SECTION 1   DATA SETTING

This section describes the commands used in the parameter set, zero set, and tool data set mode operation. The procedure for setting the zero point data and tool data is also explained. For the procedure used for setting the parameter data and the contents of the parameters, refer to III "PARAMETER" in this manual.

1.   Mode Selection Keys
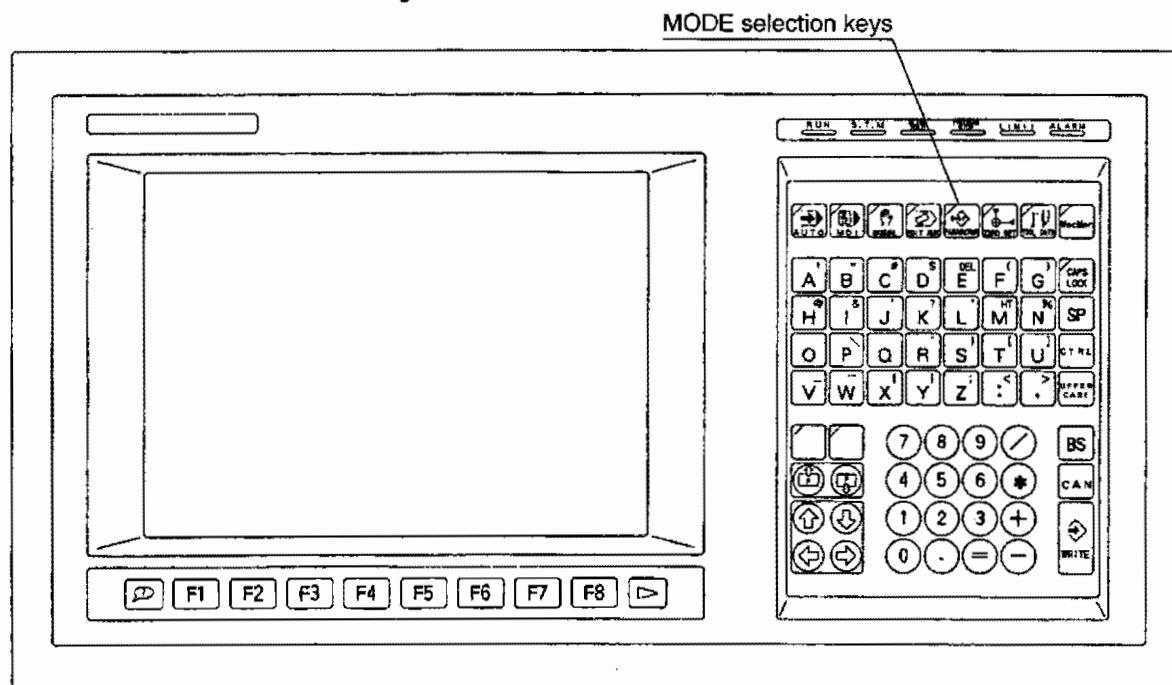
Fig. 1-1 Mode Selection Keys

The mode selection keys are classified into the three groups, operation mode, data setting mode, and machining management (MacMan) mode.

(1)   Operation Mode

(a) AUTO

In the automatic mode, the part program stored in the memory area is read into the operation buffer and then executed.

(b) MDI

In the MDI mode, the program is input from the keyboard and executed.

(c) MANUAL

In the manual mode, machine operation is controlled using the switches on the machine operation panel.

(2) Data Setting Mode

(a) EDIT/AUX

This is the mode for reading, editing, punching-out, and printing out the program, operating the tape reader, and managing files.

(b) PARAMETER

Parameters and variables, including system parameters, user parameters, common variables and NC optional parameters are set.

For details of the parameters, refer to IV "PARAMETER" in this manual.

(c) TOOL DATA

Used to set the tool data such as tool length offset values and cutter radius compensation values.

(d) ZERO SET

In this mode, the work zero offset value is set.

(3) MacMan Mode

The actual status of the production field is collected in this mode, and the result of data processing is output to the NC screen, printer, or 3.5-inch floppy disk.

For details of the MacMan mode, refer to the separately prepared manual, "MacMan INSTRUCTION MANUAL".

(4) Pressing any of the mode operation selection keys will light the lamp in the key.

(5) If the mode is switched from any of the operation modes to the data setting mode, the lamp in the selected operation mode key starts flashing. This allows the operator to recognize the selected operation mode at a glance.

(6) Once program execution is started in the AUTO mode or MDI mode, switching the mode to Data Setting Mode, such as EDIT/AUX, PARAMETER, TOOL DATA, and ZERO SET, will not interrupt the program execution. Therefore, program editing, tape punching-out, tape reading-in, parameter setting, zero setting, tool data setting and other data setting operations are possible during stored program execution.

When program execution halts in the AUTO mode operation, while data setting operation is being carried out, due to Single Block ON or Program Stop command, it is necessary to press the CYCLE START switch after switching the mode to AUTO to continue the AUTO mode operation.

(7) Switching the operation modes in the following sequence will reset the control:

(a) From manual to auto or MDI

(b) From auto or MDI to manual

(c) When the mode is switched from manual to the data setting mode, although the control is not reset, it is reset when any operation is selected after that.

⚠ CAUTION : When the data has been set, the new data is stored in memory after two to three minutes have passed. Therefore, if the power is turned off immediately after the data has been set, the data may not be updated as desired.

Before turning off the power supply and after setting the data, backup the data following the procedure explained in 3-4-1. "Back Up Command".

2.   Data Setting

Here is a list of parameters for data setting, together with their contents.

Category	Subcategory	Sub subcategory	Entry Type	Axis Type	Data Elements	Unit	Internal Data Type
Work coordinate origin	—	—	Microns; integer	Linear axis	4 to Max. 100	Inch system Metric system	4-byte integer
				Rotary axis	4 to Max. 100	Degrees	4-byte integer
Tool data	Tool data	Tool length offset	Microns; integer	Linear axis	50 to Max. 300	Inch system Metric system	4-byte integer
		Cutter radius compensation	Microns; integer	Linear axis	50 to Max. 300	Inch system Metric system	4-byte integer
Parameter	Common variable	—	Microns; integer	—	200	—	8-byte floating-point data
	System parameter	Travel end limit	Microns; integer	Liner axis	2/1 axis	Inch system Metric system	4-byte integer
				Rotary axis	2/1 axis	Degrees	4-byte integer
		Pitch error compensation range	Microns; integer	Liner axis	2/1 axis	Inch system Metric system	4-byte integer
				Rotary axis	2/1 axis	Degrees	4-byte integer
		Pitch error compensation interval	2-byte; integer	Liner axis	1/1 axis	Inch system Metric system	4-byte integer
				Rotary axis	1/1 axis	Degrees	4-byte integer
		Number of pitch error compensation points	2-byte; integer	Liner axis	1/1 axis	Inch system Metric system	4-byte integer
				Rotary axis	1/1 axis	Degrees	4-byte integer
		Zero return operation execution sequence	2-byte; integer	Liner axis	1/1 axis	Inch system Metric system	4-byte integer
				Rotary axis	1/1 axis	Degrees	4-byte integer
		In-position width	2-byte; integer	Liner axis	1/1 axis	Inch system Metric system	4-byte integer
				Rotary axis	1/1 axis	Degrees	4-byte integer

Category	Subcategory	Sub subcategory	Entry Type	Axis Type	Data Elements	Unit	Internal Data Type
Parameter	System parameter	Home position	2-byte; integer	Liner axis	32 axis	Inch system Metric system	4-byte integer
				Rotary axis	32 axis	Degrees	4-byte integer
		Machine coordinate system origin	Microns; integer	Liner axis	1/1 axis	Inch system Metric system	4-byte integer
				Rotary axis	1/1 axis	Degrees	4-byte integer
	User parameter	Programmable travel limit	Microns; integer	Liner axis	2/1 axis	Inch system Metric system	4-byte integer
				Rotary axis	2/1 axis	Degrees	4-byte integer
		G60 over-passing amount	Microns; integer	Liner axis	1/1 axis	Inch system Metric system	4-byte integer
				Rotary axis	1/1 axis	Degrees	4-byte integer
		Backlash	Microns; integer	Liner axis	1/1 axis	Inch system Metric system	4-byte integer
				Rotary axis	1/1 axis	Degrees	4-byte integer
	G/M code macro	Simple call	Character	—	10	—	4-byte character
		Axis move call	Character	—	10	—	4-byte character
	Program name	—	Character	—	4	—	4-byte character
	Pitch error compensation	—	Integer	Liner axis	384	—	2-byte integer
				Rotary axis	384	—	2-byte integer
	NC optional parameter (long word)	—	Integer	—	64	—	4-byte integer
	NC optional parameter (word)	—	Integer	—	120	—	2-byte integer
	NC optional parameter (bit)	—	Bit string	—	512 bits	—	1-byte

[Supplement]   The unit system for data setting can be set at NC optional parameter (bit) No. 9.

3.   Data Set Commands
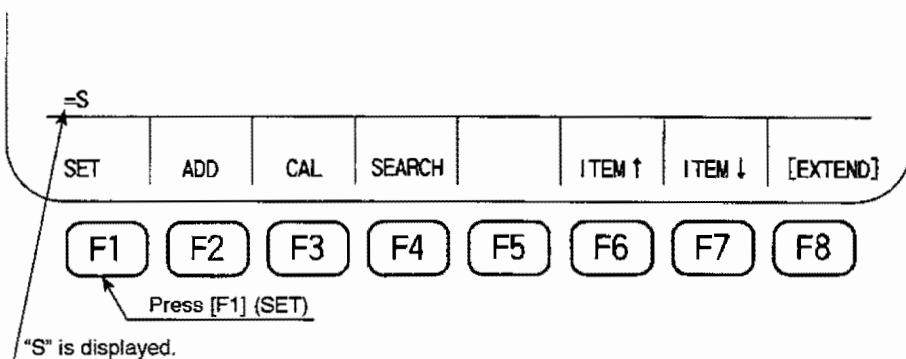
The data set commands are detailed in the following pages, and may be classified into three groups: commands for data setting, commands for parameter selection, and input/output commands for peripherals.

3-1.   Commands for Data Setting

This group consists of three commands:

SET, ADD, and CAL
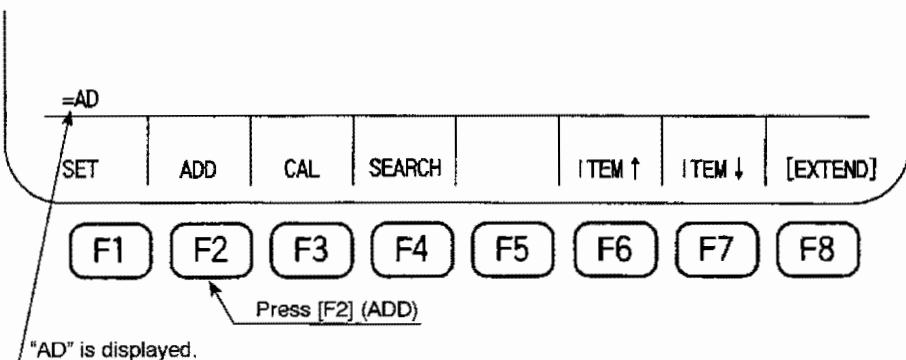
3-1-1.   SET Command

The data entered itself is a command serving as new data. The SET command is effective when function key [F1] is pressed, and displays "S" on the 21st line on the display screen.

Enter the data from the keyboard and press the WRITE key, then the keyed in data is input.

### 3-1-2.   ADD Command
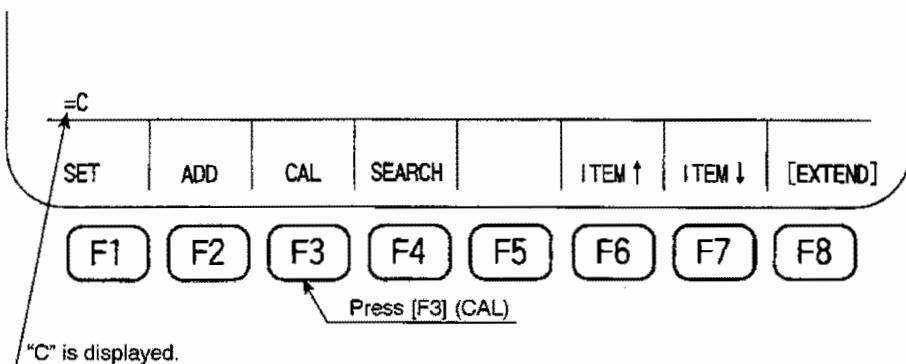
The data entered is added to the current data to form a command serving as new data.

The ADD command is effective when function key [F2] is pressed, providing a display of "AD" on the 21st line on the display screen.

Enter the data from the keyboard and press the WRITE key, and the entered data is added to the current data. The result of addition is input as the new data.

3-1-3. CAL Command

"C" is displayed.

Press [F3] (CAL)

An arithmetic operation is carried out between the data entered and the current value to form a command serving as new data for setting.

The CAL command is effective by pressing function key [F3], providing a display of "C" on the 21st line on the display screen.
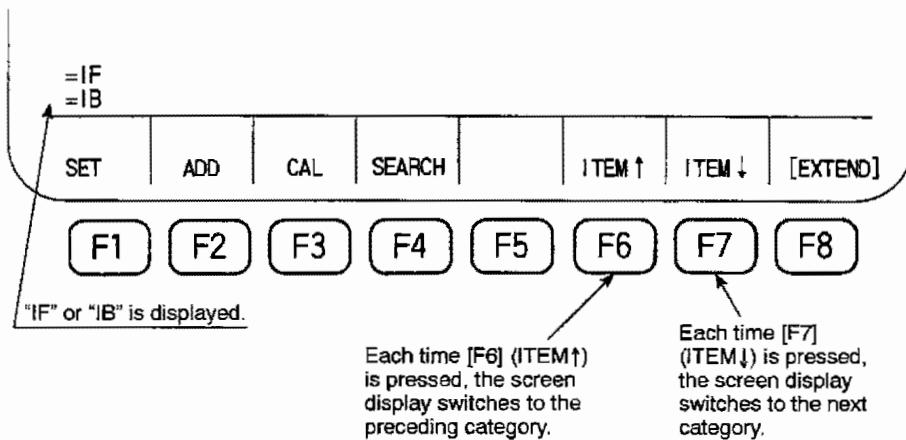
Enter the data from the keyboard and press the WRITE key, and the math operation is executed between the entered data and the current data. The result of math operation is input as the new data.

3-2. Commands for Parameter Selection

This group consists of six commands:

ITEM, SEARCH, AXIS CHANGE, CURSOR, PAGE and EXTEND

3-2-1. ITEM Command

"IF" or "IB" is displayed.

Each time [F6] (ITEM↑) is pressed, the screen display switches to the preceding category.

Each time [F7] (ITEM↓) is pressed, the screen display switches to the next category.
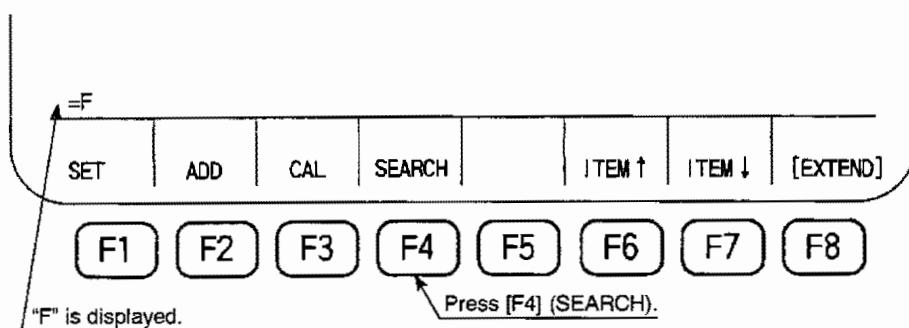
This command is intended to select an appropriate parameter from those belonging to the subcategories in the table given in Section 4, 2. "DATA SETTING".

The ITEM command is carried out when function key [F6] or [F7] is pressed. Each time one of these keys is pressed, the display screen is switched over to the preceding or next subcategory items.

"IF" or "IB" is displayed on the 21st line on the display screen and the screen display is switched over.

3-2-2. SEARCH Command

"F" is displayed.

Press [F4] (SEARCH).

In the subcategory group, which involves several data elements, the screen display is, sometimes, greater than one page. This SEARCH command, engaged by function key [F4], may be conveniently used to set the cursor with ease, although it is possible to place the cursor at the desired data using the page key and the cursor key.

Operating procedure is given below:

(1) Press function key [F4].

This produces display of "F" on the 21st line on the display screen.

(2) Key in a data number on the keyboard.

Display of "F" is followed by this particular data.

(3) Press the WRITE key.

This allows the cursor to be positioned at the data number specified. If the current screen does not include the data number, then the screen is switched over until that data number appears and the cursor is positioned there.

If there is nothing corresponding to the data number entered, the cursor is then positioned as follows:

(1) When smaller

The cursor is positioned at the first data.

(2) When greater

The cursor is positioned at the last data.

The screen for setting bit type data, NC optional parameter (bit)) allows the simultaneous setting of both parameter number searching and parameter number setting.

(1) Press function key [F4].
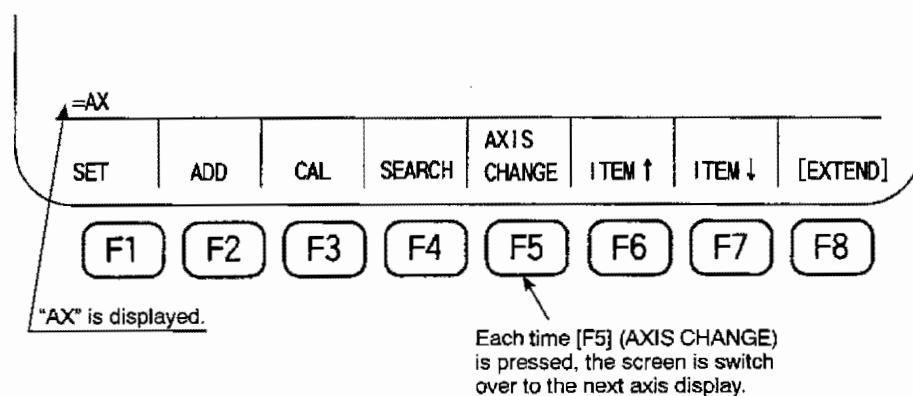
(2) Key in the parameter number.

(3) Key in a comma (,).

(4) Key in a bit number.

(5) Press the WRITE key.

3-2-3. AXIS CHANGE Command

| SET | ADD | CAL | SEARCH | AXIS CHANGE | ITEM ↑ | ITEM ↓ | [EXTEND] |

=AX

"AX" is displayed.

Each time [F5] (AXIS CHANGE) is pressed, the screen is switch over to the next axis display.
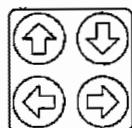
In data setting, if there is a parameter of axis type, the 5th and 6th axes cannot be displayed on the same screen. The AXIS CHANGE command is intended to switch the screen from one display to another.

Since the pitch error compensation data may be displayed on one axis per screen, the command is used for axis selection.

When function key [F5] is pressed, "AX" appears on the 21st line on the display screen, resulting in a screen switch.

3-2-4. Cursor Keys
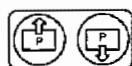
Move the cursor in the arrow-pointed direction, where the cursor stays at only the points including data to be set. When, for example, the cursor is at

rest at the extreme left, press the cursor key (←) ;

this brings the cursor to the data element located at the extreme right. When the cursor is at the

extreme right, press the cursor key (→) ; this

brings the cursor to the data element located at the extreme left. A similar relationship exists between

the cursor keys (↓) and (↑) .
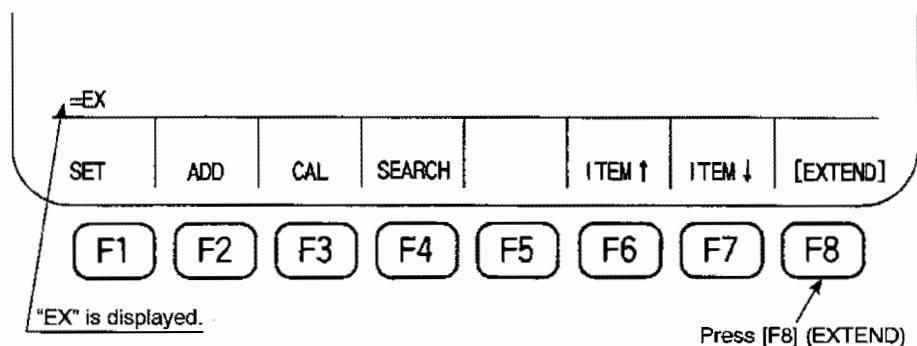
3-2-5. Page Keys

When the (↓P) key is pressed, the screen advances one page in the same category only.

When the (↑P) key is pressed, the screen returns one page in the same category only.

3-2-6. EXTEND Command

Data setting involves many commands, which cannot be displayed at the same time (there are eight function keys available). The EXTEND command is intended to switch command displays.

Function key [F8] provides a display of "EX" on the 21st line on the screen, resulting in a switched command only.
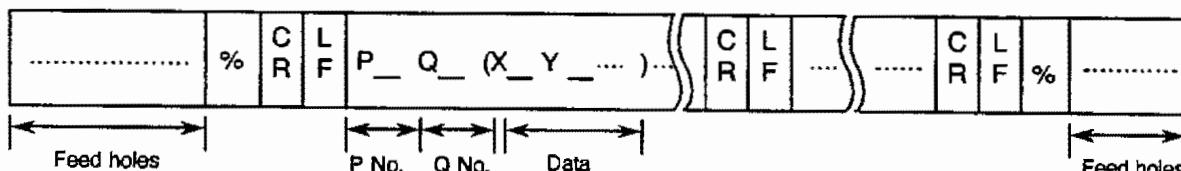
3-3. Input/Output Commands for Peripherals

This group consists of two commands: READ and PUNCH.

3-3-1. READ Command

This is a command to read a variety of data required for data setting from paper tape and set it in place.

The format is shown below:

The P and Q numbers must be specified, and are related to data setting as follows: P and Q specify the category and subcategory. Axis data is specified by X, Y, and Z, and the data not related to an axis is specified by R.

P and Q numbers are modal. It is not necessary to reprogram if the number is the same as the one programmed previously.

Both ISO and EIA codes may be used as conventional programs as the coding system.

The relationship of data setting, P numbers and Q numbers, is as indicated below.

Category	Subcategory	P No.	Q No.	Remark
Work coordinate system origin	—	100	1 to No. of work coordinate systems	
Tool data	Tool length offset	200	1 to No. of offsets	
	Cutter radius compensation	210	1 to No. of offsets	
Parameter	Common variable	300	1 to 200	
	System parameter	400	1 to 56	1 to 40 : real 41 to 56 : integer (Supplement)
	User parameter	410	1 to 32	
	Pitch error compensation	600	1 to No. of compensation points.	(Supplement)
	NC optional parameter (long word)	700	1 to 64	1 to 32 : user 33 to 64 : system
	NC optional parameter (word)	710	1 to 120	1 to 32 : user 33 to 120: system
	NC optional parameter (bit)	720	1 to 64	1 to 16 : user 17 to 64 : system

Operating procedure is given below:

① Press PARAMETER, TOOL DATA or ZERO SET.

⑤ Press WRITE.

④ Press [F1] (READ).
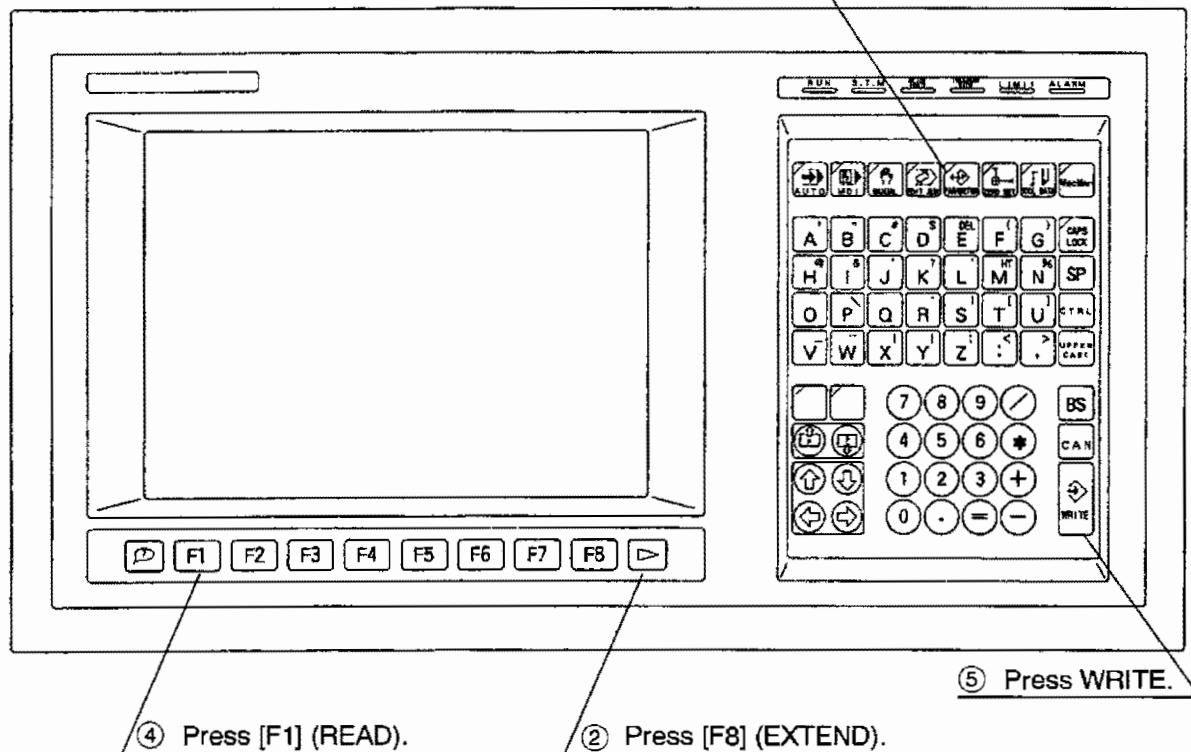
② Press [F8] (EXTEND).

Fig. 1-2 Reading the Data from Tape

① Press the PARAMETER, TOOL DATA or ZERO SET key.

② Press function key [F8] (EXTEND).

This displays the READ (F1) command.

③ Set a paper tape for data setting on the tape reader so that the leading feed holes come to the reader.

④ Press function key [F1] (READ).

This displays "R" on the 21st line on the display screen.

⑤ Press the WRITE key.

This allows the paper tape to advance, so that the data on the tape is sequentially read and set.

Note that the tape reader stops when an error occurs during the setting and the display screen shows the message below:

read continuing? (Y/N)!

Press Y or N key. Keying in "Y" permits the tape reader to continue reading. The entry of "N" aborts the read command.

3-3-2.   PUNCH Command

This is a command to punch various data required for data setting on the paper tape. Paper tape format is the same as with the READ operation. This command allows setting data to be stored on a paper tape.

Tape format is the same as indicated in Section 4, 3-3-1. "READ Command".

Punching out data under special conditions are shown below:

R   Normal data
R/0   Empty
R/1   + overflow
R/2   − overflow
R/3   + underflow
R/4   − underflow

When the data of the type indicated above is entered, the same condition is set. Operating procedure is shown as follows:

① Press PARAMETER, TOOL DATA or ZERO SET.

③ Press [F2] (PUNCH).
② Press [F8] (EXTEND).
⑥ Key in a P No.
⑤⑦ Press WRITE.
④ Enter the device name.
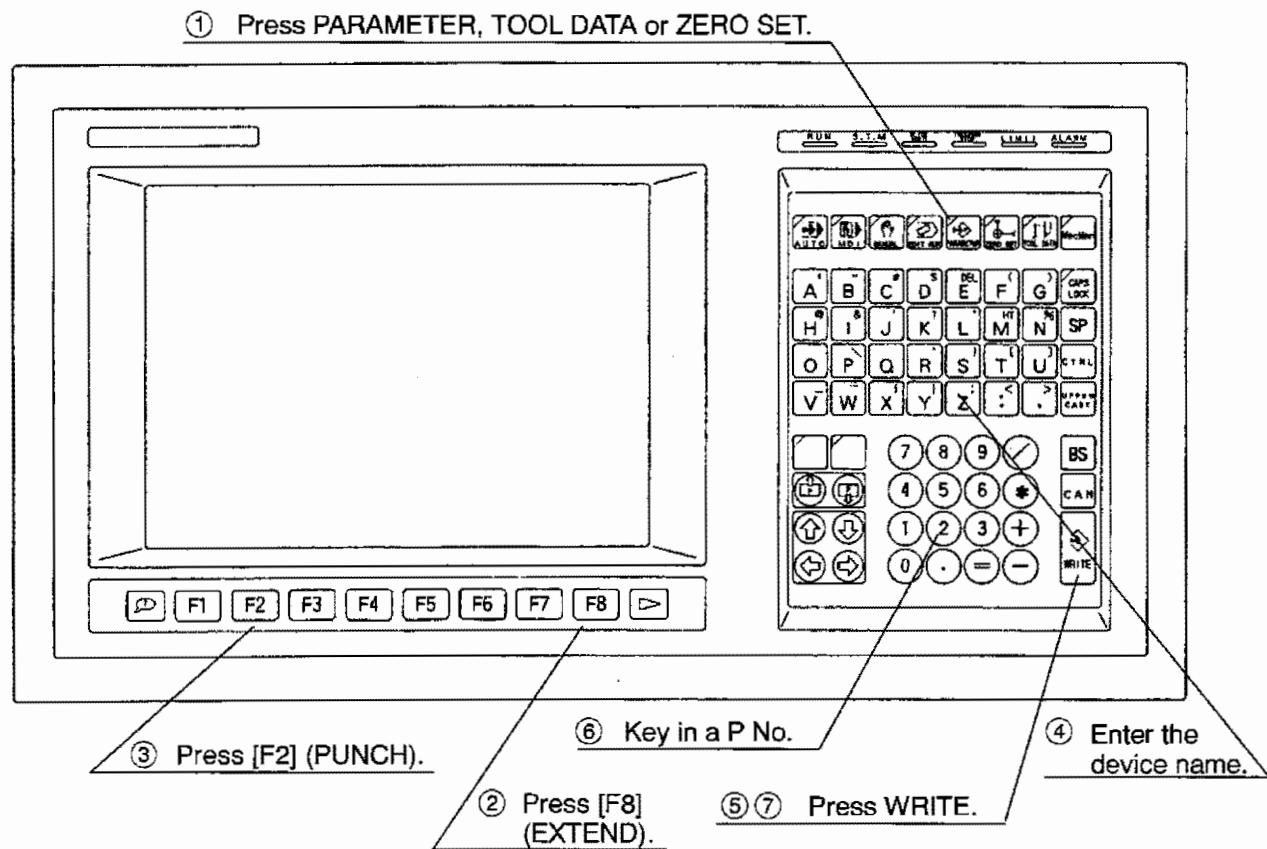
Fig. 1-3 PUNCH Command

① Press the PARAMETER, TOOL DATA or ZERO SET key.

② Press function key [F8] (EXTEND)

This displays the PUNCH [F2] command.

③ Press function key [F2] (PUNCH).

This displays "PU" on the 21st line on the display screen.

④ Enter the device name following "PU",the following names are usable.

PU ⌴ device name: File name

If a sector device is designated, a search will be conducted for the relevant sector device, and a check will be conducted to determine if a file of the same name already exists. If a file of the same name is found, a "File exists. over write? (Y/N)!" message will be displayed. Enter "Y" to execute overwriting, or "N" to cancel the punch operation. A "Comment input" message will be displayed, requesting any comments which may be required regarding the punch data. Enter a comment if necessary. If no comment is required, nothing should be entered.

* Parentheses marks "(" and ")" cannot be used in the comment. If used, an error will occur.

"5320 Wring character"

⑤ Press the WRITE key.

The punch menu is displayed under the heading of ✳PARAMETER DATA TAPE PUNCH MENU✳ and then the feed holes and the percent code (%) are punched on paper tape.

The prompt [parameter No.!] then appears.

```
┌─────────────────────────────────────────────────────────┐
│ │PARAMETER SET│                                          │
│                                        │97/07/15 14:10:00││
│              *PARAMETER DATA TAPE PUNCH MENU*             │
│                                                          │
│   P NO.   ITEM            P NO.   ITEM                    │
│                                                          │
│   100   ZERO OFFSET       700   OPTIONAL PARAMETER (LONG WORD) │
│   200   TOOL LENGTH OFFSET 710  OPTIONAL PARAMETER (WORD) │
│   210   CUTTER RADIUS COMP 720  OPTIONAL PARAMETER (BIT)  │
│   230   TOOL MANAGE DATA                                  │
│   300   COMMON VARIABLE                                   │
│   400   SYSTEM PARAMETER                                  │
│   410   USER PARAMETER                                    │
│   600   PITCH ERROR COMP                                  │
│                                                          │
│                                                          │
│   =EX                                                     │
│   =PU TT:                                                 │
│   plese imput message !                                  │
│   parameter No.  !                                       │
│  ───────────────────────────────────────────────────    │
│  READ  │ PUNCH │ VERIFY │    │    │    │ BACKUP │[EXTEND] │
└─────────────────────────────────────────────────────────┘
   ( F1 )( F2 )( F3 )( F4 )( F5 )( F6 )( F7 )( F8 )
```

⑥ From the menu, select the item number of the parameter data to be punched. The data entry format is as below:

$$\boxed{\text{P No. (A)}} \quad \left[-\boxed{\text{P No. (B)}} \right] \quad \left[,\boxed{\text{Q No. (A)}} \left(-\boxed{\text{Q No. (B)}} \right) \right]$$

Entry of a P and Q number should be made without characters P and Q. Use a delimiter "," when a Q number is entered following a P number.

When a range of P numbers or Q numbers is to be specified, the P number (B) or the Q number (B) must be larger than the P number (A) or the Q number (A), respectively.

For the data entry, see examples below:

Example 1: Punching out all tool length offset data

parameter No. !200 [or 200-200]

Example 2: Punching out all the data from tool length offset to common variable

parameter No. !200-300

Example 3: Punching out No. 8 data of tool length offset data

parameter No. !200, 8 [or 200, 8-8]

Example 4: Punching out No. 8 data of parameter data from tool length offset data to common variable data

parameter No. !200-300, 8

Example 5: Punching out tool length offset data within a required range, from No. 2 to No. 8, for example

parameter No. !200, 2-8

Example 6: Punching out within a required range of parameter data from tool length offset data to common variable data

parameter No. !200-300, 2-8

⑦ Press the WRITE key.

The data within the selected range is punched out from the specified output device.

⑧ After the completion of punching out the desired data, the parameter No. ! is displayed again. Repeat the steps f) and g) to punch out all the required data.

⑨ Finally, press only the WRITE key.

The percent (%) code and the trailing feed holes are punched out on the paper tape, thus ending the data punch mode. The display mode is also restored from the punch menu to the originally selected mode.

3-3-3. Verify

The VERIFY operation occurs in the same manner as the READ operation. The data which is read is not determined by the parameter settings. There may be cases when an error occurs even though the numeral displayed at "Parameter Set" is the same as that at the verify data. However, this is due simply to the fact that the number of digits following the displayed numeral are being held by the OSP, and there is no actual error.

A "tolerance" designation can be made at the VERIFY operation as follows:

DV Device name; file name; numeric value

By entering the optional tolerance (numeric value) designation, an error will not occur even if a data mismatch occurs, provided that part of the parameter data which is read is within the tolerance range. Only positive integers can be designated as tolerance values. The tolerance system-of-units varies according to the parameter types being compared. Regarding data items with decimal points (system parameters, user parameters, zero offset, tool data, etc.), comparison will begin from the smallest displayed numeral (following decimal point). Optional parameter words and long parameter words, etc., have no decimal points, and are therefore compared as they are.

If the data error value exceeds the tolerance range, the following error message will be displayed:

5375 DATA VERIFY ERROR "MISMATCH LINES"

The following parameters (including the tolerance value) are compared during the VERIFY operation:

PNo.

100	Zero offset
200	Tool length offset
210	Cutter radius compensation
400	System parameter (excluding home position return order data)
410	User parameter
600	Thread pitch offset
700	NC optional parameter (long word)
710	NC optional parameter (word)
720	NC optional parameter (bit)

3-3-4. Omitting the File Name and Device Name

- If the device name is not specified at the READ and VERIFY operations, the setting will be determined by optional parameter word No. 104 (initial value: 0, setting range: 0-11).

The devices which correspond to the setting values are shown below.
(This parameter setting becomes valid when power is switched ON.)

Setting Value	0	1	2	3	4	5	6	7	8	9	10	11
Device Name	TR	CN0	CN1	CN2	CN3	CN4	MD0	MD1	FD0	FD1	FD2	FD3

The "CN0" device name shown above represents the same device as the "TT" (teletyper) name.

- If the device name is not specified at the PUNCH operation, the setting will be determined by optional parameter word No. 103 (initial value: 0, setting range: 0-11). The devices which correspond to the setting values are shown below.
(This parameter setting becomes valid when power is switched ON.)

Setting Value	0	1	2	3	4	5	6	7	8	9	10	11
Device Name	CN0	CN1	CN2	CN3	CN4	PP	MD0	MD1	FD0	FD1	FD2	FD3

The "CN0" device name shown above represents the same device as the "TT" (teletyper) name.

- If the file name is not specified, a default file name will be designated according to operation function and device in question.

The default file name is "A.TOP". This function features separate default values for the file and extend names. If only the extension name is omitted, it will be designated as ".TOP".

3-4.   Other Commands

The NC has other commands as explained below:

3-4-1.   Back Up Command

⚠️CAUTION : Data such as tool length offset, cutter radius compensation, origin of work coordinate systems and parameter except system parameter is backed up to the memory in a preset interval. Therefore, turning off the power to the control right after renewing the data will cause the data to stay as it was without being updated.
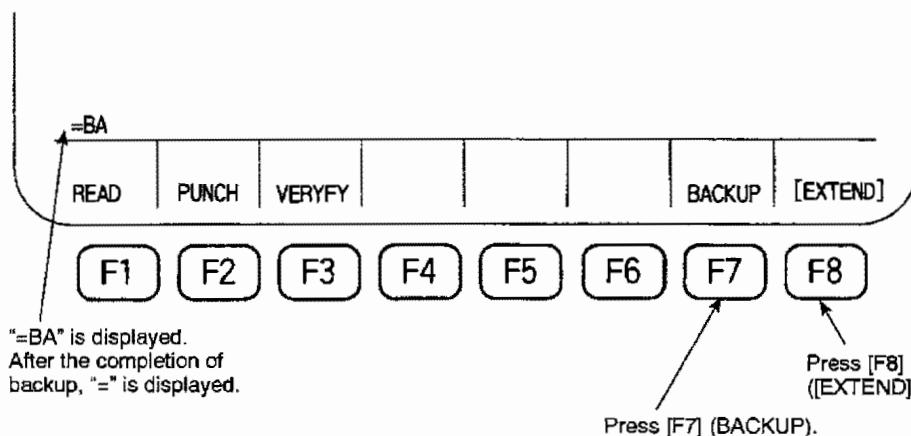
The execution of the back up command will update the data even when the power is turned off.

Operating procedure is shown below:

(1)   Press the PARAMETER, ZERO SET or TOOL DATA key.

(2)   Press function key [F8] (EXTEND).
The commands will be changed as indicated right.

"=BA" is displayed.
After the completion of
backup, "=" is displayed.

Press [F7] (BACKUP).

Press [F8]
([EXTEND]).

(3)   Press function key [F7] (BACKUP).

"= BA" appears on the console line and back up is continuously executed.
After back up completion, "=" will appear on the screen.
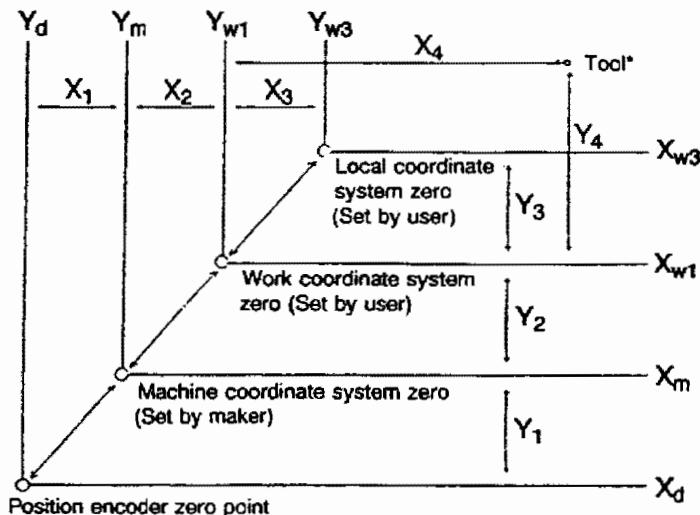
Now, all the data has been backed up and the power to the control may be turned off at any time.

4.   Zero Set Commands

4-1.   Work Coordinate System Origin

This is the origin of the work coordinate system referenced to the origin of the machine coordinate system. As the standard feature, 20 sets of the work coordinate system origin can be set, and this can be expanded to 50 or 100 sets.

Y_1:   Machine zero offset amount

Y_2:   Work zero offset amount

Fig. 1-4 Work Coordinate System Origin

A function called the machine coordinate system origin has been implemented. After the position detector is replaced, all that remains is to set another machine coordinate system origin, without having to set all of the work coordinate system origins (and others) at new values. This may be applied to soft-limits, thus helping to reduce the jobs required after the position encoder is replaced.

How to set the work coordinate system origin as follows:

(1)   Press the ZERO SET key.

This changes the display screen to Zero Set display.

(2)   One page of the screen can display 10 sets of data. To display the data for the 11th and later sets,

press the page key (▣) .

From the standpoint of axes, one page displays the data for four axes. Therefore, to display the data for the fifth and later axes, press function key [F5] (AXIS CHANGE).

When the desired page is displayed, move the cursor to the data which should be set or changed.

```
┌─────────────────────────────────────────────────┐
│ ┌─────────┐                                      │
│ │ ZERO SET│                                      │
│ └─────────┘                   ┌──────────────────┐
│                      *PROGRAM ZERO*  │97/07/15 14:10:00│
│                                      └──────────────────┘
│           NO.      X           Y           Z      1mm
│        *   1     10.000       0.000       0.000
│            2      0.000       0.000       0.000
│            3  ┌──45.000──┐    32.000       8.000
│            4     23.500    1230.000      456.200
│            5     32.580      -0.014        2.580
│            6      0.000       0.000        0.000
│            7      0.000       0.000        0.000
│            8      0.000       0.000        0.000
│            9      0.000       0.000      777.000
│           10      0.000       0.000        0.000
│
│                         X          Y          Z
│      ACT POSIT (WORK)  -260.000  -100.000   300.000
│                          A-Mtd
│     =S 45.
│     =S 32
│     =S B
│     =
│  ─────────────────────────────────────────────────
│  ┌────┬────┬────┬──────┬────┬────┬────┬───────┐
│  │SET │ADD │CAL │SEARCH│    │    │    │[EXTEND]│
│  └────┴────┴────┴──────┴────┴────┴────┴───────┘
└─────────────────────────────────────────────────┘
      (F1) (F2) (F3) (F4) (F5) (F6) (F7) (F8)
```

(3) Setting

(a) When the offset of the work zero (origin of work coordinate system) is known, press function key [F1] (SET) and enter that work zero offset. (The entry data is X_2 and Y_2 on the previous drawing.)

(b) When the work zero has been set and it serves as the reference of further offset, press function key [F2] (ADD) and enter the distance of the position to be set as viewed from the preset position.

(c) When it is necessary to set the work zero offset which causes the present actual machine position to be the new desired actual position, press function key [F3] (CAL) and enter the new actual position data. (In the previous drawing, key in X_4 and Y_4, to set X_2 and Y_2.) The entered data is displayed on the 21st line of the screen.

(4) Press the WRITE key.

This updates the data indicated by the cursor.

Note that a "∗" appears before the number of the work coordinate system which is currently selected.

[Supplement]   Local coordinate systems are effective only in programming.

5. Tool Data Set Commands

5-1. Tool Length Offset and Cutter Radius Compensation

Tool offset or compensation has been incorporated since the tip position is dependent on tool type.

Tool data consists of length offset data and cutter radius compensation data.

The system offers 50 sets of tool length offset and cutter radius compensation data as the standard feature, and this can be expanded to 100, 200, or 300 sets of offset and compensation data.

For offset number 0, only "0" can be set.

Data is set as follows:

(1) Press the TOOL DATA key.

This displays the tool data.

(2) One page of display covers 20 sets of tool length offset values and as many sets of cutter radius compensation values. If desired data is not seen on the page, operate the page keys or the [F4] (SEARCH).

```
┌─────────────────────────────────────────────────────────────┐
│ TOOL DATA SET                                                 │
│                                              97/07/15 14:10:00 │
│  *TOOL LENGTH OFFSET*  (H—)        *CUTTER R COMP*  (D—)    1mm│
│  NO.             NO.               NO.              NO.        │
│   1      1.000   11    0.000        1    10.000    11    0.000 │
│   2      2.000   12    0.000        2     5.000    12    0.000 │
│   3      1.000   13    0.000        3     2.320    13    0.000 │
│   4      1.000   14    0.000        4     0.000    14    0.000 │
│ * 5     10.000   15    0.000        5     0.000    15    0.000 │
│   6      0.000   16    0.000        6     0.000    16    0.000 │
│   7      0.000   17    0.000        7     5.000    17    0.000 │
│   8    122.432   18    0.000        8     0.000    18    0.000 │
│   9      0.000   19    0.000        9     0.000    19    0.000 │
│  10    889.499   20    0.000       10     0.000    20    0.000 │
│                              X          Y          Z          │
│  ACT POSIT (WORK)         -260.000  -100.000    300.000       │
│                              A-Mtd                            │
│  =S 5.                                                        │
│  =S 5.                                                        │
│  =S 10                                                        │
│  =                                                           │
│                                                              │
│  SET │ ADD │ CAL │ SEARCH │   │ ITEM↑ │ ITEM↓ │ [EXTEND]     │
└─────────────────────────────────────────────────────────────┘
    F1    F2    F3    F4    F5    F6    F7    F8
```

After obtaining the page displaying the desired tool data, position the cursor at the desired data element.

(3) Setting

(a) When the tool data is known, press function key [F1] (SET) and enter tool data on the keyboard.

(b) When the tool data has been set and the change amount from the set data is known, press function key [F2] (ADD) and enter the change amount.

(c) Tool length offset

Before carrying out the above steps (1) and (2), set the zero offset of the tool axis. Mount the tool for setting.

Manually align the tool tip with the reference surface.

Press function key [F3] (CAL) and key in the data, which consists of the axis name, the direction, and the current position as viewed from the origin. Note, here, that the axis name represents the axis parallel to the axis on which the tool rotates (X-, Y- and Z-axis).

Generally, the axis name is Z-axis.

Example: CAL   Z   10.5
          CAL   Z   −20.5

(4) Press the WRITE key.

The data is set at the position indicated by the cursor.

The tool offset number presently selected is identified by an asterisk (∗) appearing right before that tool offset number.

5-2. ATC Pot No./Tool No. Table

For vertical machining centers and other machines which have a small number of tools, the memory-random ATC specification is adopted and for machines with a large number of tools, the fixed address ATC specification is adopted.

5-2-1. Memory-random ATC Specification

⚠ CAUTION : In the memory-random ATC system, the tool set in the spindle is returned to the magazine toolpot of the tool to be set in the spindle next. Therefore, the correspondence between the tool number and the pot number will change each time the tool change cycle is carried out. This requires the initial correspondence between the tool number and the magazine pot number to be stored in the control memory after setting all the tools in the magazine.

In addition, since the use of a large-diameter tool will cause an interference with adjacent tools, the control must be capable of recognizing a large-diameter tool so that the toolpot which accommodates one is placed between the pots having dummy tools or no tool. That is, the large-diameter tool must be returned to the toolpot originally stored.

A large-diameter tool is distinguished from other conventional tools based on the machine specifications. Distinguishing of tools-large-diameter tools and conventional tools – is made according to the tool diameter and the value used to classify tools into these two categories depends on the machine specifications.

Setting of the correspondence between the tool number and toolpot number is made in the following two ways:

(1) The table listing the original correspondence between the tool numbers and toolpot numbers should be made beforehand on the display screen. After that, the tools are set in the toolpots in accordance with the correspondence table data.

(2) Tools are set in the spindle and they are returned to the magazine in the manual tool change operations. In this case, the toolpot where the tool in the spindle is to be returned may be specified or the one automatically selected may be used, as required.

Explanations in this section cover the procedure indicated in (1). For the procedure of (2), please refer to II, OPERATION, Section3, 4-1 "ATC".

Setting:

```
┌─────────────────────────────────────────────────────────────┐
│  TOOL DATA SET                              N         1       │
│                                          97/07/15 14:10:00    │
│                  * POT NO./ TOOL NO. TABLE *                  │
│                                                               │
│   POT TOOL    POT TOOL    POT TOOL   POT TOOL                 │
│   NO.  NO.    NO.  NO.    NO.  NO.   NO.  NO.                  │
│    1  001     11  011     21  NA             :SPCY POT    NA   │
│    2  002     12  012     22  022            :ACT TOOL   020   │
│    3  003     13  013     23  023            :NXT TOOL   007   │
│    4  004     14  014     24  024                             │
│    5  005     15  015     25  025            :MAGAZINE    10   │
│    6  006     16  016     26  026                             │
│    7  007     17  017     27  027                             │
│    8  008     18  018     28  028                             │
│    9  009     19  019     29  029                             │
│   10 [021]    20  NA      30  030                             │
│                                                               │
│                                                               │
│   SET          SEARCH         ITEM↑  ITEM↓  [EXTEND]          │
└─────────────────────────────────────────────────────────────┘
     F1    F2    F3    F4    F5    F6    F7    F8
```

Fig. 1-5 Tool Data Offset (ATC Pot No./Tool No. Table)

(1) Press the TOOL DATA key.

(2) Press function key [F7] (ITEM ↓).

   The CRT will display the page of "*POT NO./TOOL NO. TABLE*".

(3) Locate the cursor at TOOL NO. position of the POT NO. for which the tool number is to be set.

(4) Press function key [F1] (SET).

   "S ⌴ " will be displayed on the console line. (" ⌴ " indicates the space.)

(5) Key in the desired tool number through the keyboard.

   (a) Conventional tool   S ⌴ 1

   (b) Large-diameter tool   S ⌴ 7, L

   (c) Dummy tool   S ⌴ D

   (d) For clearing tool number   S ⌴ *

   (e) Planer tool (optional)   S ⌴ 6, P

   (f) Heavy tool (depending on the machine type) S ⌴ 5, M

   (g) Press the WRITE key.

   This sets the correspondence between the tool number and the toolpot number.

[Supplement]   1.   The symbols "→" and ">>" appearing in the POT NO. column indicate the position of the corresponding toolpot.

→   :   Tool change position with A TC

>>   :   Manual tool change position

2.   The range of toolpot numbers which can be set is from "1" to the magazine capacity (the number of tools accommodated in the magazine).

The usable number of tool numbers is identical to the programmable number of tool offset numbers.

3.   For a large-diameter, setting is allowed only when the two adjacent toolpots are assigned with no tool number or dummy tool code, "D". An error occurs if either of two adjacent toolpots is assigned with an actual tool number.

The setting of a large-diameter tool in a toolpot automatically sets dummy tool code, "D", at two adjacent toolpots.

4.   In case the tool number already used is again entered for a new toolpot, an alarm occurs.

5.   Any attempt to set a tool number for a toolpot which has a dummy tool in it causes an error. In this case, cancel the dummy tool code by entering "✳" code. Note that the dummy tools placed in the adjacent toolpots for the one accommodating a large-diameter tool should not be cleared. A dummy tool may be placed between two large-diameter tools as a common dummy tool.

5-2-2. Fixed Address ATC Specifications

To select a tool from the tools in the magazine using a tool number command, it is necessary to set the correspondence between the tool numbers and the toolpot numbers.

If an interference occurs between the tools stored in the two adjacent toolpots due to their diameters, set the safety adaptor in toolpots at both sides of the toolpot which holds a large-diameter tool. This prevents an occurrence of interference between the tools during tool setting operations.

Setting:

```
┌─────────────────────────────────────────────────────────────────┐
│ TOOL DATA SET                                    N         1      │
│                                          97/07/15 14:10:00        │
│                      * POT NO./ TOOL NO. TABLE *                  │
│                                                                   │
│   POT TOOL   POT TOOL    POT TOOL    POT TOOL                     │
│   NO.  NO.   NO.  NO.    NO.  NO.    NO.  NO.                      │
│    1 001     11 011      21  NA                   :RET TOOL   NA   │
│    2 002     12 012      22 022                   :ACT TOOL   020  │
│    3 003     13 013      23 023                   :NXT TOOL   007  │
│    4 004     14 014      24 024                                   │
│    5 005     15 015      25 025                   :MAGAZINE   10   │
│    6 006     16 016      26 026                                   │
│    7 007     17 017      27 027                                   │
│    8 008     18 018      28 028                                   │
│    9 009     19 019      29 029                                   │
│   10 [021]   20  NA      30 030                                   │
│                                                                   │
│                                                                   │
│                                                                   │
│  SET   │    │    │ SEARCH │    │ ITEM↑ │ ITEM↓ │ [EXTEND]         │
└─────────────────────────────────────────────────────────────────┘
    F1    F2    F3    F4    F5    F6    F7    F8
```

(1) Press the TOOL DATA key.

(2) Press function key [F7] (ITEM ↓).

The CRT will display the page of "*POT NO./TOOL NO. TABLE*".

(3) Locate the cursor at the TOOL NO. position of the POT NO. for which the tool number is to be set.

(4) Press function key [F1] (SET).

The CRT will display "S" on its console line. (⌴ indicates the space.)

(5) Key in the desired tool number through the keyboard.

    (a) Tool number   S ⌴ 1

    (b) Safety adaptor   S ⌴ D

    (c) For clearing tool number   S ⌴ *

    (d) Planer tool (optional)   S ⌴ 6, P

    (e) Heavy tool   S ⌴ 1, M

(6)   Press the WRITE key.

This sets the correspondence between the tool number and the toolpot number.

[Supplement]   1.   If the tool number already used is again entered for a new toolpot, an alarm occurs.

2.   An attempt to set a tool number for a toolpot which has a dummy tool in it causes an error. In this case, cancel the dummy tool code by entering "✻" and set.

5-2-3.   Clearing/Setting Tool Numbers

It is possible to clear tool numbers set on the "✻POT NO./TOOL NO. TABLE✻" screen at one time, or to set tool numbers which correspond to toolpot numbers (having same numbers) on the "ATC POT NO./TOOL NO. TABLE" at one time.

Note tool numbers can not ben set at one time unless the correspondence between toolpot numbers and tool numbers are all cleared.

Operating Procedure

(1)   Clearing tool numbers

(a)   Press the TOOL DATA key.

(b)   Press function key [F7] (ITEM ↓).

The CRT will display the page of "✻POT NO./TOOL NO. TABLE✻".

(c)   Key in "ATC" through the keyboard and press the WRITE key.

The following message will be displayed on the screen.

Tool table initialize OK? (Y/N)

(d)   Key in "Y" and press the WRITE key to clear tool numbers.

Key in "N" and press the write key to cancel clearing tool numbers.

(2)   Setting tool numbers

(a)   Press the TOOL DATA key.

(b)   Press function key [F7] (ITEM ↓).

The CRT will display the page of "ATC POT NO./TOOL NO. TABLE".

(c)   Key in "TSET" and press the WRITE key.

[Supplement]   When an attempt is made to set tool numbers without clearing the correspondence between toolpot numbers and tool numbers, no data will be set.

6.   Parameter Setting

To perform NC operations, for example positioning, program editing, and so forth, data such as axis travel ranges, tape output code, and others, are predetermined for each individual function. However, there are cases in which these data needs to be changed in accordance with the change in operation conditions.

The data elements used to control NC functions are called parameters.

The parameters can be classified into 18 types as indicated below.

(1)   Display selection (selection for display/not-display of parameter setting screens)

(2)   Common variable

(3)   User parameter

(4)   G/M code macro

(5)   NC optional parameter (long word)

(6)   NC optional parameter (word)

(7)   NC optional parameter (bit)

(8)   Input unit system

(9)   NC optional parameter – RS232C (CN0:)

(10) NC optional parameter – Spindle (OKUMA VAC)

(11) Machine axis parameter

(12) System parameter

(13) Pitch error compensation data

(14) NC run timer

(15) Spindle load monitor

(16) Tapping torque monitor

(17) Machine user parameter

(18) Machine system parameter

Contents and setting procedures of parameters are detailed in IV "PARAMETER".