1" has been completed. Formatting with #1060 SETUP "20" has been completed.
NON SETUP	• Completed without executing simple setup. (When “N” has been set for both “BASE PARA. SET? (Y/N)” and “EXECUTE FORMAT? (Y/N)”.)
CONFIRM OPE? (Y/N)	• Confirmation for erasing operating time or alarm history.
INPUT? (Y/N)	• Waiting for the key input of tool length data by manual measurement.
V-ANALIZER EXEC.	• Waveform display data cannot be output while waveform is displayed.
ROM WRITE? (Y/N)	• The system is waiting for a key input to indicate whether to write the macro programs into the FROM.
DATA WRITING	• The macro program is being written into the FROM.
WRITE COMPLETE	• The macro program has been written into the FROM.

Revision History

Date of revision	Manual No.	Revision details
Feb. 1998	BNP-B2180*	First edition created.
Mar. 2000	BNP-B2180A	The content of "MELDAS64 INSTRUCTION MANUAL (SUPPLEMENT)" (BNP-B2248) was reflected.
May 2000	BNP-B2180B	• The explanation about synchronous error monitor screen was added. • The parameters and messages were added and changed. • Mistakes or illegal descriptions are corrected.
Sep. 2001	BNP-B2180C	• The contents revised to correspond to M64AS, M64S, M65S and M66S. • Miswrite is corrected. • Design of the cover and the back cover were changed. • Manual name changed from "MELDAS 64/65 Operation Manual" to "MELDAS 60/60S Series Operation Manual". • MODEL, MODEL CODE, and Manual No. were added on the back cover.
Apr. 2002	BNP-B2180D	• Revisions to comply with M60S Series Version B0 • Miswrite is corrected.
Mar. 2003	BNP-B2180E	• Revisions to comply with M60S Series Version B3 • Miswrite is corrected.
May 2004	BNP-B2180F	• Revisions to comply with M60S Series Version C0 and C1.
Feb. 2008	BNP-B2180G	• Precautions for the program check were added. • Precautions were added for each deletion of the common variables, tool offset data, tool registration data, and the tool life management data. • Precautions for the operation mode were added.
Feb.2010	BNP-B2180J	• Reviewed "Precautions for Safety". • Corrected the mistakes.

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)

Central Region Service Center
500 CORPORATE WOODS PARKWAY, VERNON HILLS, IL., 60061, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Western Michigan Service Satellite
ALLEGAN, MICHIGAN., 49010, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-269-673-4092

Ohio Service Satellite
LIMA, OHIO, 45801, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Minnesota Service Satellite
RICHFIELD, MINNESOTA, 55423, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Western Region Service Center
5665 PLAZA DRIVE, CYPRESS, CALIFORNIA, 90630, U.S.A.
TEL: +1-714-220-4796 / FAX: +1-714-229-3818

Eastern Region Service Center
200 COTTONTAIL LANE SOMERSET, NEW JERSEY, 08873, U.S.A.
TEL: +1-732-560-4500 / FAX: +1-732-560-4531

Western Pennsylvania Service Satellite
ERIE, PENNSYLVANIA, 16510, U.S.A.
TEL: +1-814-897-7820 / FAX: +1-814-987-7820

Southern Region Service Center
2810 PREMIERE PARKWAY SUITE 400, DULUTH, GEORGIA, 30097, U.S.A.
TEL: +1-678-258-4500 / FAX: +1-678-258-4519

Northern Texas Service Satellite
1000, NOLEN DRIVE SUITE 200, GRAPEVINE, TEXAS, 76051, U.S.A.
TEL: +1-817-251-7468 / FAX: +1-817-416-5000

Southern Texas Service Satellite
FRIENDSWOOD, TEXAS, 77546, U.S.A
TEL: +1-832-573-0787 / FAX: +1-678-573-8290

Central Florida Service Satellite
SATELITE BEACH, FLORIDA, 32937, U.S.A.
TEL : +1-321-610-4436 / FAX : +1-321-610-4437

Canadian Region Service Center
4299 14TH AVENUE MARKHAM, ONTARIO, L3R OJ2, CANADA
TEL: +1-905-475-7728 / FAX: +1-905-475-7935

Mexico City Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-9171-7662 / FAX: +52-55-9171-7649

Monterrey Service Satellite
ARGENTINA 3900, FRACC. LAS TORRES, MONTERREY, N.L., 64720, MEXICO
TEL: +52-81-8365-4171 / FAX: +52-81-8365-4171

Brazilian Service Center
ACESSO JOSE SARTORELLI, KM 2.1 CEP 18550-000, BOITUVA-SP, BRAZIL
TEL: +55-15-3363-9900 / FAX: +55-15-3363-9911

Brazilian's Sites Service Center
CITIES OF PORTO ALEGRE AND CAXIAS DO SUL BRAZIL
CITIES OF SANTA CATARINA AND PARANA STATES
TEL: +55-15-3363-9927

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V. (EUROPE FA CENTER)
GOTHAER STRASSE 10, 40880 RATINGEN, GERMANY
TEL: +49-2102-486-0 / FAX: +49-2102-486-5910

Germany Service Center
KURZE STRASSE. 40, 70794 FILDERSTADT-BONLANDEN, GERMANY
TEL: + 49-711-3270-010 / FAX: +49-711-3270-0141

France Service Center
25, BOULEVARD DES BOUVETS, 92741 NANTERRE CEDEX FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

France (Lyon) Service Satellite
120, ALLEE JACQUES MONOD 69800 SAINT PRIEST
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

Italy Service Center
VIALE COLLEONI 7-PALAZZO SIRIO CENTRO DIREZIONALE COLLEONI,
20041 AGRATE BRIANZA MILANO ITALY
TEL: +39-039-60531-342 / FAX: +39-039-6053-206

Italy (Padova) Service Sattelite
VIA SAVELLI 24 - 35129 PADOVA ITALY
TEL: +39-039-60531-342 / FAX: +39-039-6053-206

U.K. Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-27-6100 / FAX: +44-1707-27-8992

Spain Service Center
CTRA. DE RUBI, 76-80-APDO. 420
08190 SAINT CUGAT DEL VALLES, BARCELONA SPAIN
TEL: +34-935-65-2236 / FAX: +34-935-89-1579

Poland Service Center
UL.KRAKOWSKA 50, 32-083 BALICE, POLAND
TEL: +48-12-630-4700 / FAX: +48-12-630-4727

Poland (Wroclaw) Service Center
UL KOBIERZYCKA 23,52-315 WROCLAW,POLAND
TEL: +48-71-333-77-53 / FAX: +48-71-333-77-53

Turkey Service Center
BAYRAKTAR BULVARI, NUTUK SOKAK NO:5, YUKARI DUDULLU
ISTANBUL, TURKEY
TEL: +90-216-526-3990 / FAX: +90-216-526-3995

Czech Republic Service Center
TECHNOLOGICKA 374/6,708 00 OSTRAVA-PUSTKOVEC, CZECH REPUBLIC
TEL: +420-59-5691-185 / FAX: +420-59-5691-199

Russia Service Center
213, B.NOVODMITROVSKAYA STR., 14/2, 127015 MOSCOW, RUSSIA
TEL: +7-495-748-0191 / FAX: +7-495-748-0192

Sweden Service Center
STRANDKULLEN, 718 91 FROVI, SWEDEN
TEL: +46-581-700-20 / FAX: +46-581-700-75

Bulgaria Service Center
4 A. LYAPCHEV BOUL., 1756 - SOFIA, BULGARIA
TEL: +359-2-8176000 / FAX: +359-2-9744061

Ukraine (Kharkov) Service Center
APTEKARSKIY PEREULOK 9-A, OFFICE 3, 61001 KHARKOV, UKRAINE
TEL: +38-57-732-7744 / FAX: +38-57-731-8721

Ukraine (Kiev) Service Center
4-B, M. RASKOVOYI STR., 02660 KIEV, UKRAINE
TEL: +38-044-494-3355 / FAX: +38-044-494-3366

Belarus Service Center
703, OKTYABRSKAYA STR., 16/5, 220030 MINSK, BELARUS
TEL: +375-17-210-4626 / FAX: +375-17-227-5830

South Africa Service Center
P.O. BOX 9234, EDLEEN, KEMPTON PARK GAUTENG, 1625 SOUTH AFRICA
TEL: + +27-11-394-8512 / FAX: +27-11-394-8513

ASEAN

MITSUBISHI ELECTRIC ASIA PTE. LTD. (ASEAN FA CENTER)

Singapore Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 / FAX: +65-6476-7439

Indonesia Service Center
WISMA NUSANTARA 14TH FLOOR JL. M.H. THAMRIN 59, JAKARTA 10350 INDONESIA
TEL: +62-21-3917-144 / FAX: +62-21-3917-164

Malaysia (KL) Service Center
60, JALAN USJ 10 /1B 47620 UEP SUBANG JAYA SELANGOR DARUL EHSAN, MALAYSIA
TEL: +60-3-5631-7605 / FAX: +60-3-5631-7636

Malaysia (Johor Baru) Service Center
No. 16, JALAN SHAH BANDAR 1, TAMAN UNGKU TUN AMINAH, 81300 SKUDAI, JOHOR MALAYSIA
TEL: +60-7-557-8218 / FAX: +60-7-557-3404

Vietnam Service Center-1
47-49 HOANG SA ST. DAKAO WARD, DIST. 1, HO CHI MINH CITY, VIETNAM
TEL: +84-8-910-4763 / FAX: +84-8-910-2593

Vietnam Service Center-2
THUAN KIEN PLAZA 190 HONG BANG ROAD.TOWER C. SUITE 3002. DIST.5,
HO CHI MINH CITY,VIETNAM
TEL: +84-8-240-3587 / FAX: +84-8-726-7968

Vietnam (Hanoi) Service Center
5TH FL., 93B KIM LIEN ST., PHUONG LIEN WARD, DONG DA DIST.
TEL: +84-8-573-7646 / FAX: +84-4-573-7650

Philippines Service Center
UNIT No.411, ALABAMG CORPORATE CENTER KM 25. WEST SERVICE ROAD
SOUTH SUPERHIGHWAY, ALABAMG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416 / FAX: +63-2-807-2417

MITSUBISHI ELECTRIC AUTOMATION (THAILAND) CO., LTD. (THAILAND FA CENTER)

BANG-CHAN INDUSTRIAL ESTATE No.111 SOI SERITHAI 54
T.KANNAYAO, A.KANNAYAO, BANGKOK 10230, THAILAND
TEL: +66-2906-8255 / FAX: +66-2906-3239

Thailand Service Center
898/19,20,21,22 S.V. CITY BUILDING OFFICE TOWER 1 FLOOR 7
RAMA III RD BANGPONGPANG,YANNAWA,BANGKOK 10120. THAILAND
TEL: +66-2-682-6522 / FAX: +66-2-682-9750

INDIA

MITSUBISHI ELECTRIC ASIA PVT LTD

FIRST & SECOND FLOOR, AVR BASE, MUNICIPAL No.BC-308,
HENNURE BANASWADI ROAD, HRBR RING ROAD, BANGALORE-560 043,INDIA
TEL: +91-80-4020-1600 / FAX: +91-80-4020-1699

India (Pune) Service Center
EL-3, J BLOCK, M.I.D.C., BHOSARI PUNE 411026, INDIA
TEL: +91-20-2710-2000 / FAX: +91-20-2710-2185

India (Bangalore) Service Center
S 615, 6TH FLOOR, MANIPAL CENTER, BANGALORE 560001, INDIA
TEL: +91-80-509-2119 / FAX: +91-80-532-0480

India (Delhi) Service Center
1197, SECTOR 15 PART-2, OFF DELHI-JAIPUR HIGHWAY BEHIND 32nd MILESTONE
GURGAON 122001, INDIA
TEL: +91-98-1024-8895

OCEANIA

MITSUBISHI ELECTRIC AUSTRALIA LTD.

Oceania Service Center
348 VICTORIA ROAD, RYDALMERE, N.S.W. 2116 AUSTRALIA
TEL: +61-2-9684-7269 / FAX: +61-2-9684-7245

CHINA

MITSUBISHI ELECTRIC AUTOMATION (SHANGHAI) LTD. (CHINA FA CENTER)

China (Shanghai) Service Center
4/F ZHI FU PLAZA, NO. 80 XIN CHANG ROAD,
SHANGHAI 200003,CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2322-2800
China (Ningbo) Service Dealer
China (Wuxi) Service Dealer
China (Jinan) Service Dealer

China (Beijing) Service Center
9/F, OFFICE TOWER 1, HENDERSON CENTRE, 18 JIANGUOMENNEI AVENUE
DONGCHENG DISTRICT, BEIJING, CHINA 100005
TEL: +86-10-6518-8830 / FAX: +86-10-6518-8030
China (Beijing) Service Dealer

China (Tianjin) Service Center
B-2-801-802, YOUYI BUILDING. 50 YOUYI ROAD, HEXI DISTRICT
TIANJIN, CHINA 300061
TEL: +86-22-2813-1015 / FAX: +86-22-2813-1017
China (Shenyang) Service Satellite

China (Chengdu) Service Center
BLOCK B-1, 23F, CHUAN XIN MANSION, 18 SECTION 2
RENMIN ROAD (SOUTH), CHENGDU, SICHUAN, CHINA 610016
TEL: +86-28-8619-9730 / FAX: +86-28-8619-9805

China (Changchun) Service Satellite
BING 3RD ROAD., INDUSTRIAL&ECONOMICAL DEVELOPMENT ZONE, CHAOYANG DISTRICT
TEL : +86-0431-5021-546 / FAX : +86-0431-5021-690

China (Shenzhen) Service Center
ROOM 2512-2516, GREAT CHINA INTERNATIOANL EXCHANGE SQUARE, JINTIAN RD.S.,
FUTIAN DISTRICT, SHENZHEN, CHINA 518034
TEL: +86-755-2399-8272 / FAX: +86-755-8218-4776

KOREA

MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD. (KOREA FA CENTER)

Korea Service Center
1480-6, GAYANG-DONG, GANGSEO-GU SEOUL 157-200 ,KOREA
TEL: +82-2-3660-9602 / FAX: +82-2-3664-8668

Korea Taegu Service Satellite
603 CRYSTAL BUILDING 1666, SANBYEOK-DONG, BUK-KU, DAEGU, 702-010, KOREA
TEL: +82-53-604-6047 / FAX: +82-53-604-6049

TAIWAN

MITSUBISHI ELECTRIC TAIWAN CO., LTD(TAIWAN FA CENTER)

TAIWAN (Taichung) Service Center
No.8-1, GONG YEH 16TH RD., TAICHUNG INDUSTRIAL PARK TAICHUNG CITY, TAIWAN R.O.C
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

TAIWAN (Taipei) Service Center
3TH. FLOOR, No.122 WUKUNG 2ND RD., WU-KU HSIANG, TAIPEI HSIEN, TAIWAN R.O.C
TEL: +886-2-2299-2205 / FAX: +886-2-2298-1909

TAIWAN (Tainan) Service Center
2F(C),1-1, CHUNGHWA-RD, YONGKANG CITY, TAINAN HSIEN, TAIWAN R.O.C
TEL: +886-6-313-9600 / FAX: +886-6-313-7713

Notice

Every effort has been made to keep up with software and hardware revisions in the contents described in this manual. However, please understand that in some unavoidable cases simultaneous revision is not possible.

Please contact your Mitsubishi Electric dealer with any questions or comments regarding the use of this product.

Duplication Prohibited

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : TOKYO BUILDING,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	MELDAS 60/60S Series
MODEL CODE	005—232
Manual No.	BNP-B2180(ENG)

(1002) MEE