SECTION 5 AUTOMATIC MODE OPERATION

1. Main Program Selection and Operation

In order to execute a part program in the automatic mode, first select the part program. Programs stored in the memory have their own file names since programs are managed as files. The specified main program is read from the part programs stored in the file and the subprogram, called in the main program, is searched out in the specified sub file to be loaded to the NC.

For the selection of file, the directory-selection-based file operation screen is used.

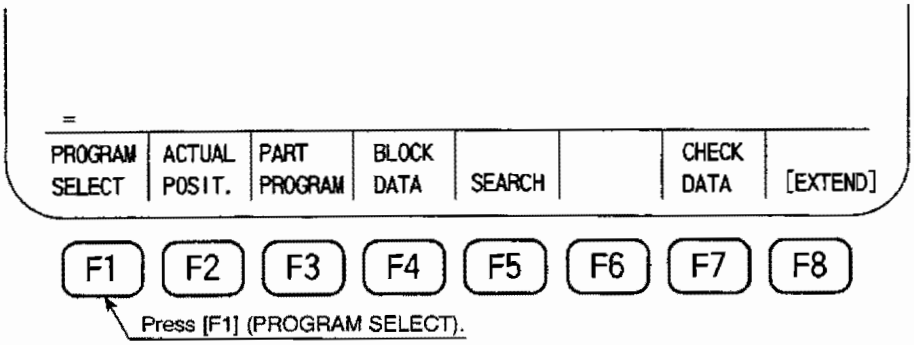
The following explanation gives basic information on program selection operation. In addition to the basic information given below, there are various functions including the function to display the registered part program files in batch. For details of the functions, refer to III. DATA OPERATION, Section 2, 15. "Directory-Selection-Based File Operation Function".

The operating procedure is described below:

(1) Press the AUTO key.

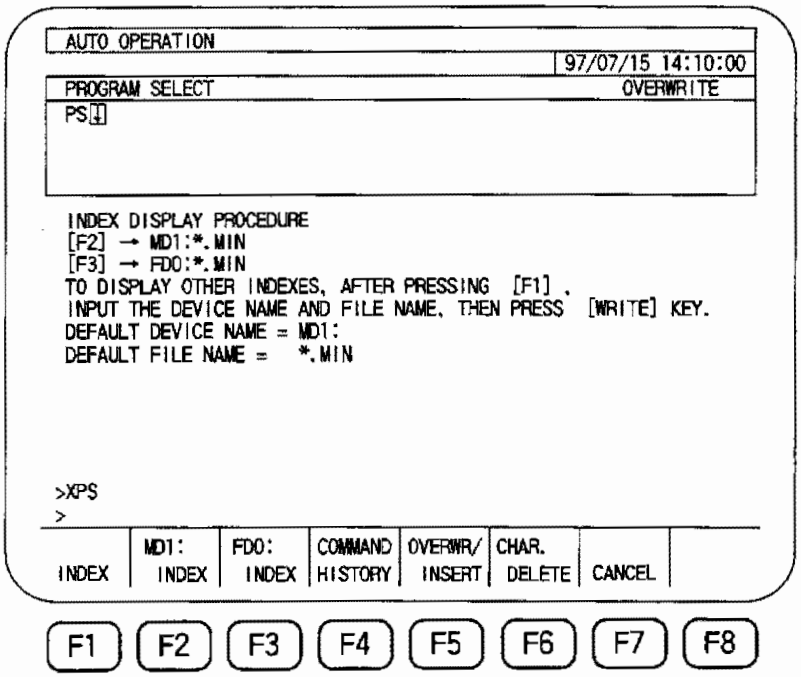
(2) The lamp at the upper left corner in the key lights and the screen changes to the automatic operation screen.

Press function key [F1] (PROGRAM SELECT).



The screen changes to the auto operation screen and the following is displayed on the screen.

PROGRAM SELECT
PS



(3) Enter the designation mode from the table below.

The input format is as indicated below and entry of an asterisk (*) instead of a file name, will display a file name directory.

=PS␣ main-file-name, main-program-name, sub-file-name; option

Main-file-name ..... File name of main programs

Sub-file-name ..... File name of sub programs which are called from a main program

Option ..... Designation of A, B and S option

Procedure	Designation Mode	Contents	Remarks
(a)	PS␣*	Designates main file name.	Designation of device name for calling out main file is also possible.
	PS␣ *;	Designates main file name and option.	Same as above.
(b)	PS␣*,*	Designates main file name and main program name.	Same as above.
	PS␣ *,*;	Designates main file name, main program name and option.	Same as above.
(c)	PS␣ ,*,*,*	Designates main file name, main program name and sub file name.	Designation of device name for calling out main and sub file is also possible.
	PS␣*,*,*;	Designates main tile name, main program name, sub file name and option.	Same as above.

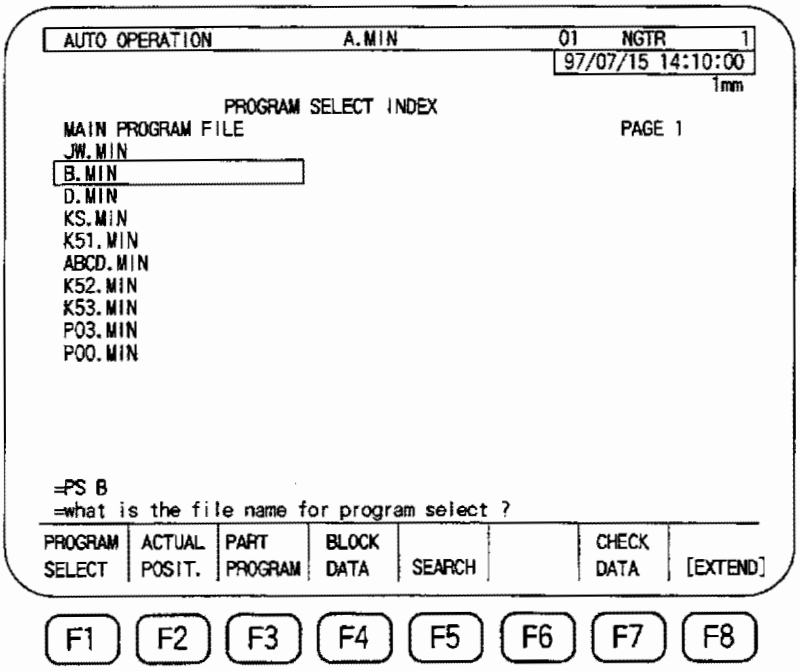
Procedure	Designation Mode	Contents	Remarks
(d)	PS␣*,,*	Designates main file name and sub file name.	
	PS␣ ,*,*	A.MIN is automatically selected as main file name. Designates main program name and sub tile name.	

(a) Designation mode PS␣ * (or PS␣ *;)

1) Key in as PS␣ * or PS␣ *;.

2) Press the WRITE key.

The display is changed to the PROGRAM SELECT INDEX screen and main file names registered are displayed. (This searches files having extension "MIN" from MD1:.)



3) Position the cursor at the desired file name.

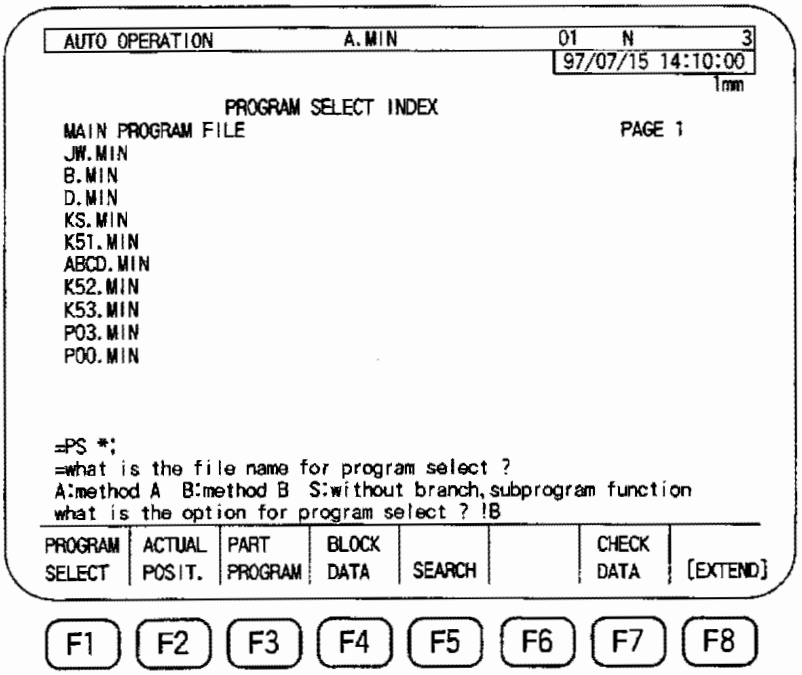
4) Press the WRITE key.

5) If there is an option designation ";", the screen automatically goes to the option designation mode and the messages below are displayed at the lower block on the display screen.

A:method A B:method B S:without branch, subprogram function

What is the option for program select? !

Key in "A", "B", or "S" as desired. (Option B and option S can be specified simultaneously.)



6) Press the WRITE key.

Entry of "B" in response to the prompt "What is the option for program select? !" selects the operation method B (large capacity operation method). If the control is not supported by this operation method, the entry is ignored.

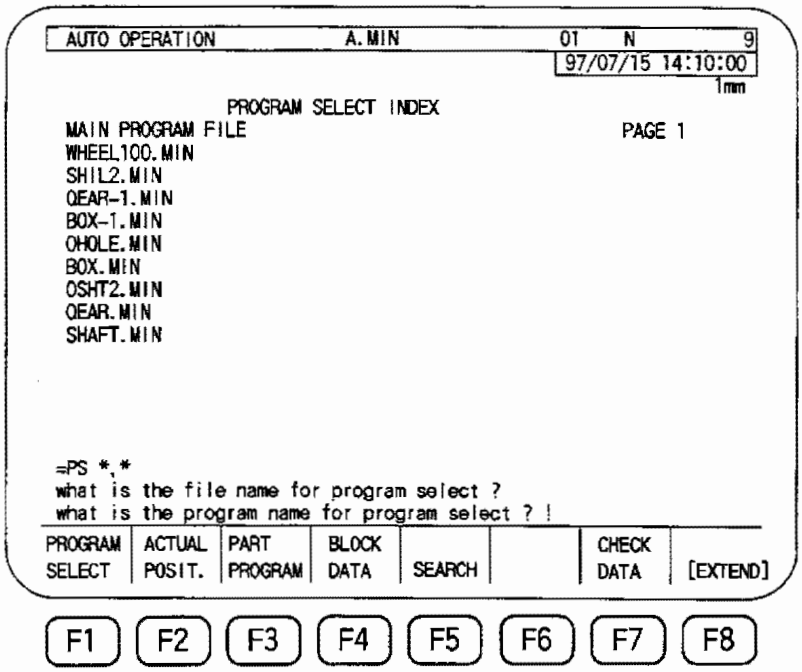
When the WRITE key is pressed without entering any character in response to the prompt "What is the option for program select? !", the control operates in accordance with the setting of the NC optional parameter (word) No. 11.

(b) Designation mode PS␣ ∗,∗ (or PS␣ ∗,∗;)

1) Key in as PS␣ ∗,∗ or PS␣ ∗,∗;.

2) Select main file names in the same procedures as 2), 3) and 4) in (a).

3) Enter the main program name when the prompt "What is the program name for program select? !" is displayed at the lower section of the screen.



For example, selecting main program name O100 is as follows:

What is the program name for program select? !O100

4) Press the WRITE key.

If the WRITE key is pressed without entering the program name, the first program in the main file is selected.

5) When there is an option designation ";", the display screen allows the entry of option designation code. Follow the steps 5) and 6) in (a).

(c) Designation mode PS␣ *,*,* (or PS␣ *,*,*;)

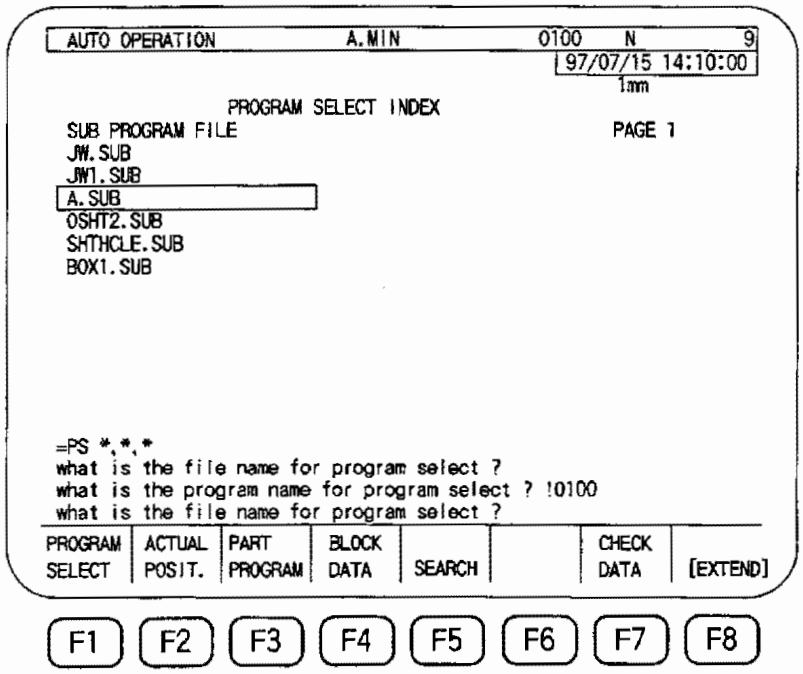
1) Key in as PS␣ *,*,* or PS␣ *,*,*;.

2) Select the main file name in the same procedures as 2), 3) and 4) in (a).

3) Select the program name in the same procedures as 3) and 4) in (b).

4) The screen will then display the subprogram file names.

Files having extension "SUB" in MD1:.



5) Position the cursor at the desired file name.

6) Press the WRITE key.

7) If there is an option designation ";", the display screen allows the entry of option designation code. Follow the steps 5) and 6) in (a).

(4) By pressing the WRITE key, the main program can be read from the specified main file, while the subprogram called up in the main program is loaded to the NC, and the main program is displayed on the display screen.

If the subprogram called up in the main program cannot be found in the specified sub file, it should be searched for in the sub file of the extension SSB so that loading can start. If it still cannot be found, an error occurs.

(5) Press the CYCLE START switch.

By pressing the CYCLE START switch, the main program can be started.

[Supplement]

1. If a main file name is omitted, A.MIN is used. If the main program is omitted, the first program in the main file name is used.
2. Search of the subprogram which has been stored as a part of the main file is made first.
3. When the sub file name is omitted, the search of a subprogram called in the main program is made only for the sub file of the extension SSB or MSB. Therefore, the sub file with extension .SUB should be input without fail. Only one kind of sub file can be input.

   (If the subprogram in the main file calls the other subprogram, the subprogram to becalled must be stored after the one from which it is called.)
4. If there is no specified file name or program name, an error occurs. Then, the program selected previously becomes invalid. Always confirm that the valid file name or program name is displayed on the first line of the display screen.
5. A program once selected is valid until the next program is selected. Selecting the schedule program is invalid.
6. Direct specification of the file name without using symbol "*"is also allowed.
7. Main and sub file name directory can be searched for using alphabetic character, "-", "?", or "*".
8. An asterisk (*) is displayed at the beginning of the file name of the file which is selected currently.

   When the PROGRAM SELECT INDEX screen is displayed, the cursor is positioned on the file name prefixed by an asterisk.
9. When there is no file where asterisk should be set, the first page of the PROGRAM SELECT INDEX screen is displayed with the cursor at the top of the file names.
10. An asterisk (*) is not displayed in program selection such as external program selection, DNC-C program selection, and PPC program selection, other than the selection made by an operator.

Table 5-1 Operation Comparison between Normal Storage Capacity Memory and Large Storage Capacity Memory

<table>
<tr><th colspan="2" rowspan="2">Item</th><th rowspan="2">Selection and Operation of Normal Storage Capacity</th><th colspan="2">Selection and Operation of Large Storage Capacity</th><th>Remarks</th></tr>
<tr><td colspan="2">Method B</td><td>Method S</td></tr>
<tr><td colspan="2" rowspan="2">Specification of S option in PROGRAM SELECT command mode</td><td rowspan="2">Invalid</td><td colspan="2">Valid</td><td>—</td></tr>
<tr><td>S option not specified</td><td>S option specified</td><td>—</td></tr>
<tr><td rowspan="4">Program size limitation</td><td>Main program</td><td rowspan="4">Up to the operation buffer area size.*</td><td colspan="2">Up to total length of the stored main program</td><td>Same as Method B</td></tr>
<tr><td>Subprogram</td><td rowspan="3">Total tape length varies depending on the selected operation buffer area capacity.*</td><td>—</td><td>—</td></tr>
<tr><td>Library program</td><td rowspan="2">Total tape length varies depending on the selected operation buffer area capacity.*</td><td>Same as Method B</td></tr>
<tr><td>Schedule program</td><td>Same as Method B</td></tr>
<tr><td colspan="2">Subprogram function</td><td>Available</td><td>Available</td><td>Unavailable (alarm)</td><td>Same as Method B</td></tr>
<tr><td colspan="2">Branch function</td><td>Available</td><td>Available</td><td>Unavailable (alarm)</td><td>Same as Method B</td></tr>
<tr><td rowspan="4">Instruction for jump destination of branch instruction</td><td>Main program</td><td rowspan="4">Sequence label, sequence number</td><td>Sequence label only</td><td>—</td><td>—</td></tr>
<tr><td>Subprogram</td><td rowspan="3">Sequence label or sequence number</td><td>—</td><td>—</td></tr>
<tr><td>Library program</td><td>—</td><td>—</td></tr>
<tr><td>Schedule program</td><td>—</td><td>—</td></tr>
<tr><td colspan="2">Main program sequence label limit</td><td>No limit</td><td>Fewer than 30 pcs.</td><td>No limit</td><td>Same as Method B</td></tr>
<tr><td colspan="2">Execution time for PSELECT command</td><td>Several tens seconds to several minutes</td><td>Several tens seconds to several minutes</td><td>Ends at once</td><td>Same as Method B</td></tr>
</table>

*: This capacity can be extended by selecting the operation buffer expansion specification.

2. Schedule Program Selection and Operation

(1) The schedule program function is provided to continuously machine different types of workpieces automatically using the pallet changer, etc. without operator's intervention.

In this item, selection and operation of the schedule program are explained.

For the programming of the schedule program, refer to Section 12, 3. "Schedule Programs" in Programming Manual.

(2) For the selection of a schedule program, the directory-selection-based file operation screen is used.

The following explanation gives basic information on selection and registration of the schedule program. In addition to the basic information given below, there are various functions including the function to display the registered part program files in batch. For details of the functions, refer to III. DATA OPERATION, Section 2, 15. "Directory-Selection-Based File Operation Function".

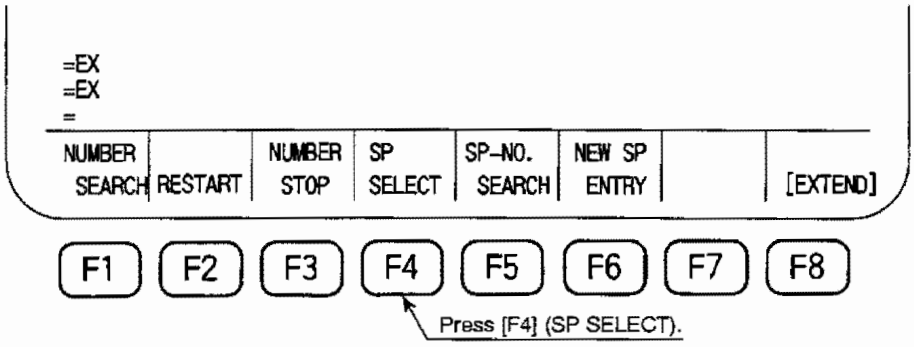
2-1. Selection and Operation of Schedule Program

(1) Press the AUTO key.

(2) The lamp at the upper left corner in the key lights and the screen changes to the automatic operation screen.

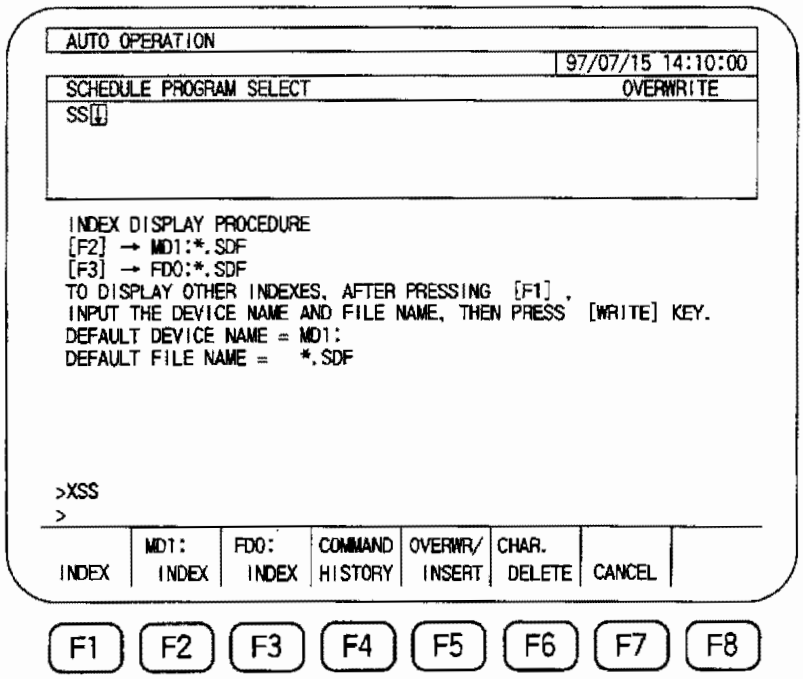
(3) Press function key [F8] (EXTEND) two times.

(4) Press function key [F4] (SP SELECT).



The screen changes to the auto operation screen and the following is displayed on the screen.

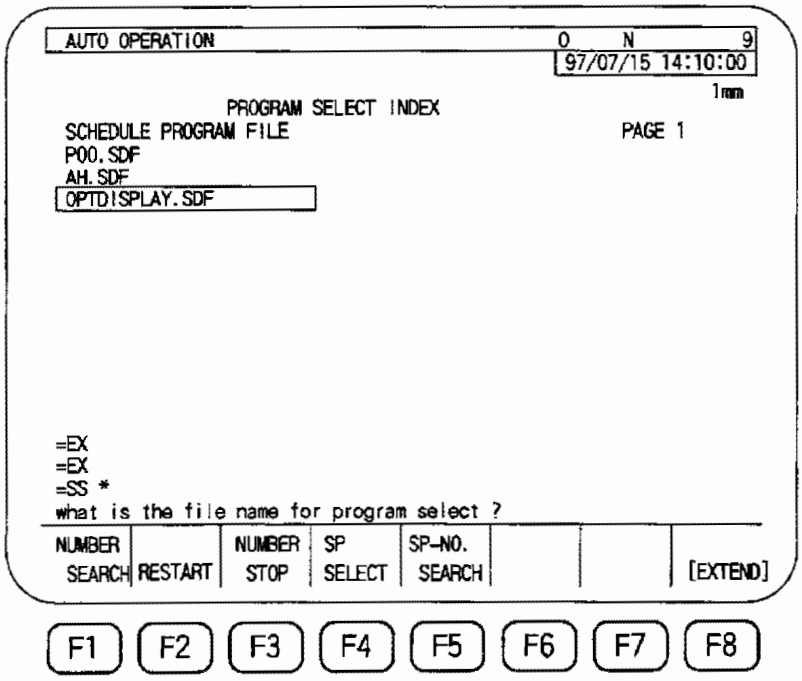
SCHEDULE PROGRAM SELECT
SS



(5) Enter "*" following "SS".

(6) Press the WRITE key.

The display will be switched to the PROGRAM SELECT INDEX page and the schedule program file names registered are shown. (Files having an extension of "SDF" are searched from the MD1:.)



(7) Position the cursor at the desired file name.

(8) Press the WRITE key.

The schedule program is selected and the NC enters the schedule operation mode.

(9) Press the CYCLE START switch.

This starts the continuous operation in accordance with the programmed schedule.

When the schedule operation cycle stop key on the machine operation panel is pressed, the NC enters the cycle stop mode. In this mode, operation cycle stops after the execution of a main program. To resume the operation, press the CYCLE START switch.

[Supplement]

1. Selection of a schedule program file by directly keying in the file name is also possible.

   SS ␣ schedule-program-file-name [WRITE]

2. Main and sub file name directory can be searched for using alphabetic character, "-", "?", or "*".

3. Schedule program selection should be done only after resetting the NC. If the schedule program is selected during operation, an error will occur.

4. When the normal automatic operation (AUTO mode operation by main program selection) is done after selecting the schedule program, the program should be selected again.

5. When the CYCLE START switch is pressed with the SINGLE BLOCK switch set ON in schedule program operation mode, the main program will be selected by the schedule program and the machine will wait in the start state. Then, if the CYCLE START switch is pressed, the machine returns to the normal single block mode state.

   But the machine will not stop in the blocks containing VSET, IF and GOTO instructions.

6. When the RESET switch is pressed during the operation in accordance with a schedule program, the part program selected when the NC has been reset will be executed again from the start if the CYCLE START switch is pressed.

   If the repetition number of the part program is specified in the program block selection block in the schedule program, the program execution stopped during machining will not be counted.

7. When the CYCLE START switch is pressed after; selecting the schedule program, the main program is first selected and machine operation using the selected main program begins after the completion of main program selection. If the control is reset while a main program is being selected, the main program is not selected.

8. The main program executed in the schedule program operation mode is cleared from the operation buffer after the completion of the program execution.

9. An asterisk (*) is displayed at the beginning of the file name of the file which is selected currently.

   When the PROGRAM SELECT INDEX screen is displayed, the cursor is positioned on the file name prefixed by an asterisk.

[Supplement]

10. When there is no file where asterisk should be set, the first page of the PROGRAM SELECT INDEX screen is displayed with the cursor at the top of the file names.
11. An asterisk (*) is not displayed in program selection such as external program selection, other than the schedule program selection made by an operator.

3. Cycle Start and Slide Hold

(1) Cycle Start

Press the CYCLE START switch on the machine operation panel to start the NC operation with either the selected part program or the one-block program entered in the MDI mode.

(a) Cycle start after NC reset:

This is effective during automatic operation or MDI operation. The program is read and executed for each separate mode, cycle start requires the following conditions.

[In automatic operation]

The schedule program or the main program has been selected correctly.

[In MDI operation]

The one-block instruction has been entered in the MDI buffer.

(b) Cycle start after shutdown by single block or program stop:

The next block can be executed by pressing the CYCLE START switch in automatic mode.

(c) Cycle start in slide hold mode:

When the CYCLE START switch is pressed, function generation which was interrupted, begins again.

[Supplement]

1. Press and release the CYCLE START switch to begin the operation, but when the machine is stopped temporarily due to the activation of the SLIDE HOLD switch, cycle start is made when the CYCLE START switch is only pressed.
2. Pressing the CYCLE START switch during the program selection, sequence number searching and return search is ineffective.
3. While the SLIDE HOLD switch is being pressed, the CYCLE START switch is inoperative.
4. During the operation, the RUN lamp on the NC operation panel comes on, excluding the slide hold mode.
5. Even in the data setting mode, cycle start is possible provided that the mode previously selected is auto or MDI and the setting of bit 5 of NC optional parameter (bit) No. 2 is "1".
6. When alarms (P, A, B, and C) are on, cycle start by pressing the CYCLE START switch is impossible.
7. An error occurs when the CYCLE START switch is pressed after return search has been executed. In this case, cycle start is possible by pressing the SEQUENCE RESTART switch.

(2) Slide Hold

By pressing the SLIDE HOLD switch on the machine operation panel while the machine is operating as initiated by pressing the CYCLE START switch, explained in item (1) above, axis feed is suspended or program execution is stopped.

(a) Slide hold means the NC halt is made during axis movement. Start-up hold means the NC halt made before or after the completion of an axis movement.

(b) Slide hold during function generation:

- During axis movement by rapid feed or cutting feed

Axis movement stops after deceleration. When the axis stops halfway in a commanded axis travel, it is in a slide hold state. If the axis reaches the target point before it is stopped, or the operation mode stops after the execution of the other commands in that block are completed, then this is the start-up hold state, and is the same as a stop in the single block or program stop.

- During dwell

Dwell immediately stops and the machine is brought to the slide hold state.

(c) During operation, excluding function generation
(during the execution of miscellaneous commands):

The slide hold is not effective for miscellaneous functions but the SLIDE HOLD lamp stays on. The machine is brought to a start-up hold state, since the operation halts after the execution of miscellaneous functions.

[Supplement] 1. When axis movement and miscellaneous functions are in the same block, there are two cases for the execution order of the commands.

a) With ..... The execution of axis movement commands and auxiliary functions start simultaneously.

b) After ..... The axis movement is completed the execution of miscellaneous function starts.

Depending on the above conditions,

if machine hold occurs during axis movement
(including dwell) ..... Slide hold

if the machine halts at a time other than
axis movement ..... Start-up hold

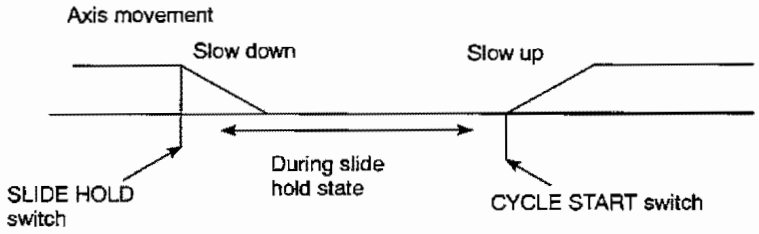
2. During the slide hold (excluding start-up hold), the SLIDE HOLD lamp on the NC operation panel comes on (although the slide hold state is a part of the machine operating state, the RUN lamp will go off).

[Supplement] 3. If the machine is brought to the start-up hold state by the activation of the SLIDE HOLD switch, both the SLIDE HOLD and the RUN lamp go off.

CAUTION : Cycle start in slide hold is activated once the CYCLE START switch is pressed. (In the start-up hold state, cycle start is activated when the pressed CYCLE START switch is released.)

Examples of slide hold:

[During axis movement]



[During miscellaneous function execution]

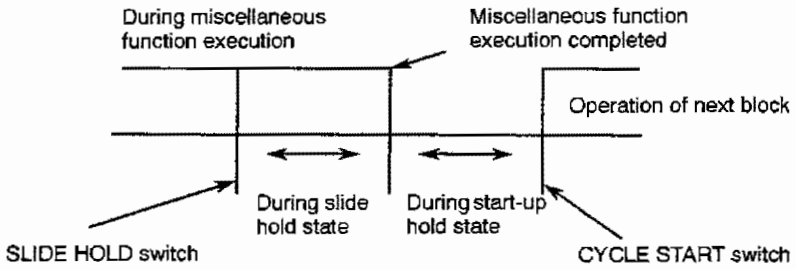
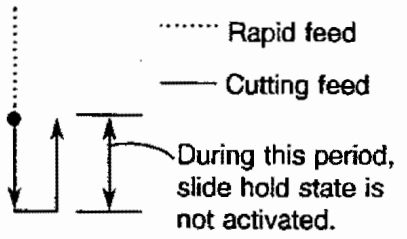


Fig. 5-2 Examples of Slide Hold

4. Even if the SLIDE HOLD switch is pressed during the execution of a tapping cycle – G84 (tapping cycle) and G74 (reverse tapping cycle), the tapping cycle is not interrupted. Slide hold is activated after the completion of the tapping cycle. The machine will not stop operation, but after the operation is completed, it stops operation temporarily. Note that the synchronized tapping cycle (G284 and G274) is different from above described tapping cycle. For details, refer to SPECIAL FUNCTIONS MANUAL No. 1, "SYNCHRONIZED TAPPING".



5. The slide hold function can be activated and deactivated by the programmed M codes, M140 and M141.

   M140 : Slide hold ineffective

   M141 : Slide hold effective

   The SLIDE HOLD switch is inoperative for the blocks containing M140 and M141, and the slide hold function is not activated even when the SLIDE HOLD switch is pressed during such period.

   Note that the control is in the M141 mode after it is reset.

4. Resetting NC

The NC reset means initializing the internal NC status.

The NC system is reset when:

(1) the RESET switch on the machine operation panel is pressed.

(2) the external reset signal is input.

(3) The MACHINE LOCK key on the machine operation panel is turned on or off.

(4) the operation mode is changed over from MANUAL mode to AUTO or MDI mode by pressing the AUTO or MDI key on the NC operation panel.

(5) the operation mode is changed over from AUTO or MDI mode to MANUAL mode by pressing the AUTO or MDI key on the NC operation panel.

(6) When the operation mode is changed from MANUAL mode to DATA SET mode by pressing the DATA SET key on the NC operation panel, the system is not reset; however, when the mode is changed to AUTO or MDI next, the system is reset.

The state that the NC is reset by the change of the operation mode is called mode reset.

NC resetting operation:

The NC resetting operation stops the machine operation immediately and the NC system is initialized at the same time.

[Supplement]
1. When the RESET switch is pressed during axis movement, the machine slows down and then stops axis movement. The actual reset is done after axis movement has stopped.
2. Even if the RESET switch is continuously pressed, reset is done only one time.

5. Sequence number Search and Mid-Start

Sequence search is used to start the operation from a required sequence of a main program. The specified sequence is searched by the sequence name or the cursor, and then the operation is started from the searched sequence by pressing the CYCLE START switch.

The operating procedure for the sequence number search is described below.

Sequence Number Search by Sequence Name or No. of Blocks

When the main program is selected correctly in automatic mode, perform the following operation.

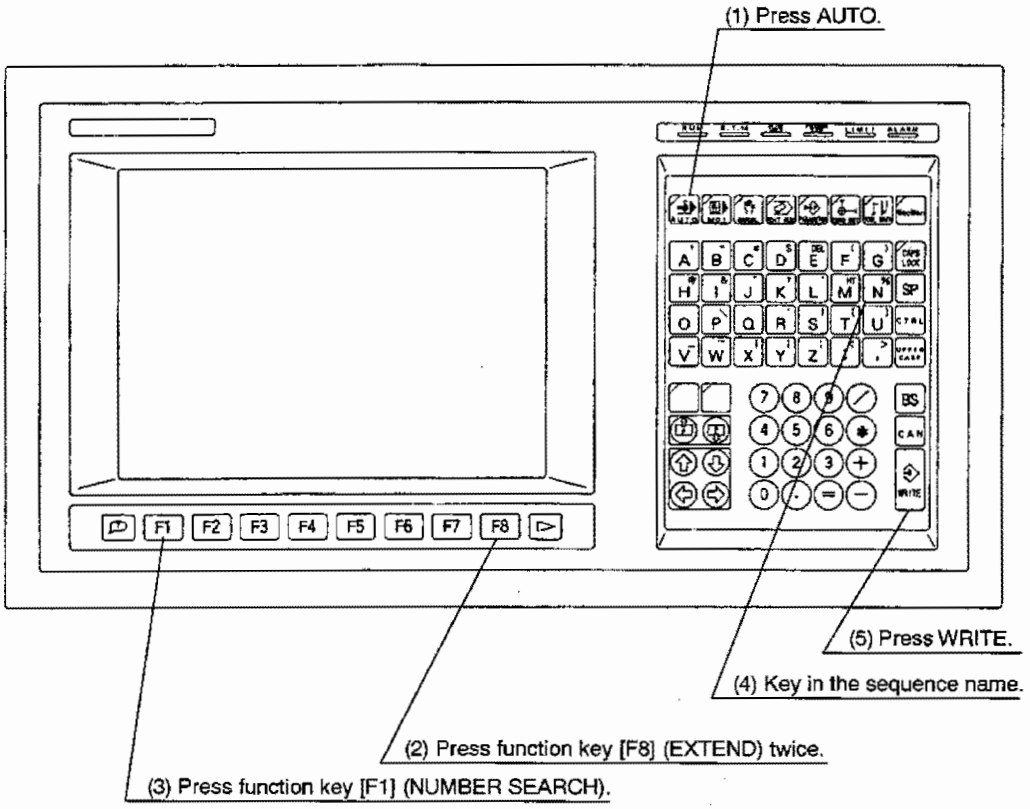


Fig. 5-3 Sequence Search by Sequence Name or No. of Blocks

(1) Press the AUTO key.

(2) Press function key [F8] (EXTEND) twice.

Function names such as [F1] NUMBER SEARCH will be displayed.

(3) Press function key [F1] (NUMBER SEARCH). "NS" is displayed on the 21st line of the display screen.

(4) Enter the specified sequence name or the required number of blocks.

(5) Press the WRITE key.

The sequence name is searched for from the main program head currently selected, and the sequence pointer moves to the found sequence name position. When the number of blocks has been keyed in, the search is made in the specified number (either positive or negative) of blocks from the currently located pointer position.

The following cases result in errors:

(a) When the specified sequence name is not found in the program.

(b) While the schedule program is executed (from schedule program start-up to the end).

(c) Whether the search is made or not during the execution of a main program can be selected by setting proper data at bit 3 of NC optional parameter (bit) No. 4.

(d) When the main program is not selected correctly.

[Supplement] In cases (b) and (c), the sequence number search can be executed after resetting the NC.

Sequence Number Search by Cursor

In the automatic mode, the sequence number pointer may be moved as desired by the cursor keys when the main program has been selected correctly and the screen displays the program.

⇩ ..... one sequence advance

⇧ ..... one sequence return

In the following cases, the cursor keys are inoperative:

- While the schedule program is being executed
- When the display screen is not the program display
- When the sequence pointer leaves the selected main program by the cursor key operation

Restart after Sequence Number Search

Program execution starts from the sequence identified by the sequence pointer when the CYCLE START switch is pressed.

Since programmed commands which were not read during the sequence number search are not valid, a model status must be set as needed by entering the necessary commands from the keyboard so that the actual status and the programmed status match.

[Supplement]

1. During the sequence number search, the read pointer is moved, while the modal instruction value and coordinate instruction value are disregarded. Subprogram CALL and GOTO are not done.
2. The sequence number search is used for start-up after a pause during the machining work, while the return search is used for the return to the block while the machining is underway.
3. The number of the sequence repetition cannot be specified during sequence number search.
4. Additionally, the optional block skip does not affect the sequence number search.

6. Return Search and Sequence Restart

When the machining cycle is interrupted during automatic operation because of tool breakage or other troubles, this function is used to restart the operation after necessary measures such as tool replacement have been taken.

After locating the sequence pointer to the specified sequence by return-search-operation, in which the commands are processed in the CPU, press the SEQ. RESTART switch. This positions the axes at the point commanded last at the manual cutting feedrate. Press the CYCLE START switch, then the operation will be resumed from the same sequence.

6-1. Return Search

When the main program is selected correctly in the automatic mode, carry out the steps following:

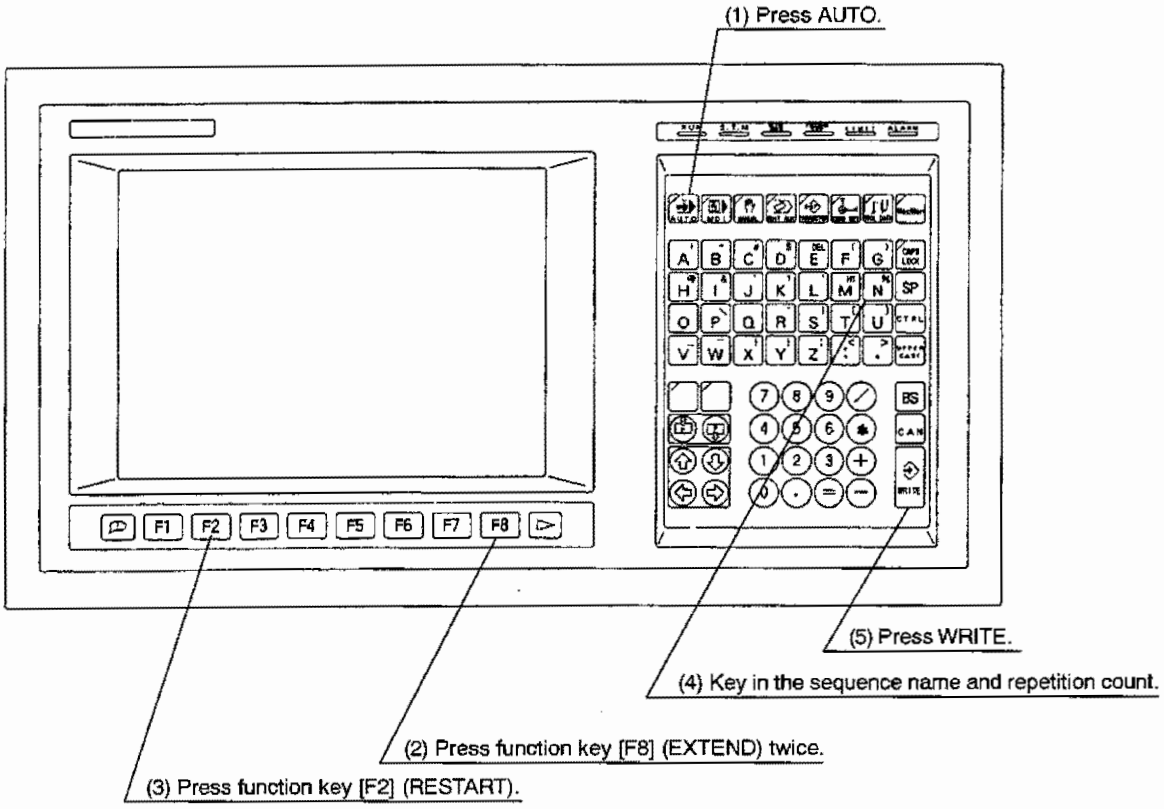


Fig. 5-4 Return Search and Sequence Restart

(1) Press the AUTO key.

(2) Press function key [F8] (EXTEND) twice.

Function name "RESTART" is displayed for function key [F2].

(3) Press function key [F2] (RESTART).

"RS" is displayed on the 21st line on the display screen.

(4) Enter the sequence name and repetition count, or the block count.

Input format is as follows:

= RS␣ sequence-name, repetition-count or block-count-value

When the sequence name is specified, the repetition count must be less than 9999. If entry of the repetition counter is omitted, it is regarded as "1".

When the block count is specified, the count value must be less than 99999999.

For entering a block count, the relative number may be given as indicated below.

Example:

592 ..... 592nd block

* ..... Block count value which was counted at NC reset

*–2 ..... The block two blocks ahead of the block count value above

(5) Press the WRITE key.

The return operation is executed up to the specified sequence.

Restarting operation refers to the operation in which all the commands are processed within the control, without giving output signals for axis motions, and S, T, M and B functions. CALL command, RTS command and coordinate system shifting are also processed.

The block count is the count of the sequence executed from the program start after reset. Control statements such as GOTO, CALL, etc. are not counted. The count value and sequence name are not cleared by NC reset or by turning the power source on or off, and can therefore, be used for return after reset. They are cleared when the operation begins.

[Supplement]

1. If return search operation is intended while a schedule or a main program is being executed, an error occurs.
2. The return operation up to M02 of the program is possible by the [F2] (RESTART) ␣ E [WRITE] key operation. Reset by M02 (or M03) is not carried out.

6-2. Sequence Restart

This function can be used only after the return search. (For the return search operation, refer to 6-1. "Return Search" in this section.)

When the SEQ. RESTART switch on the machine operation panel is pressed, the program status up to the specified block is returned automatically.

(1) Automatic Restoration of Miscellaneous Functions

(a) Restoring the last S code

If there is an S command, the S code of that command is executed unconditionally.

(b) A T code is not restored automatically.

Since a T code (next tool command) is not restored automatically, it is necessary to set the correct tool number if the next tool number presently active is incorrect. If the next tool number is "0", input the tool number in the MDI operation before executing search.

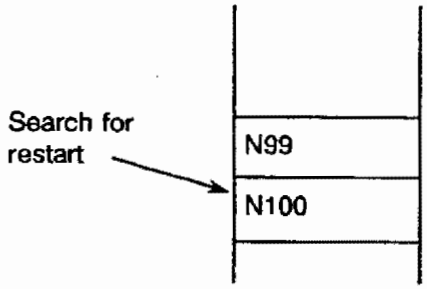
(c) Only M codes related to spindle operation can be restored.

Since M codes are processed in groups, the last modal state of the individual M code groups is restored.

- M03, M04, M05, and M19 are regarded as M codes in the same group, and the last state of them is restored.
- M codes related to ATC operation (M06, M63, M64, and M65) and APC operation (M60, etc.) are not restored automatically. The operator must restore the correct status before restarting the operation.
- One-shot M codes (M00, M01, etc.) are not restored.

(d) The axes return to the point programmed in the return sequence at a manual cutting feedrate.

(e) After axes have reached the return point in the sequence return operation, the operation stops at that point independent of SINGLE BLOCK key setting. To resume the operation, the CYCLE START switch must be pressed.



1) Key in N100 after pressing function key [F2] (RESTART).

Then, press the WRITE key. This prepares for the return of the latest state up to the N100.

2) Press the SEQ. RESTART switch.

The miscellaneous commands (S, T, M) are output to the PLC, then after the confirmation of the respective answer signals, the axes return to the point commanded last at the cutting feedrate.

3) After axes have reached the return point in the sequence return operation, the operation stops at that point independent of SINGLE BLOCK key setting. To resume the operation, the CYCLE START switch must be pressed.

7. Sequence Stop (Option)

This is the function to stop the program execution at a desired sequence while the automatic operation is carried out. Note that the sequence to be set must be the one already executed. The operation stops in the same manner as a single block stop. The operation can be resumed by simply pressing the CYCLE START switch.

The operation procedure is described below:

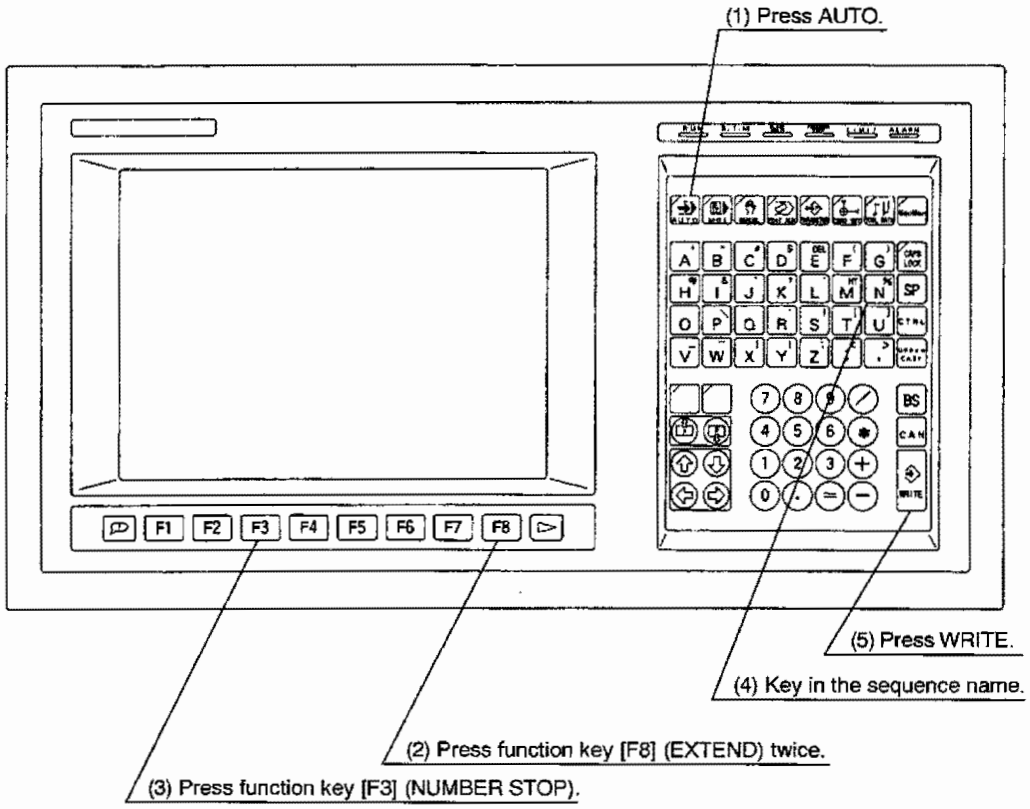


Fig. 5-5 Sequence Stop Operation

(1) Press the AUTO key.

(2) Press function key [F8] (EXTEND) twice.

Function name "NUMBER STOP" is displayed for function key [F3].

(3) Press function key [F3] (NUMBER STOP).

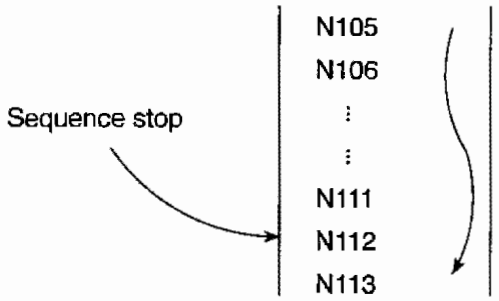
"NST" is displayed on the 21st line on the display screen.

(4) Key in the sequence name where the operation is to be halted on the keyboard.

(5) Press the WRITE key.

When the block of the specified sequence name is read during automatic operation, the blocks up to the sequence before the specified sequence are executed and the machine stops.

Example:



When N112 is entered following function key [F3] (NUMBER STOP) during the execution of N105 sequence, the machine stops after N112 execution and before N113 execution.

The sequence name, where the sequence stop is to be executed, can be changed by specifying another required sequence name. It is also possible to cancel the sequence stop by pressing the WRITE key directly following function key [F3] (NUMBER STOP) without keying in any sequence name data.

[Supplement]

1. The sequence stop setting is cleared by NC reset.
2. The sequence name entered for the step sequence name is handled as a character string and therefore, comparison as a number is not made.
3. Setting during the scheduled operation causes errors.
4. If the main program is selected correctly, setting is possible either before or during execution of the main program.
5. Even if the sequence name which has been already read is set, the program cannot be stopped at that sequence.

8. Single Block

When the SINGLE BLOCK switch on the machine operation panel is switched ON, the single block function is turned on and the program stops after executing the current program block.

There are two types of single blocks.

(1) Execute Single Block

This stops by the block accompanying axis movement execution, miscellaneous function operation or coordinate system setting. This does not stop by control statements such as CALL, GOTO, etc., or a macro call command, return command, and NOEX statement.

(2) Read Single Block

This stops at all blocks including the control statement.

(a) Execute single block is generally used, but for checking programmed operation, read single block is used.

(b) Read single block is switched on by setting 1 at bit 0 of NC optional parameter (bit) No. 2.

(c) Execute single block is determined by switch status after one-block execution.

The read single block is determined by switch status when one block is read.

(d) If the single block is switched on during automatic operation and single block OFF state, the block under execution is completed and the machine stops operation. If there is any buffered block at this stage, one block is executed every time the CYCLE START switch is pressed. If the buffer empties, a new block is read and executed.

(e) Read single block is used to stop the execution of a program by each control statement, IF, GOTO and VSET in the schedule program.

(f) During the program execution called in MDI mode, setting of the SINGLE BLOCK key is effective.

(g) For the details of the stopping manner during fixed cycle operation, refer to "FIXED CYCLES" in the Programming Manual.

(h) During the area machining mode, the axis stops at the completion of each motion when the single block is on.

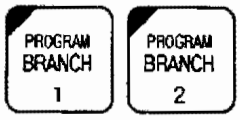
9. Optional Block Skip

When the BLOCK SKIP switch on the machine operation panel is switched ON, the block skip function is turned on and the blocks preceded by the slash code (/) are skipped.

For details, consult "OPTIONAL BLOCK SKIP" in the Programming Manual.

10. Program Branch

When the PROGRAM BRANCH switch on the machine operation panel is switched ON, the program branch function is turned on and the program branch commands in the program are executed.



Two PROGRAM BRANCH keys are provided and each of them turns on/off a program branch command respectively.

For details, refer to "SUBPROGRAM FUNCTIONS" in the Programming Manual.

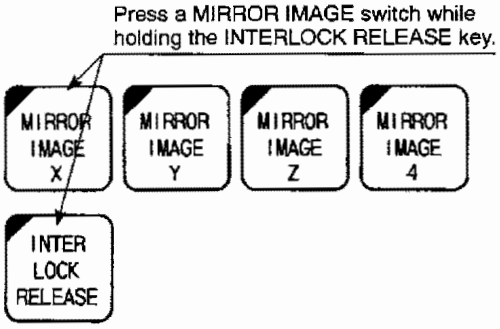
11. Optional Stop

When the OPTIONAL STOP switch on the machine operation panel is switched ON, the optional stop function is turned on and program execution is suspended at the block in which M01 is specified. Pressing the CYCLE START switch cancels the optional stop state and the program is restarted.

When the switch is off, M01 is neglected. In this case, the program is executed continuously without being stopped.

12. Mirror Image

When the MIRROR IMAGE switch on the machine operation panel is switched ON, and the mirror image function is turned on for the corresponding axis. For the axis for which the mirror image function is called, the sign of the coordinate values is reversed.



In addition, there is a programmable mirror image (G62) function, having the mutual relation described below.

Mirror Image by Switch	Mirror Image by G Code	Switch of the Coordinate Value Data Signs
OFF	OFF	Does not switch
OFF	ON	Switches
ON	OFF	Switches
ON	ON	Does not switch

The relation is established for each axis, independently.

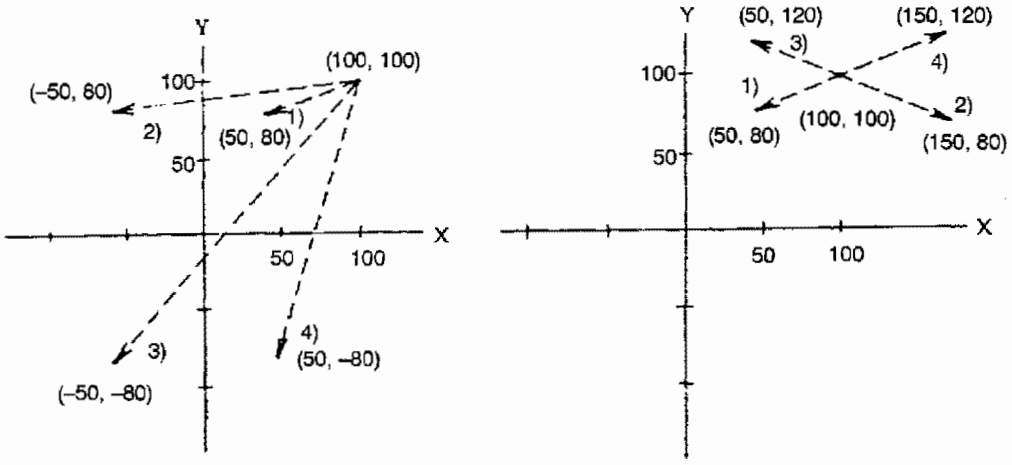
For details of the programmable mirror image function, refer to "Programmable Mirror Image (G62)" in the Programming Manual.

Example:

a) In absolute dimensioning
N80 G90 X100 Y100
N81 X50 Y80

MIRROR IMAGE
Setting changed
here

b) In incremental dimensioning
N80 G90 X100 Y100
N81 G91 X–50 Y–20



1) X,Y: Normal
2) X : Mirror image ON
Y : Normal
3) X,Y: Mirror image ON
4) X : Normal
Y : Mirror image ON

Fig. 5-6 Example of Mirror Image

When the mirror image is on, as shown in the example, the sign of the programmed command itself is switched, disregarding whether the selected dimensioning system is absolute or incremental.

[Supplement]

1. When the MIRROR IMAGE key is changed during automatic operation, the state will be changed over from the newly read block. The buffered commands are executed in the previously selected mirror image/normal state.

2. During the single block mode operation switching, the MIRROR IMAGE key from on to off or vice versa after the completion of a block of commands will change over the state from the next block.

   Example:

   N100
   N101
   N102
   Mirror image ON

   When, after the execution block N101 in the single block mode, start-up is done with mirror image ON, the operation starts from N102 in the mirror image ON state.

3. Be aware of the following related capabilities:

   The mirror image function is valid for the instruction G92 P__ which establishes a work coordinate system.

4. The mirror image function is valid for the instruction G51 P__ which indicates the coordinates of the center for scaling.

5. The mirror image function is invalid for the coordinate system shift G11 P__.

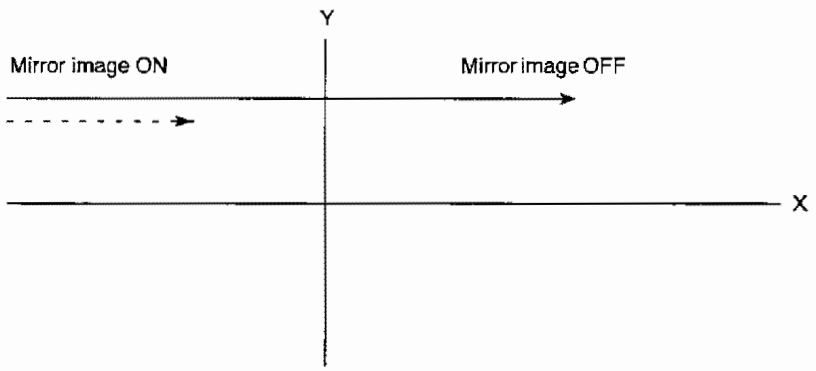
[Supplement] 6. While the mirror image function is active, G codes which indicate the direction of axis movement and I, J, and K commands are changed as needed.

Circular interpolation (G02,G03)

Cutter radius compensation (G41,G42), etc.

7. The positioning direction for 660 one-direction positioning is not changed even in a mirror image ON state.

Example:



8. On the display screen, the mirror image ON axes are identified by the "–" sign before the axis address of X, Y, Z, etc.

12-1. Mirror Image in the Work Coordinate System

Normally, when programming is executed in the local coordinate system, the mirror image function is effective in the local coordinate system. However, it is also possible to activate the mirror image function in the work coordinate system currently selected by changing parameter data.

(1) Parameter Setting

Whether the mirror image function is activated in the local coordinate system or in the work coordinate system can be set at the following parameter:

NC Optional Parameter (bit) No. 34, bit 2	
0	The mirror image function is activated in the local coordinate system.
1	The mirror image function is activated in the work coordinate system.

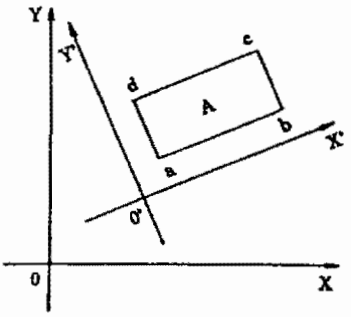
CAUTION

: (1) The initial setting is "0".

(2) When the setting has been changed, press function key [F7] (BACKUP). After the completion of backup operation, turn off the power to the NC and turn it back on again. The new setting does not become effective only by changing the setting.

(2) Comparison of Mirror Image Between in the Local Coordinate System and in the Work Coordinate System

In the figure to the right, the local coordinate system X'-Y' is set on the X-Y plane of the work coordinate system X-Y, and area A is machined. Under such conditions, the relation of mirror image between in the local coordinate system and in the work coordinate system is as explained below.



(a) In the G90 (absolute command) mode

1) Mirror image in the local coordinate system

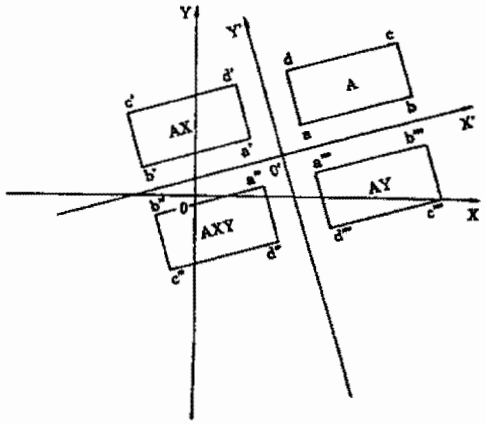


Fig. 5-7 Mirror Image in the Local Coordinate System (G90 Mode)

2) Mirror image in the work coordinate system

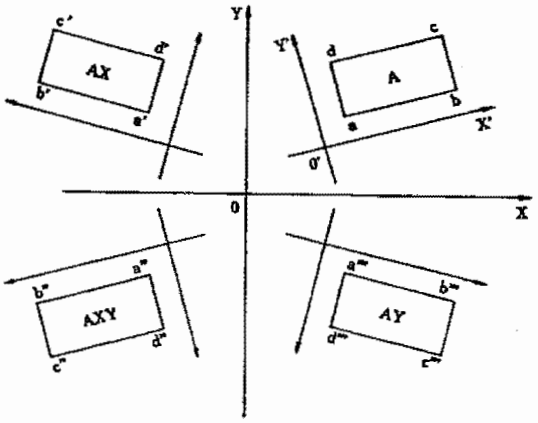


Fig. 5-8 Mirror Image in the Work Coordinate System (G90 Mode)

(b) In the G91 (incremental command) mode

1) Mirror image in the local coordinate system

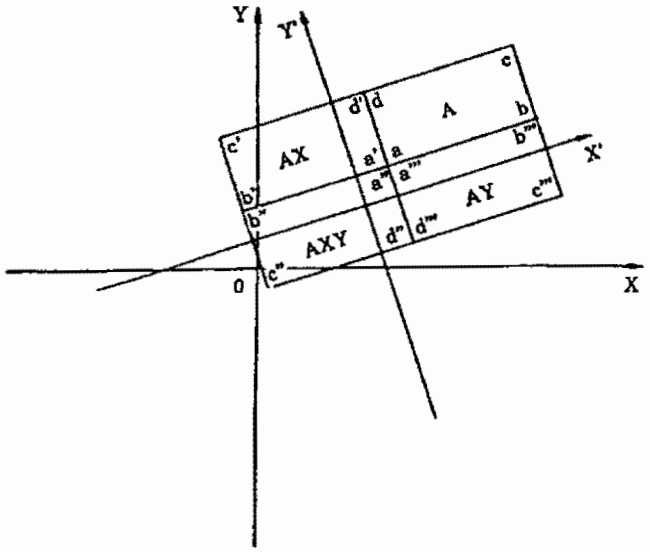


Fig. 5-9 Mirror Image in the Local Coordinate System (G91 Mode)

2) Mirror image in the work coordinate system

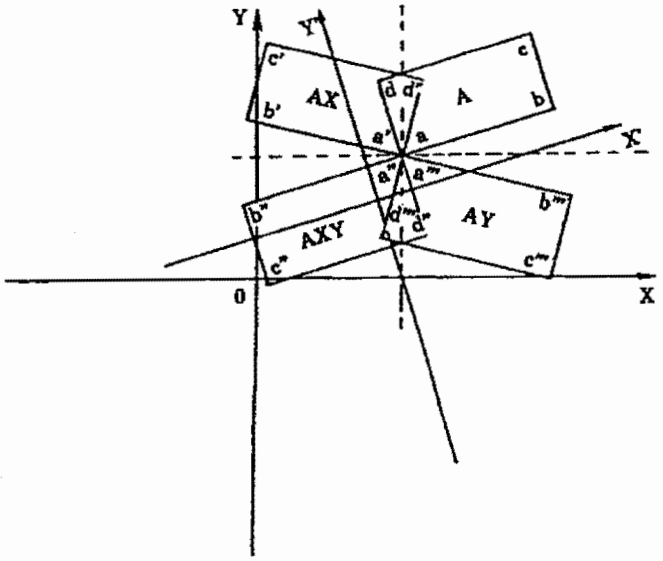


Fig. 5-10 Mirror Image in the Work Coordinate System (G91 Mode)

In the G91 mode, if no rotational elements are included in the local coordinate system and the coordinate system is shifted only parallel, the mirror image function has the same effect in the local and work coordinate systems.

Explanation for the figures:

AX ..... X-axis mirror image is turned on for machining area A.

AY ..... Y-axis mirror image is turned on for machining area A.

AXY ..... X-axis mirror image and Y-axis mirror image are turned on for machining area A.

(Machining order : a → b→ c→ d)

13. Override

The override function changes the feedrate or the spindle speed during machine operation within a certain range. The function includes the following:

- Rapid feedrate override
- Cutting feedrate override
- Spindle speed override

13-1. Feedrate Override

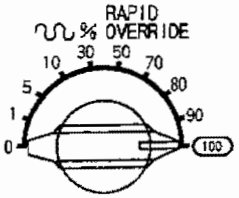
Programmed, manually input, or dial-set feedrates can be changed during operation.

13-1-1. Rapid Feedrate Override

This is effective during manual rapid feed operation and programmed rapid feed mode (G00, G60, etc.).

Actual rapid feedrate is "rapid feedrate" × "override value".

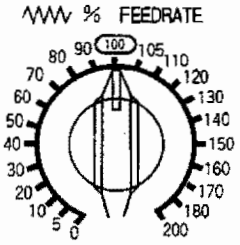
When 100% is selected, the axes move at the rapid feedrate determined by the machine specification.



13-1-2. Cutting Feedrate Override

This is effective for the programmed cutting feedrates in G01, G02, G03 and other modes.

Actual cutting feedrate is "F command value" × "override value".



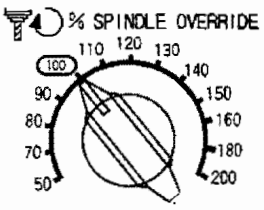
[Supplement]

1. Override rotary switch setting is ignored in tapping cycles called by G74 and G84, or in synchronized tapping cycles called by G274 and G284.
2. After the M136 (cutting feedrate override ineffective command) execution, the override rate is fixed at 100% irrespective of the rotary switch setting until M137 (M136 cancel command) is commanded and executed.
3. The axis does not move at a 0% override switch setting.

13-2. Spindle Speed Override

Spindle speed can be changed while the spindle is rotating.

The NC operates according to the spindle speed command given from the PLC. The actual spindle speed is displayed on the screen.



14. Manual Intervention During Automatic Operation and Restart

Manual intervention refers to the function in which manual operation is performed during AUTO or MDI mode operations.

(1) During the AUTO or MDI mode operation, press either the SLIDE HOLD switch or the SINGLE BLOCK switch to stop the cycle.

(2) Press the MANUAL INT ON switch.

The control is now placed in the manual intervention mode.

(3) Carry out necessary manual operations.

- Manual axis cutting feed
- Manual axis rapid feed
- Manual axis feed by pulse handle
- Spindle rotation
- Tool change, etc.

After manual axis feed, that distance is displayed on the display screen - 2nd page of ACT POSIT pages:

[Display screen]

MANUAL SHIFT ACTUL ..... Manually shifted amount in the present manual intervention operation

MANUAL SHIFT TOTAL ..... A total of manually shifted amounts until the present manual intervention operation

(4) Before restarting the sequence operation, restore the miscellaneous functions to the previous conditions, and locate the axes near the position where the intervention has been made using manual cutting feed or pulse handle. Then, press the SEQ. RESTART switch. With this, the axes return to the original position. The data displayed at MANUAL SHIFT ACTUL will become zero at the same time.

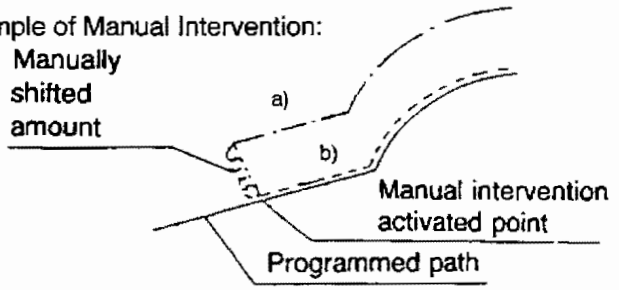
⚠CAUTION

: In this return operation, the axes are fed at a rapid feedrate. Therefore, it is necessary to confirm that the return motion will not cause interference between the spindle or the cutting tool with the workpiece or the fixture on the table.

If the SEQ. RESTART switch is not pressed, the manually fed amount is not zeroed.

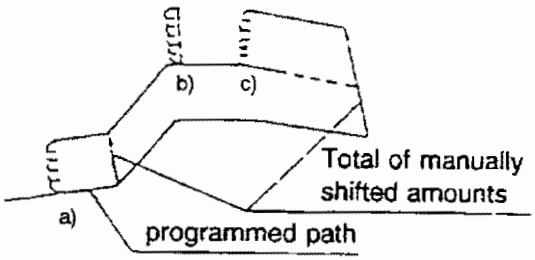
(5) Pressing the MANUAL INT OFF switch or SEQ. RESTART switch automatically cancels the manual intervention mode.

Example of Manual Intervention:



a) —— · —— · ——
SEQ.RESTART not pressed after manual axis feed

b) - - - - - - - - - - -
SEQ.RESTART pressed after manual axis feed



a) SEQ.RESTART not pressed after manual axis feed
b) SEQ.RESTART pressed after manual axis feed
c) SEQ.RESTART not pressed after manual axis feed

[Supplement] During manual intervention mode, automatic return of miscellaneous functions is not made. This permits the change of spindle speed or tools during this mode.

CAUTION

: Even when tool offset and/or cutter radius compensation data has been changed for a newly set tool (manual change), this data does not become effective at the point when the return to the originally located position is completed.

15. Inserting Pulse Handle Operations

On a machining center, it is sometimes required to feed the Z-axis manually while positioning of the X- and Y-axis is made as programmed. Or when machining a cast workpiece on which stock removal amount varies greatly, it is necessary to adjust the depth of cut manually. In such operations, axis motion controlled by the pulse handle can be inserted to the programmed axis movements.

Operation to insert axis movement by the pulse handle is explained below:

(1) Press the PULSE HANDLE SHIFT key on the machine operation panel to turn it on.

(2) Set the AXIS SELECT selector on the pulse handle operation panel to the axis which is to be moved. Also select the multiplication factor.

(3) Feed the axis by turning the pulse handle.

The amount fed by the pulse handle is shown in the MANUAL SHIFT TOTAL line. Automatic return is not made for the amount fed by the pulse handle. If it is then necessary to return the axis by turning the pulse handle while observing the MANUAL SHIFT TOTAL data on the display screen.

[Supplement]

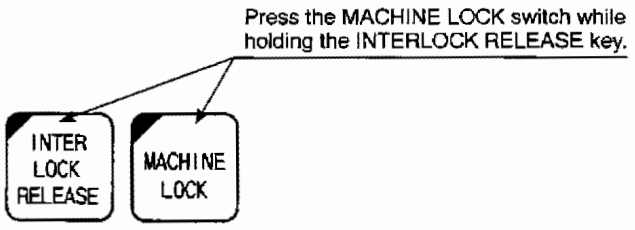
1. The MANUAL SHIFT ACTUL data is cleared ("0") when the MANUAL INT. ON key is pressed.
2. The MANUAL SHIFT TOTAL data is cleared ("0") when the POWER ON switch is pressed.

   It is also cleared when the RESET switch is pressed when parametric data (bit 2 of NC optional parameter (bit) No. 4) is so set.
3. Axis travel amount using the pulse handle is added to the axis position data which is used for judging travel end.
4. While the PULSE HANDLE SHIFT key is off, the pulse handle is inoperative.

16. Lock Functions

16-1. Machine Lock

When the MACHINE LOCK switch on the machine operation is switched ON, the machine lock function is turned on. In this state, actual position values on the screen are updated as the program is executed while the machine is stopped.



[Supplement] When the machine lock function is turned on or off, the NC is reset.

16-2. Cancellation of Axis Command

When the AXIS COM. CANCEL switch on the machine operation panel is switched ON, the axis command cancel function is turned on and the commands of the axis set for the corresponding parameter are ignored to disable movement of that axis. The axis for which the commands are ignored is set for NC optional parameter (bit) No. 7.

[Supplement] With the home position return command given by an external signal, the axis set to be ignored is also moved according to the command.

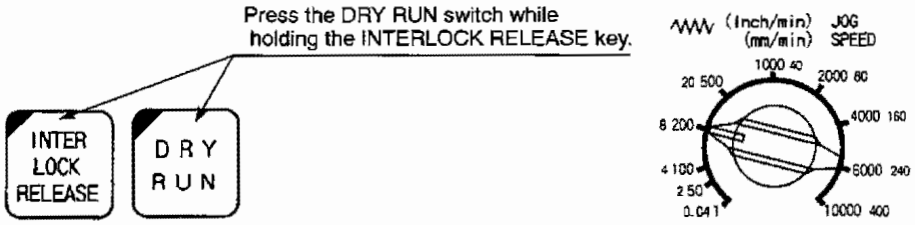
16-3. S, T, and M Function Lock

When the STM LOCK switch on the machine operation panel is switched ON, the STM lock function is turned on. In this state, the miscellaneous function operation specified by the S, T, and M codes is not executed, and only axis feed is executed.

[Supplement]
1. Override or dry function is also effective as in normal cutting operation.
2. The axis specified by the axis command cancel function may be moved by manual or manual intervention operation.
3. Spindle operation control is not executed.
4. With the home position return command given by an external signal, the axis set to be ignored is also moved according to the command.
5. While the STM lock function is active, manual operation of S, T and M functions is possible by manual intervention operation.

17. Dry Run

Dry run is a function for running the machine at the feedrate set by the cutting feedrate rotary switch on the machine operation panel, while disregarding the programmed feedrates in G01, G02, G03 and other similar modes.



In the rapid feed (G00) mode, the data set at bit 2 of NC optional parameter (bit) No. 2 determines whether the dry run is effective or not.

Switching the DRY RUN switch on/off is possible even while commands in a block are being executed. When changed over, the machine is immediately set to the state selected.

S, T and M functions are executed as usual when the DRY RUN switch is on.

The dry run function is effective while the machine lock function is active.

[Supplement] Be careful when activating this dry run function since it is effective during a G74 and G84 tapping cycle, or during the G31 skip function.

18. Library Program Registration

This is the function necessary for executing such items as a subprogram and a G code macro through the MDI operation or for operating the program which contains subprogram call commands in the large capacity mode operation after specifying the S option. That is, the registration of subprograms is possible with this library program registration function. Such a registration is usually made when the PROGRAM SELECT function key is pressed.

The operating procedure is shown below:

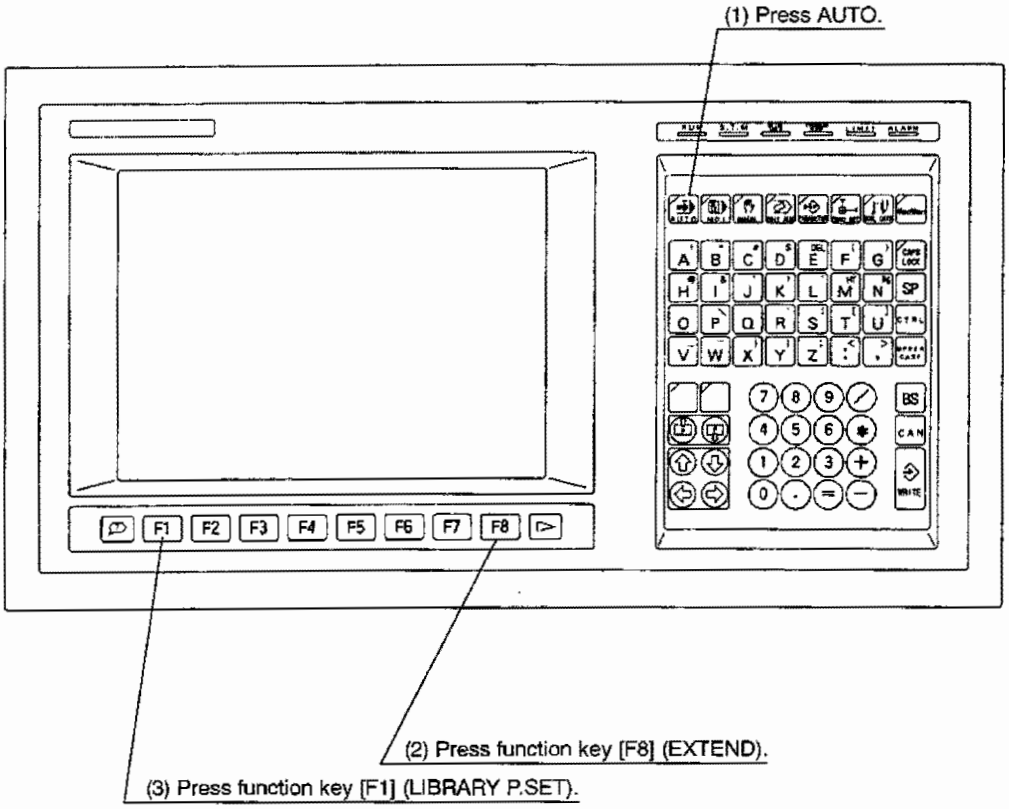


Fig. 5-11 Registering Library Programs

(1) Press the AUTO key.

(2) Press function key [F8] (EXTEND).

The function messages will change as shown above. (The message "LIBRARY P.SET" appears above F1 .)

(3) Press function key [F1] (LIBRARY P.SET).

"LP" is displayed on the 21st line on the display screen.

Operating procedure for library program registration:

(1) Library Program Directory Display

Press function key [F1] (LIBRARY P.SET), key in "LP ␣ " and then press the WRITE key.

The screen displays the library program directory and the remaining capacity in the number of bytes.

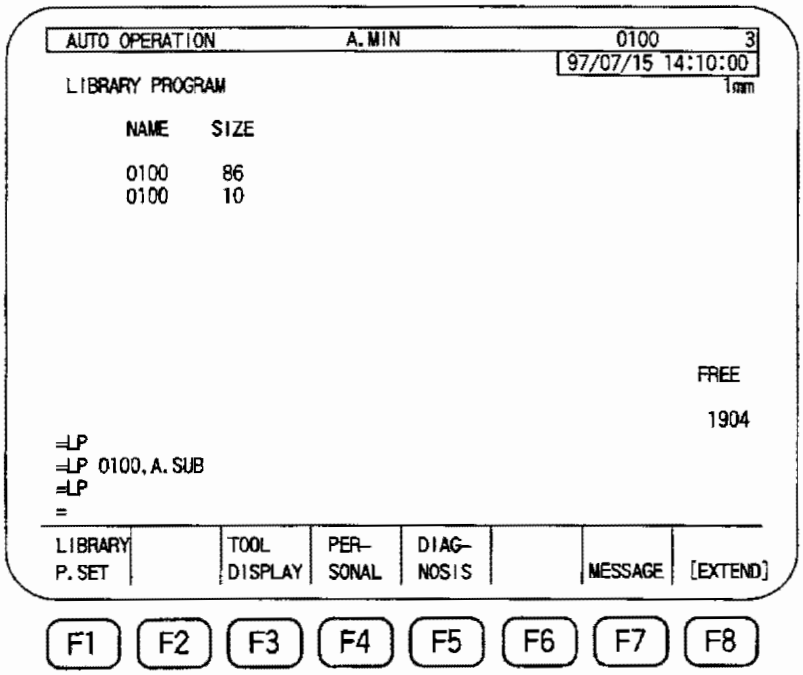


Fig. 5-12 Library Program Directory Display

When library program directory is displayed on two pages, use the BS or WRITE key to change the display screen.

(2) Library Program Registration

Press function key [F1] (LIBRARY P.SET) and key in program name and file name as follows:

LP ␣ program name, file name [WRITE]

The library program can now be registered.

When a file name is omitted, subprograms in all the SSB files are registered when the entered program name is "O∗∗∗∗".

When a program name is omitted, all the subprograms in the file specified following the comma (,) are registered as library programs.

Up to 65 subprograms can be registered.

Note that a main program cannot be registered as a library program.

(3) Deleting Library Program

Press function key [F1] (LIBRARY P.SET) and key in the program name as follows:

LP ␣ program name;C [WRITE]

With the operation above, the programs registered as library programs are deleted.

(4) Initializing Library Program

Press function key [F1] (LIBRARY P.SET) and key in as follows:

LP␣ ;I [WRITE]

With the operation above, the programs registered as library programs are all deleted.

(5) Specifying Buffer Size of Library Program

Press function key [F1] (LIBRARY P.SET) and key in as follows:

LP␣ n;I [WRITE]

With the operation above, the buffer size of "n" bytes for registering the library programs is ensured.

Note that when the buffer size specification is changed, the library program registration state and the program selection state are all cleared.

To ensure the NC program registration area for operation without using library programs as much as practicable, zero the library program registration by the operation below.

Press function key [F1] (LIBRARY P. SET) and key in as follows:

LP␣ 0;I [WRITE]

The library programs registered in the operations above, can be accessed by the required MDI mode operation. They can also be referenced in the S option mode or the equivalent operation mode (DNC, for instance).

(6) The subprograms in the file which have an extension LIB are automatically registered as library programs when power supply to the control is turned on. Therefore, G code macro or other subprograms which are frequently used are recommended to be stored in the LIB file. This permits them to be called any time as conventional G codes.

NOTICE

: (1) A library program is distinguished only by its program name. Therefore, it is impossible to register more than one library program which have the same name.

If the subprogram in the user program has the same name as the library program, the library program is given priority when such a subprogram is intended to be called.

(2) Turn off and on the power alter adding a new library program.

19. Operation End Light (Option)

The operation end light goes on when the following conditions have been satisfied. Note that the setting for the related machine user parameter must be set "effective".

Mode	Condition
Automatic operation Single block off	When program is completed (1) schedule program after executing END in a schedule program. (2) main program after executing M00, M01, M02 or M30 in a main program.

Restarting will extinguish the operation end light.

When the alarm occurs, resetting the CNC unit will extinguish the operation end light.

20. Operation End Buzzer (Option)

Operates in the same conditions as the operation end light. The operation end buzzer sounds when these conditions have been satisfied. Note that the setting for the related machine user parameter must be set "effective".

The operation end buzzer sounds for the length of time set for machine user parameter, "Buzzer, 7. Operation end buzzer timer".

Restarting will cut off the operation end buzzer.

21. Error Light (Option)

The ERROR lamp is turned on at the occurrence of an alarm. Note that the setting for the related machine user parameter must be set "effective".

The lamp is turned off when the NC is reset.

22. Auto Power Off (Option)

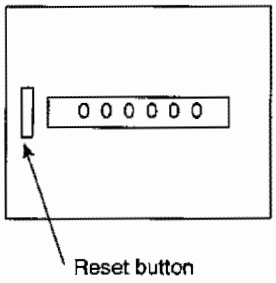
The power is shut down when the following conditions are satisfied. Note that the setting for the related machine user parameter must be set "effective".

Mode	Condition
Automatic operation Single block off	(1) Schedule program after executing END in a schedule program (2) Main program after executing M02 or M30 in a main program

When this function is used and there are blocks changing/setting parameters on the program, it is recommended to make the backup function effective with M02/M03 (Set "1" at NC optional parameter (bit) No. 33, bit 1).

23. Work Counter (Option)

Count data is incremented when the machining completion signal (M02 or M30) is executed. The counter has a six-digit capacity (0 to 999999). It is provided with the reset function.

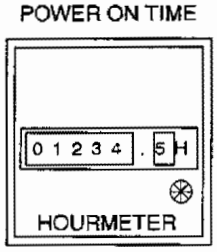


24. Hour Meter (Option)

24-1. POWER ON TIME Hour Meter

The hour meter accumulates length of time in which operation power has been turned on.

Hour meter range: 0 to 99999.9 hours

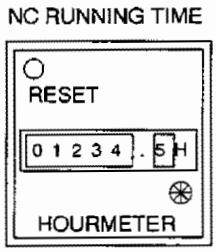


24-2. NC RUNNING TIME Hour Meter

The hour meter accumulates length of time in which the RUN lamp in the NC operation panel status indicating lamps has been lit.

Hour meter range: 0 to 9999.9 hours

The hour meter is provided with the RESET button.

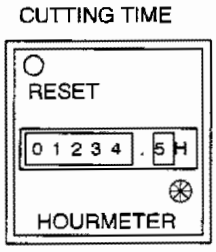


24-3. CUTTING TIME Hour Meter

The hour meter accumulates length of time in which cutting has been conducted (G01, G02 or G03 mode) in the automatic or MDI operations.

Hour meter range: 0 to 9999.9 hours

The hour meter is provided with the RESET button.

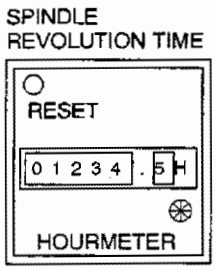


24-4. SPINDLE REVOLUTION TIME Hour Meter

The hour meter accumulates length of time in which the spindle has been rotating (CW or CCW) disregarding the operation mode.

Hour meter range: 0 to 9999.9 hours

The hour meter is provided with the RESET button.



[Supplement] When hour meters are selected, 24-1. "POWER ON TIME Hour Meter" and 24-2. "NC RUNNING TIME Hour Meter" are usually equipped.